SYNTEC TECHNOLOGY CO.,LTD.

6D/6-H/6-E/21/21-H/21-E/210-E/22/220 Series_Serial Bus Milling Machine Operation Manual

2020 Dec 08, 07:44

1 **Applicable Models**

This milling machine operation manual is applicable for Syntec serial controllers including:

| Communication Command	Model
Mechatrolink II	6MD, 6MD-H 21MA, 21MA-H
Mechatrolink III	6MD-H、6MA-E, 6MB-E 21MA-H、21MA-E 210MA-E, 210MB-E, 210MA-E5, 210MB-E5 22MA 220MA, 220MB, 220MA-5, 220MB-5
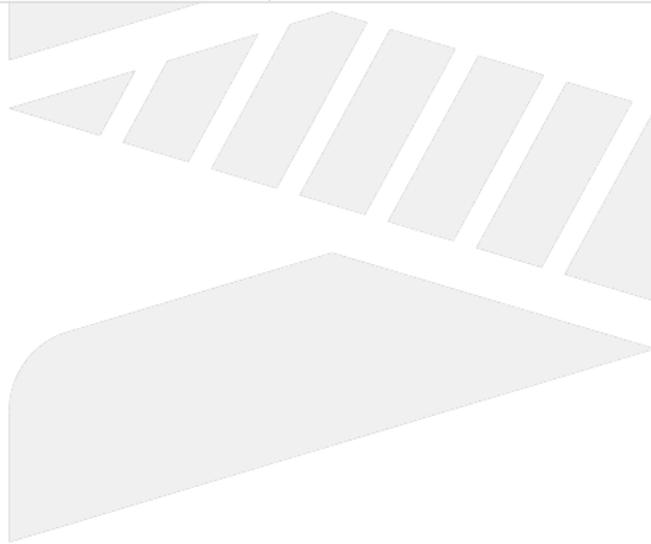
# 2 **Fenubar and System Introduction**

2.1 Main Screen Introduction
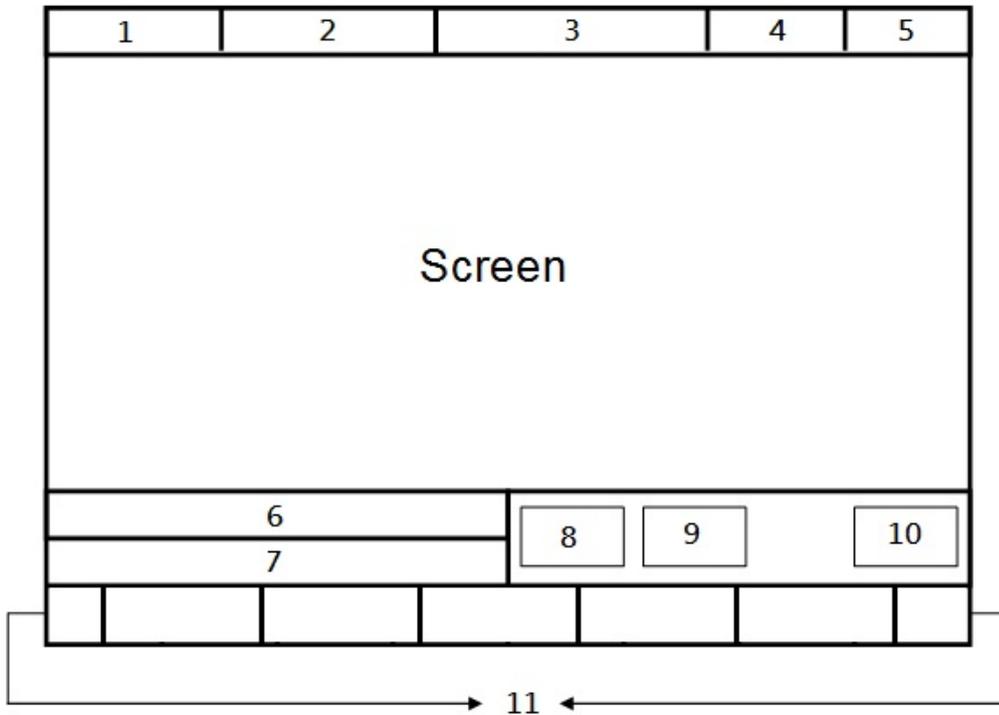
- Interface Explanation:
  a. Current workpiece coordinate
  b. NC File Name and Line Number
  c. Screen Title
  d. Date
  e. Time
  f. Data Input
  g. Hint
  h. State
  i. Mode
  j. Alarm
  k. Fenubar

2.2 Tree Diagram of Fenubar

8-Key System

Serial-8 key-M

- **1.Coord.**
  - 1.Switch Coordinate
  - 2.Half Coordinate
  - 3.Zero Rel. Coord.
  - 4.Zero All Rel. Coord.
- **2.Program**
  - 1.Execute
  - 2.Delete Line
  - 3.Search/Replace
  - 4.CanCycle
  - 5.Block Copy
  - 6.Teach
  - 7.Simulation
  - 8.File Manager
  - 1.Goto Line
- **3.Offset/Setting**
  - 1.Workpiece Coord.
  - 2.Tool Set
  - 3.Tool Tip Measure
  - 4.User Param.
  - 6.Tilt Work Plane Teach
- **4.Monitor**
  - 1.Edit
  - 2.Simulation Switch
  - 3.MDI Input
  - 4.Param. Set
  - 5.Tool Wear Set
  - 6.Start MPG Shift
  - 7.Work Record
  - 8.Clear Accum. Cycle Time
  - 1.Graph Adjust
  - 4.Gantry Calibration
- **5.Maintain**
  - 1.Alarm
  - 2.Set Kernel Server
  - 3.Fast Diag.
  - 4.PLC Param.
  - 5.Sys. Setting
  - 6.Backup Sys.
  - 8.Use Time Setting
  - 1.Login
  - 2.Logout
  - 3.Forget Password
  - 4.Authority Mgt.
  - 5.Reset Password
  - 8.About
- **6~8.Defined By "My Favor"**

5-Key System

Serial-5 key-M

- **1.Coord.**
  - 1.Switch Coordinate
  - 2.Half Coordinate
  - 3.Zero Rel. Coord.
  - 4.Zero All Rel. Coord.
- **2.Program**
  - 1.Execute
  - 2.Delete Line
  - 3.Search/Replace
  - 4.File Manager
  - 5.Simulation
  - 1.CanCycle
  - 2.Block Copy
  - 3.Teach
  - 4.Goto Line
- **3.Offset/Setting**
  - 1.Workpiece Coord.
  - 2.Tool Set
  - 3.Tool Tip Measure
  - 4.User Param.
- **4.Monitor**
  - 1.Edit
  - 2.Simulation Switch
  - 3.MDI Input
  - 4.Param. Set
  - 5.Tool Wear Set
  - 1.Start MPG Shift
  - 2.Work Record
  - 3.Clear Accum. Cycle Time
  - 4.Graph Adjust
  - 2.Gantry Calibration
- **5.Maintain**
  - 1.Alarm
  - 2.Set Kernel Server
  - 3.Fast Diag.
  - 4.PLC Param.
  - 5.Sys. Setting
  - 1.Backup Sys.
  - 3.Use Time Setting
  - 1.Login
  - 2.Logout
  - 3.Forget Password
  - 4.Authority Mgt.
  - 5.Reset Password
  - 5.About

10+8 key system

Serial-10+8Key-M

1. Information
1.Switch Coordinate | 2.Half Coordinate | 3.Zero Coord. | 4.WorkPiece Coord. | 5.Tool Set | 6.Pending Alarm | 7.Login | 8.Logout | 9.Work Record | 10.File Manager | 11.Param.

2.Program
1.Execute | 2.Delete Line | 3.Search/Relapce | 4.CanCycle | 5.Block Copy | 6.Teach | 7.— | 8.Goto Line | 9.Simulation | 10.File Manger

3.Offest / Setting
1.WorkPiece Coord. | 2.Tool Set. | 3.Tool Tip Measure | 4.User Param | 6.Tilt Work Plane Teach | 10.Next Group Page

4.Monitor
1.Edit | 2.Simulation Switch | 3.MDI Input | 4.Param Set | 5.Tool Wear Set | 6.Start MPG Shift | 7.Graph Adjust | 8.File Manager | 9.Work Record | 10.Clear Accum. Cycle Time

5.Maintain
1.Alarm | 2.Set Kernel Server | 3.Fast Diag. | 4.PLC Param. | 5.Sys Setting | 6.Backup Sys. | 7.Login | 8.Logout | 9.Authority Mgt. | 10.User Time Setting | 11.Reset Password

6.App

7.Service
8.Online Service | 9.Syntec EagleEye

8.System
1.PLC Status | 2.Diag. | 3.Param. | 4.Sys Admin | 10.About

2.3 Coordinate

G54		N0 L1	Coord.	2020/1/8	17:01:48	DEFAULT

Machine

X	0.000
Z	0.000
Y	0.000

Relative
X	0.000
Z	0.000
Y	0.000

Absolute
X	0.000
Z	0.000
Y	0.000

Dist. To Go
X	0.000
Z	0.000
Y	0.000

F	1.0 mm/min 100 %	S	1000 RPM 100 %
	0.0 mm/min (Actual)		1000 RPM (Actual)

Run Time 0 : 0 : 0   Part Count 0   T 0000

●Ready | Auto | Alarm

- Path
    - F1 Coordinate
    - G1 Information (10+8 key series controller)

- Explanation
    - To switch the coordinate system displayed on the screen.
    - To display frequent used information for machining.
    - Could switch to this screen instantly by hotkey [POS] .

2.3.1 Screen Description

Coordinate Display

- This screen can display 4 coordinate systems simultaneously.
- You can switch the coordinate system currently displayed on the top left area through the "Switch Coordinate" button.

F (Feedrate)

- Displays the feedrate defined by user.
- Displays the actual feedrate of the machine.
- Displays the override of the feedrate defined by user.

S (Spindle Speed)

- Displays the spindle speed defined by user.
- Displays the actual spindle speed of the machine.
- Displays the override of the spindle speed defined by user.

Run Time

- Displays the execution time of the NC program while machining.

Part Count

- Displays how many times a NC program is executed already.
- Cooperating with Pr3804, every time the specified M code is executed, part count will automatically plus one, and timer for single part will return to zero.
- When the required quantity is reached, the machine would switch to "B Stop" state automatically.

T (Tool Number)

- Displays the tool number and the tool compensation number currently used.

2.3.2 Switch Coordinate

- Path
    - 5 key, 8key：F1 Coordinate → F1 Switch Coordinate
    - 10+8 key：G1 Information → F1 Switch Coordinate
- Explanation
    - This key can switch the displaying order of the coordinate systems on the screen.

2.3.3 Half Coordinate

- Path
    - 5 key, 8 key：F1 Coordinate → F2 Half Coordinate
    - 10+8 key：G1 Information → F2 Half Coordinate
- Explanation
    - Divides the value of relative coordinate by 2.

- With "F3 Zero Rel. Coord.", this function can instantly calculate the coordinate of the middle point between two points.

- Operation Method
  - a. Enter the axis name which you want to calculate the coordinates, and then press this key.
  - b. Example:
    - i. The current relative coordinate value of X axis is 10.000.
    - ii. Enter "X", and then press "F2 Half Coordinate".
    - iii. The relative coordinate value of X axis will then become 5.000.

2.3.4 Zero Rel. Coord.

- Path
  - 5 key, 8 key：F1 Coordinate → F3 Zero Rel. Coord.
  - 10+8 key：G1 Information → F3 Zero Coord.

- Explanation
  - Sets the relative coordinates value to zero.

- Operation Method
  - Enter the axis name which you want to calculate the coordinates, and then press this key.

- Example
  - a. The relative coordinate value of X axis is 10.000.
  - b. Enter "X" and press "F3 Zero Rel. Coord."
  - c. The relative coordinate value of X axis will then become 0.000.

2.3.5 Zero All Rel. Coord.

- Path
  - 5 key, 8key：F1 Coordinate → F4 Zero All Rel. Coord.
  - 10+8 key (milling machine)：G3 Offset/Setting → F1 WorkPiece Coord. → F7 Zero All Rel. Coord.
  - 10+8 key (lathe)：G1 Information → F4 Zero All Rel. Coord.

- Explanation
  - Set the relative coordinates of all axes to zero.

- Example
  - a. The relative coordinate values of X and Y axis are 10.000 and 5.000 respectively.
  - b. Press "Zero All Rel. Coord."
  - c. The relative coordinate values of X and Y axis will then become 0.000.

2.4 Program

G54		ATEST-161 N0 L1		Program		2013/7/2		15:14:50

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

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

	●Ready	Auto		Alarm
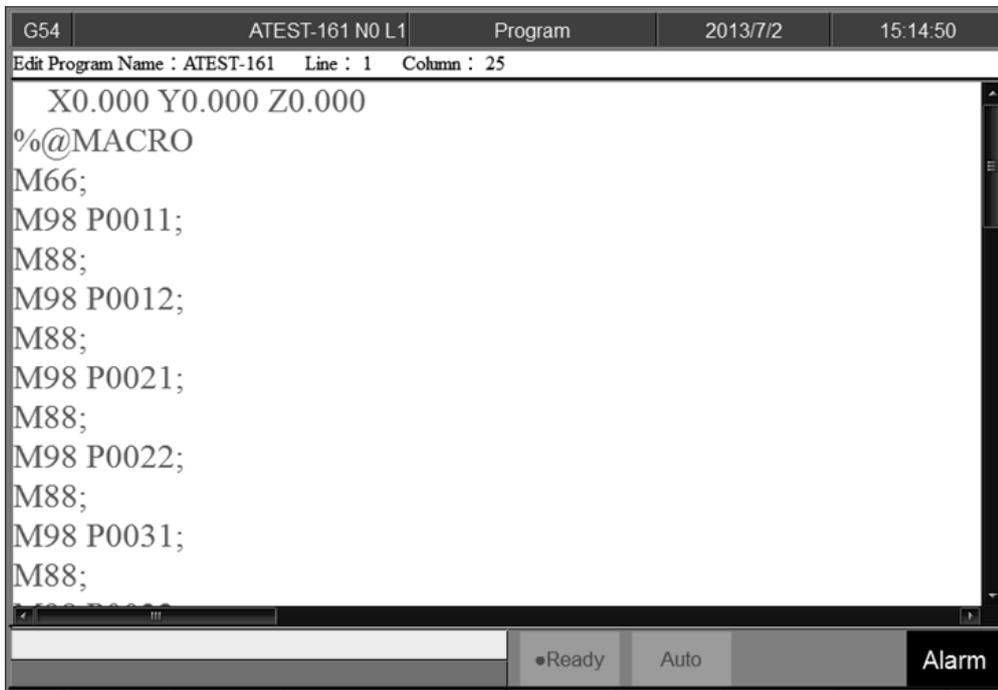
- Path
    - 5 key, 8 key：F2 Program
    - 10+8 key：G2 Program

- Explanation
    - Syntec controller supports Program Edit function, users are able to edit programs on this screen.

- Example
    a. Use arrow keys【↑】【↓】【←】【→】to move the cursor.
    b. Use【Page Up】【Page Down】to switch between pages.
    c. Use【Home】【End】to quickly move to the beginning or the end of the line.
    d. Use hotkey【Prog/File】to quickly switch the screens between "Program" and "File Manager".

- Compatibility changes
    - 2 new auto-save functions are provided for WinCE system. (valid version- 10.116.32, 10.117.32)
        - Auto-save every 30 seconds: the system will save the NC file under editing every 30 seconds automatically.
        - RESET save: the system will save the NC file under editing automatically when pressed "Reset".

2.4.1 Execute

- Path
    - 5 key, 8 key：F2 Program → F1 Execute
    - 10+8 key：G2 Program → F1 Execute

- Explanation
    - This key can assign the program currently under editing as the program to be executed, and switch the screen to "Monitor" page.

- Attention

- This key will be disabled during machining (busy state).

2.4.2 Delete Line

- Path
  - 5 key, 8 key：F2 Program → F2 Delete Line
  - 10+8 key：G2 Program → F2 Delete Line

- Explanation
  - Deletes the line which the cursor is located.

2.4.3 Search/Replace

G54		TEST N0 L1	Program	2020/1/10	15:30:41	Admin

EditProgName：test　　　Line: 1　　　Column: 15

G01 X100. F2000

Replace

Search String：

Replace String：

●Ready　　Auto　　Alarm

- Path
  - 5 key, 8 key：F2 Program → F3 Search/Replace
  - 10+8 key：G2 Program → F3 Search/Replace

- Explanation
  - This function is used to search or replace strings in the editing program.
  - Pressing "F3 Search/Replace" will pop up a dialogue window for setting the search string/replace string.

- Operation Method
  - Use【ENTER】to switch the cursor between "Search String" and "Replace String".

Find Next

- Path
  - 5 key, 8 key：F2 Program → F3 Search/Replace → F1 Find Next
  - 10+8 key：G2 Program → F3 Search/Replace → F1 Find Next

- Explanation

- In the current editing program, start searching for the same as a string in "Search String" after the cursor line.
- The cursor and the screen will move to the next string and highlights it.

Replace

- Path
    - 5 key, 8 key：F2 Program → F3 Search/Replace → F2 Replace
    - 10+8 key：G2 Program → F3 Search/Replace → F2 Replace

- Explanation
    - In the current editing program, start searching for the same as a string in "Search String" after the cursor line, and then replace it with a string in "Replace String".

- Operation Method
    a. If there is no highlighted string on the screen, the system will run "F1 Find Next" instead.
    b. Press "F2 Replace" again will replace the highlighted string with a new string and then run "F1 Find Next".
    c. If want to skip the current highlighted string, press "F1 Find Next" to move to the next one.

Replace All

- Path
    - 5 key, 8 key：F2 Program → F3 Search/Replace → F3 Replace All
    - 10+8 key：G2 Program → F3 Search/Replace → F3 Replace All

- Explanation
    - In the current editing program, replace all strings which are the same as strings in "Search String" with a new string and move the cursor to the final line of the program.

2.4.4 CanCycle
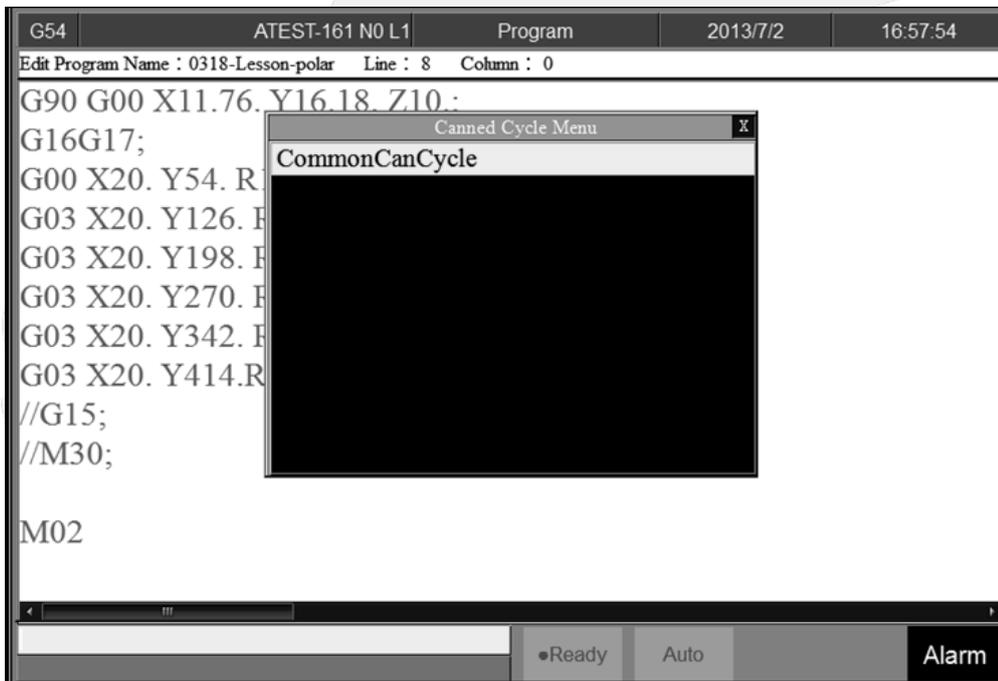
- Path
    - 5 key：F2 Program → Next -> F1 CanCycle
    - 8 key：F2 Program → F4 CanCycle
    - 10+8 key：G2 Program → F4 CanCycle

- Explanation
    - When editing the program, since so many G codes are provided and the definitions of arguments are different, thus CanCycle is able to help users to edit the G codes more easily.

Insert Cycle

- Path
    - 5 key：F2 Program → Next → F1 CanCycle → F1 Insert Cycle
    - 8 key：F2 Program → F4 CanCycle → F1 Insert Cycle
    - 10+8 key：G2 Program → F4 CanCycle → F1 Insert Cycle

- Operation Method
    a. In the current editing program, move the cursor to the line which you want to add cycle and press "F1 Insert Cycle". Follow the instructions to select the inserting cycle and set the required arguments.
    b. Press "OK" after editing. The edited cycle will be added to the line below the cursor.

Edit Cycle

- Path
    - 5 key：F2 Program → Next → F3 CanCycle → F2 Edit Cycle
    - 8 key：F2 Program → F4 CanCycle → F2 Edit Cycle
    - 10+8 key：G2 Program → F4 Cancycle → F2 Edit Cycle

- Explanation
    - Edit the cycle where the cursor is located.

- Operation Method
    - Move the cursor to the cycle which you want to modify and press "F2 Edit Cycle", the cycle editing window will then be displayed. Users can modify contents and press OK after finish editing.

2.4.5 Block Copy

| G54 | | N0 L1 | Program | 2020/4/29 | 15:48:32 | DEFAULT |

EditProgName： | Line: 1 | Column: 0

●Ready　　Auto　　Alarm
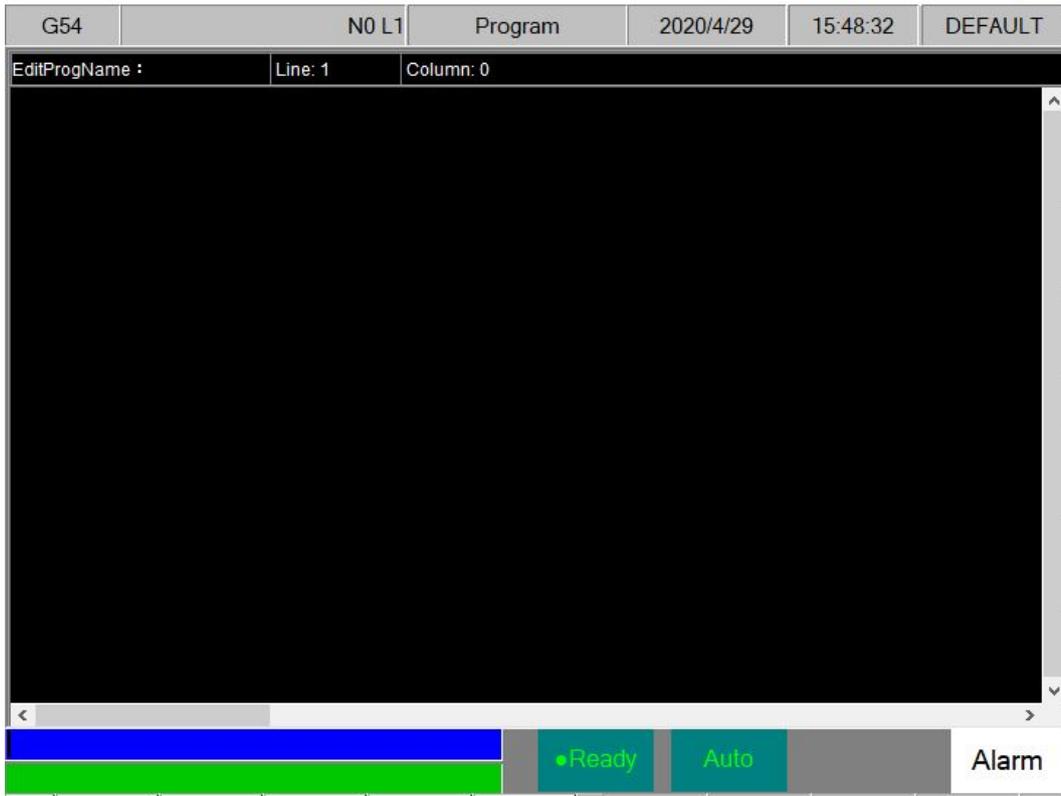
- Path
    - 5 key：F2 Program → Next → F2 Block Copy
    - 8 key：F2 Program → F5 Block Copy
    - 10+8 key：G2 Program → F5 Block Copy

- Explanation
    - This button can be used to select, cut, copy, paste multiple lines in current editing program.

Start Line

- Path
    - 5 key：F2 Program → Next → F2 Block Copy → F1 Start Line
    - 8 key：F2 Program → F5 Block Copy → F1 Start Line
    - 10+8 key：G2 Program → F5 Block Copy → F1 Start Line

- Explanation
    - Select the start line of the editing block.

End Line

- Path
    - 5 key：F2 Program → Next → F2 Block Copy → F2 End Line
    - 8 key：F2 Program → F5 Block Copy → F2 End Line
    - 10+8 key：G2 Program → F5 Block Copy → F2 End Line

- Explanation
    - Select the end line of the editing block.

Block Cut

- Path
    - 5 key：F2 Program → Next → F2 Block Copy → F3 Block Cut
    - 8 key：F2 Program → F5 Block Copy → F3 Block Cut
    - 10+8 key：G2 Program → F5 Block Copy → F3 Block Cut

- Explanation
    - Cut the block that has been selected in current program (highlighted area).

Block Copy

- Path
    - 5 key：F2 Program → Next → F2 Block Copy → F4 Block Copy
    - 8 key：F2 Program → F5 Block Copy → F4 Block Copy
    - 10+8 key：G2 Program → F5 Block Copy → F4 Block Copy

- Explanation
    - Copy the block that has been selected in current program (highlighted area).

Block Paste

- Path
    - 5 key：F2 Program → Next → F2 Block Copy → F5 Block Paste
    - 8 key：F2 Program → F5 Block Copy → F5 Block Paste
    - 10+8 key：G2 Program → F5 Block Copy → F5 Block Paste

- Explanation
    - Paste the block that has been cut or copied to the line where the cursor is located.

Operation

1. Move the cursor to the start line of editing block and press "Start Line".
2. The "End Line" button will then be enabled.
3. Use arrow keys【↑】【↓】【Page Up】【Page Down】to determine the area of the required editing block, the selected area will be highlighted.
4. Press "End Line" after confirmed the editing area.
5. The "End Line" button will then be disabled and buttons "Block Copy" & "Block Cut" will be enabled.
6. Press "Block Cut", the highlighted block will be cut out.
7. For the meanwhile, buttons "Block Copy" & "Block Cut" will be disabled, and the button "Block Paste" will be enabled.
8. Move cursor to the line you want to paste, press "Block Paste" to paste the copied block to the line where the cursor is located.
9. If used "Block Copy", the highlighted area will then be dis-selected instead of being cut out.

Notifications

1. After pressing "Start Line", using【↑】or【Page Up】will move the start line upwards.
2. When pressing "Block Cut", the contents of the block will be removed from the program. The contents will be deleted if "Block Copy" is not executed.
3. When pressing "Block Cut", the highlighted block can only be pasted once.
4. When pressing "Block Copy", the highlighted block can be pasted for unlimited times.

2.4.6 Teach
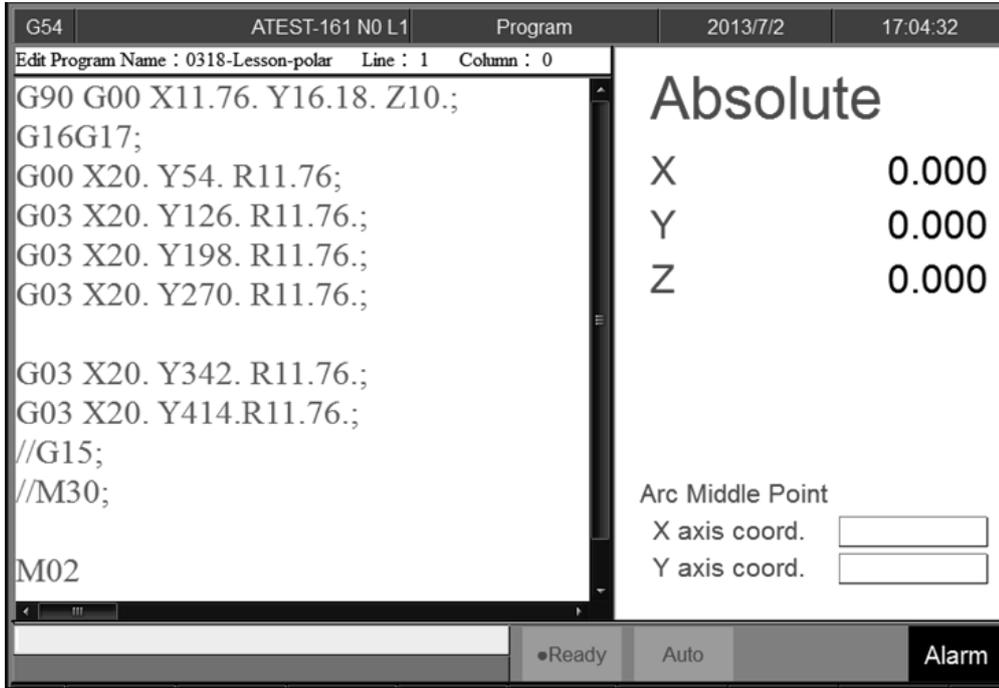
- Path
    - 5 key：F2 Program → Next → F3 Teach
    - 8 key：F2 Program → F6 Teach
    - 10+8 key：G2 Program → F6 Teach

- Explanation
    - Move machine to the assigned coordinate with "MPG"/"JOG"/"INJOG" and use "Teach" to set the current absolute coordinate of system into NC program.
    - It would be more convenient than input manually..

Rapid Teach

- Path
    - 5 key：F2 Program → Next → F3 Teach → F1 Rapid Teach
    - 8 key：F2 Program → F6 Teach → F1 Rapid Teach
    - 10+8 key：G2 Program → F6 Teach → F1 Rapid Teach

- Explanation
    - The current absolute coordinate will be used as the argument of G00, and be inserted into the line of current editing program where the cursor is located.

Line Cut Teach

- Path
    - 5 key：F2 Program → Next → F3 Teach → F2 Line Cut Teach
    - 8 key：F2 Program → F6 Teach → F2 Line Cut Teach
    - 10+8 key：G2 Program → F6 Teach → F2 Line Cut Teach

- Explanation

- The current absolute coordinate will be used as the argument of G01, and be inserted into the line of current editing program where the cursor is located.

Arc Cut Teach

- Path
  - 5 key：F2 Program → Next → F3 Teach → F3 Arc Cut Teach
  - 8 key：F2 Program → F6 Teach → F3 Arc Cut Teach
  - 10+8 key：G2 Program → F6 Teach → F3 Arc Cut Teach

- Explanation
  - The previous point set by "Rapid Teach", "Line Cut Teach", and "Point Teach" will be used as the start point for "Arc Cut Teach".
  - Move the machine to set the middle point and end point, G02 or G03 will then be inserted into the line where the cursor is located automatically.

- Operation Method
  a. Run any of "Rapid Teach", "Line Cut Teach", or "Point Teach" to set the start point for "Arc Cut Teach".
  b. Move machine to the middle point of the arc.
  c. Press "Arc Cut Teach".
  d. The current absolute coordinate will be set as the middle point of arc.
  e. Move machine to the end point of the arc.
  f. Press "Arc Cut Teach" again.
  g. The controller will automatically calculate the relationship between the middle point and the end point to see if it is G02 or G03.
  h. The result will be inserted into the line where the cursor is located.

Cancel Arc Middle Point

- Path
  - 5 key：F2 Program → Next → F3 Teach → F4 Cancel Arc Middle Point
  - 8 key：F2 Program → F6 Teach → F4 Cancel Arc Middle Point
  - 10+8 key：G2 Program → F6 Teach → F4 Cancel Arc Middle Point

- Explanation
  - Clear the set middle point value.
  - This button will be disabled when no middle point value is set.

Point Teach

- Path
  - 5 key：F2 Program → Next → F3 Teach → F5 Point Teach
  - 8 key：F2 Program → F6 Teach → F5 Point Teach
  - 10+8 key：G2 Program → F6 Teach → F5 Point Teach

- Explanation
  - Add the current absolute coordinate to the line where the cursor is located in point coordinate form.

2.4.7 Simulation
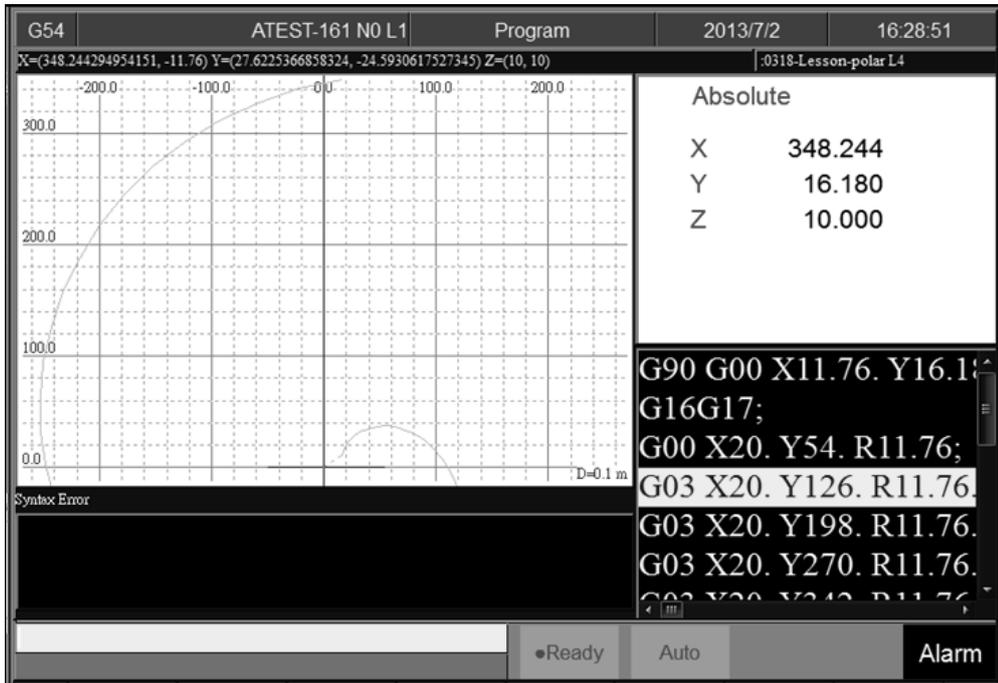
- Path
    - 5 key：F2 Program → F5 Simulation
    - 8 key：F2 Program → F7 Simulation
    - 10+8 key：G2 Program → F9 Simulation

- Explanation
    - This function can preview the machining path of the current editing program.
    - Syntax Error/ Estimation Data （Bottom left corner）
        - Simulation function can check program errors and show the result in this box.
        - Also have prediction function and will show the data in the box (Supporting Version：After 10.118.22)
            - If there are errors in current program, it'll show error code in this box.
            - Only shows the prediction result when the function is turned on and no alarms.
            - The function is turned off in default, and could be turned on through SI (SyntecIDE).
            Simulation Result Support:

Support	Not Support
G0、G53	G43、G44
G01、G02、G03、G02.4、G03.4	G68.2、G68.3、G53.1、G53.6
G04	G31、G33、G34、G28.1
G40、G41、G42	G10

Support	Not Support
G54~G59.9	Overlapping G81.1、G80.1 G12.1、G13.1 G114.1、G114.2、G114.3、G113、G51.2、G50.2
F code	M, S, T, B, D, H code
G17、G18、G19	Functions applied through system variables ex: #1000、#1501、#1517、#1814、#1822、#1829、#1881~#1898、#1901~#1918、#1930、#1931~#1933、#20001~#20658、#26001~#26078 etc
Rapid tapping/ Rapid drilling G900.81、G01.84	G05.1、G05(HPCC)、G120.1 G64、G09、G61、G63 G94、G95 G10.9、G07.1 G70、G71 G04.1
G43.4、G43.5、G49	

- The default display range is the maximum limit of the entire program.
- Users can modify relevant functions of the simulation through "Simulation Setting".
- It can display the axis name and coordinate information of the six axes XYZABC.
- When there are multiple identical geometric axes (XYZ) at the same time, only one geometry axis can be simulated normally. The simulation specifications are as follows:
    - For example：When z1 and z2 axes exist at the same time, the simulation function will regard z1 as the z axis. Thus, z1 can be simulated normally, but z2 cannot.
    - Similarly, when z2 and z3 axes exist at the same time, the simulation function will regard z2 as the z-axis. Thus, z2 can be simulated normally, but z3 cannot.

Step

- Path
    - 5 key：F2 Program → F5 Simulation → F1 Step
    - 8 key：F2 Program → F7 Simulation → F1 Step
    - 10+8 key：G2 Program → F9 Simulation → F1 Step
- Explanation
    - The path will be simulated block by block.
    - You can check the coordinate changes after the execution of block.
    - If single block function is disable by #1502, this step function won't work.

Continue

- Path
    - 5 key：F2 Program → F5 Simulation → F3 Continue
    - 8 key：F2 Program → F7 Simulation → F3 Continue
    - 10+8 key：G2 Program → F9 Simulation → F3 Continue

- Explanation
    - The system will first scan the entire program, then start the simulation.

Zoom

- Path
    - 5 key：F2 Program → F5 Simulation → F4 Zoom
    - 8 key：F2 Program → F7 Simulation → F4 Zoom
    - 10+8 key：G2 Program → F9 Simulation → F4 Zoom

- Explanation
    - Zoom in the scale of the simulation.
    - Provided for software versions after 10.116.40(included)

- Operation Method
    a. Press "Zoom", a square selection box will appear. You can use the arrow keys【↑】【↓】【←】【→】to move and specify the area you want to zoom in.
    b. Use【Page Up】【Page Down】to change the size of the square box.
    c. Select the area and zoom in with【ENTER】

Graph Reset

- Path
    - 5 key：F2 Program → F5 Simulation → F5 Graph Reset
    - 8 key：F2 Program → F7 Simulation → F5 Graph Reset
    - 10+8 key：G2 Program → F9 Simulation → F5 Graph Reset

- Explanation
    - Reset the scale to display the simulation of entire program.

Undo Step

- Path
    - 5 key：F2 Program → F5 Simulation → F2 Undo Step
    - 8 key：F2 Program → F7 Simulation → F2 Undo Step
        - Note：The original fenubars from F2 are moved next
    - 10+8 key：G2 Program → F9 Simulation → F2 Undo Step

- Explanation
    - Available in software versions after 10.116.40 (included).
    - Can be used to return to the previous path block by block to check the coordinate change and line number display.
    - The "Undo Step" function can be executed after using the "Step" function.
    - The most steps you can go back is 10 steps.
    - This function can't be used right after using "Continue", "Zoom", and "Graph Reset". (software version 10.116.45 and after)

- While using "Undo Step" function, the executions of "Zoom" and "Graph Reset" will cancel the "Undo state". The path and line number will return to the final result we using "Step" function. (Software version 10.116.45 or after)

Simulation Parameter Setting
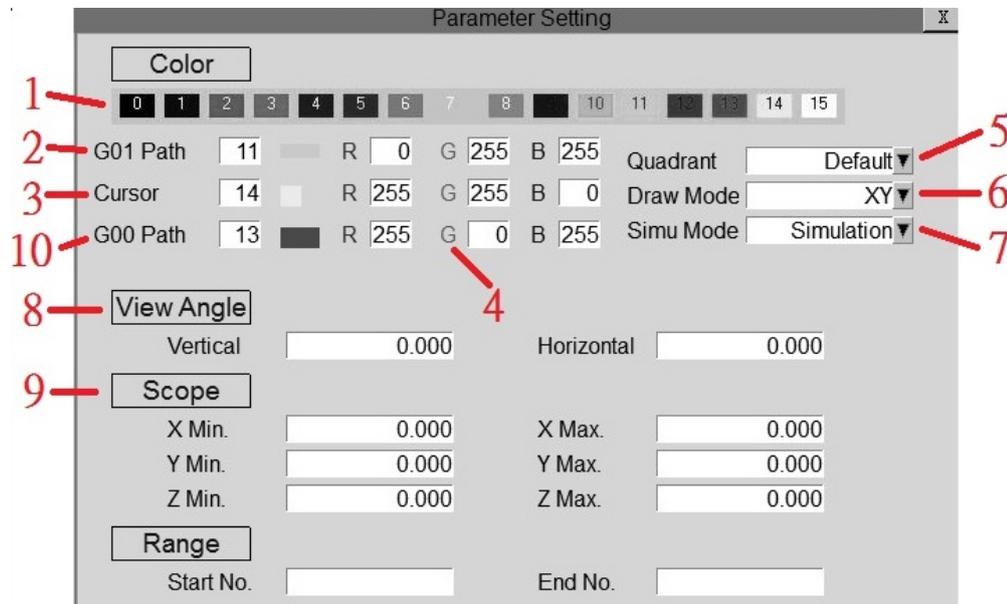
- Path
    - 5 key：F2 Program → F5 Simulation → F5 Simulation Setting
    - 8 key：F2 Program → F7 Simulation → F5 Simulation Setting
    - 10+8 key：G2 Program → F9 Simulation → F6 Simulation Setting

- Explanation
    - Setup related options of simulation.

- Simulation setting
    a. Color
        - Provide 16 colors from number 0-15.

    b. G01 Path
        - Path color of G01.
        - Provide 16 choices from 0-15.
        - Change the RGB value simultaneously after setting color number.

    c. Cursor
        - Color of cutting point.
        - Provide 16 choices from 0-15.
        - Change the RGB value simultaneously after setting color number.

    d. RGB Value
        - In addition to the colors provided by the system, user can use RGB Value to set your own color for simulation.
        - After setting, the column corresponding to each RGB Value will be changed. If the RGB setting value does not fit to any color code, the color column shows "-".

    e. Quadrant
        - Users can set the quadrant of the simulation plane specified by "Draw Mode".
        - Available Quadrant

- First quadrant
- Second quadrant
- Third quadrant
- Forth quadrant

f. Draw Mode
- Users can set the plane of the simulation.
- Available plane
  - XYZ
  - XY
  - YZ
  - ZX
  - YX
  - ZY
  - XZ

g. Simulation Mode
- Set the simulation mode
  1. Simulation
     - When user switches to "Monitor" page, the simulation action will be automatically performed.
     - There is no need to set the drawing boundary. After the system scans the entire program, the simulation will begin.

  2. Direct Draw
     - When user switches to "Monitor" page, the simulation action will not be performed immediately, However, the cutting point will be displayed.
     - Drawing boundaries must be set in advance.
     - When the cutting begins, the simulation also begins and the cutting point will move along the simulation path.

  3. Not Simulation
     - Turns off simulation.

h. View Angle
- When Draw Mode is set to XYZ, the 3D view angle can be set through this parameter.
- Available option
  - Vertical
  - Horizontal

i. Scope
- Set the simulation scope.
  - Minimum
    - X axis
    - Y axis
    - Z axis
  - Maximum
    - X axis
    - Y axis
    - Z axis

- If there is no value for Scope setting, the scope can be specified as follows:
  1. According to the boundary commands in NC file (Commands reference：产机使用说明 G10 L10000).
  2. If G10 L10000 is not used, the scope will be calculated according to NC file by the function automatically.

j. G00 Path
- Path color of G00.
- Provide 15 color choices from 1-15.
- Change the RGB value simultaneously after setting color number.

2.4.8 File ManagerStart MPG Shift

G54		ATEST-161 N0 L1	File Manager	2013/7/2	15:52:57

\DiskC2\OpenCNC\NcFiles   Free Space: 144506093KB

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

	●Ready	Auto		Alarm

- Path
    - 5 key: F2 Program → F4 File Manager
    - 8 key: F2 Program → F8 File Manager
    - 10+8 key: G2 Program → F10 File Manager / G4 Monitor → F8 File Manager

- Explanation
    - This button can perform file management function to all NC files stored in the disk designated by Pr3213.
    - The NC file set as main program and the folder to store this file are marked by arrow icon.
    - The sorting type can be set through [FileList Sorting Type] in "Set MMIConfig" screen. (Note 1)
    - When entering this screen, the files may not have been loaded to the list. During the loading process, the screen will flash but will not affect the operation. The "Select All" button will be enabled after the loading process is completed.

- Operation Method
    a. Use the arrow keys [↑] [↓] to move the cursor.
    b. Use【Page Up】【Page Down】to switch pages.
    c. Press【ENTER】and the file where the cursor is located would be specified as the editing file, the screen will show the content of the program, users are then able to edit the file.

New File

- Path
    - 5 key: F2 Program → F4 File Manager → F1 New File
    - 8 key: F2 Program → F8 File Manager → F1 New File

- • 101+8 key: G2 Program → F10 File Manager / G4 Monitor → F8 File Manager → F1 New File
- • Explanation
    - • Press this button to add a new file, the newly added file will be specified as the current editing file.
- • Operation Method
    - • Press "New File", input the file name in the pop-up dialog window.
- • Attention
    - • The file has no filename extension in default, if you want to create a file with extension, EX: *.NC, just input "*.NC" while inputting the file name.
    - • The length of file name (including the extension) cannot be more than 31 characters.

Copy File

- • Path
    - • 5 key: F2 Program → F4 File Manager → F2 Copy File
    - • 8 key: F2 Program → F8 File Manager → F2 Copy File
    - • 10+8 key: G2 Program → F10 File Manager → F2 Copy File / G4 Monitor → F8 File Manager → F2 Copy File
- • Explanation
    - • Press this button to copy the file currently highlighted.
- • Operating Method
    a. Use the arrow key【↑】【↓】to choose the copying file.
    b. Select and press "Copy File"
    c. Input a new file name in the pop-up dialog window
- • Attention
    - • The file has no filename extension in default, if you want to create a file with extension, EX: *.NC, just input "*.NC" while inputting the file name.
    - • The length of file name (including the extension) cannot be more than 31 characters.

Delete File

- • Path
    - • 5 key: F2 Program → F4 File Manager → F3 Delete File
    - • 8 key: F2 Program → F8 File Manager → F3 Delete File
    - • 10+8 key: G2 Program → F10 File Manager → F3 Delete File / G4 Monitor → F8 File Manager → F3 Delete File
- • Explanation
    - • Press this button to delete the file currently highlighted.
- • Operation Method
    - • After pressing "Delete File", the file selection window would appear on the screen. Use the arrow key【↑】【↓】to choose the file to delete.
- • Sub-Function key
    a. Select: Select the file to be deleted, you can choose more than one file at a time, you can also cancel the selection.
    b. Select All: Select all files.
    c. Cancel Select: Cancel selection.
    d. Delete File: Delete all selected files.
    e. Delete All: Delete all files.
- • Attention

- The executing files of files under editing cannot be deleted.

File Transfer

- Path
  - 5 key: F2 Program → F4 File Manager → F4 File Transfer
  - 8 key: F2 Program → F8 File Manager → F4 File Transfer
  - 10+8 key: G2 Program → F10 File Manager → F4 File Transfer / G4 Monitor → F8 File Manager → F4 File Transfer

- Explanation
  - Transfer files between the controller and the external file folder.

File Import

- Path
  - 5 key: F2 Program → F4 File Manager → F4 File Transfer → F1 File Import
  - 8 key: F2 Program → F8 File Manager → F4 File Transfer → F1 File Import
  - 10+8 key: G3 Program → F10 File Manager → F4 File Transfer → F1 File Import / G4 Monitor → F8 File Manager → F4 File Transfer → F1 File Import

- Explanation
  - Imports folders or files into the controller from external device.

- Screen Explanation
  - The external device options are shown on the screen, the options are:
    - USBDisk
    - DiskA
    - Network
    - USBDisk2

  - The bottom left shows the file structure of external device.
  - The bottom right shows the NC files currently saved in the internal disk of controller.

- Sub-Function key
  a. Copy: import the chosen folder or file from the external device to the internal disk.
  b. Select: Press this button to select a file, it's able to select more than one at a time, you can also cancel the selection.
  c. Select All: Select all files.
  d. Cancel Select: cancel selection.
  e. Device Select: change the source of external device.

- Operation Method
  a. Pressing "File Import" will show the selection window.
  b. The default external device is USBDisk.
  c. If you want to change the source of the importing file, press "Device Select", move the cursor to the external device option, use arrow keys【←】【→】to select the external device and press【Enter】, the bottom left section of the screen will also be updated.
  d. Use arrow keys【↑】【↓】to choose the file to be imported.
  e. Move the cursor to the importing files and press "Select" or【Space】to select the file.
  f. Press "Copy" after all importing files are selected, all the selected files and folders would be imported from the external device to the internal disk.
     i. Please note that all files in the folder will be copied.

- Restriction of copy folder
  a. Folders cannot be imported into sub-folder in \DiskA\NcFile.

b. If a folder contains sub-folders and is going to be imported into \DiskA\NcFile, a hint "Sub-folder detected, please note that the sub-folder will not be imported" will appear.

File Export

- Path
  - 5 key: F2 Program → F4 File Manager → F4 File Transfer → F2 File Export
  - 8 key: F2 Program → F8 File Manager → F4 File Transfer → F2 File Export
  - 10+8 key: G2 Program → F10 File Manager → F4 File Transfer → F2 File Export / G4 Monitor → F8 File Manager → F4 File Transfer → F2 File Export

- Explanation
  - Exports the files or folders in the controller to external device.

- Screen Explanation
  - The external device options are shown on the screen, the options are:
    - USBDisk
    - DiskA
    - Network
    - USBDisk2

  - The bottom left is NC files currently saved in internal disk of the controller.
  - The bottom right is the file structure of external device.

- Sub-Function key

1. Copy: Export the chosen folders or files from the internal disk to the external device.
2. Select: Press this button to select a file, it's able to select more than one at a time, you can also cancel the selection.
3. Select All: select all files.
4. Cancel select: cancel selection.
5. Device Select: change the source of external device.
6. Window Select: switch cursor between internal disk and external device.

- Operation Method
  a. Pressing "File Export" will show the selection window.
  b. The default external device is USBDisk.
  c. If want to change the export destination, press "Device Select", move the cursor to the external device option, use arrow keys【←】【→】to select the external device and press【Enter】, the bottom right part of the screen will also be updated.
  d. Press "Window Select" to switch cursor from external device to internal disk, vice versa.
  e. If want to switch to the folder in external device, use arrow keys【↑】【↓】to move cursor to the target folder and press【Enter】.
  f. Use the arrow keys【↑】【↓】to select the files to be exported from the internal disk.
  g. Move cursor to the exporting file and press "Select" or【Space】to select the file.
  h. Press "Copy" after all the exporting files are selected, all the selected files and folders will be exported from the internal disk to the external device.
    i. Please note that all files in the folder will be copied.

Execute

- Path
  - 5 key: F2 Program → F4 File Manager → F5 Execute
  - 8 key: F2 Program → F8 File Manager → F5 Execute
  - 10+8 key: G2 Program → F10 File Manager → F5 Execute / G4 Monitor → F8 File Manager → Execute

- Explanation

- This button can assign the current editing program as the program to be executed and change the screen to "Monitor" page.
- Operation Method
    a. Use arrow keys【↑】【↓】 to select the file. Press "Execute" after finished selecting, the selected file will be assigned as the machining file.
    b. The screen will be switched to "Monitor" screen.

- Attention
    - This button is disabled during machining.

Select DNC File

- Path
    - 5 key: F2 Program → F4 File Manager → Next → F6 Select DNC File
    - 8 key: F2 Program → F8 File Manager → F6 Select DNC File
    - 10+8 key: G2 Program → F10 File Manager → F6 Select DNC File / G4 Monitor → F8 File Manager → F6 Select DNC File

- Explanation
    - Specification modification
        - In versions 10.116.26C, 10.116.36O, 10.116.44, 10.117.44 or earlier, the main program of this function is called USBDNC .
        - In versions 10.116.26D, 10.116.36P, 10.116.45, 10.117.45 or later, the main program of this function is renamed as FILEDNC.

    - This function is a built-in FILEDNC file transfer function of the controller, which is independent and different from NETDNC, which transfers through customized front-end.
    - With this function, larger files can be stored in external device (USB flash drive or network), and be uploaded to the controller while machining.

- Operation Method

- Use arrow keys【↑】【↓】to select the file, press "OK" after selecting.
- The selected file will be assigned as the machining file, the file name will be DNC: + specified file name.

- Attention
    - The function is supported from version 6MDs/6s: 2.2.0(10.116.10) 11s: 2.2.1(10.116.10A) 21s: 2.2.0(10.116.10) 20s: 10.116.24
    - When the file cannot be read, an error message will pop up after retrying 100 times.
    - In DNC mode, the main program needs to end with M30. Do not use M99 for repetitive machining, otherwise it will cause unpredictable results.
    - The main program applying FILEDNC does not support MACRO syntax.
    - FILEDNC main programs does not support jumping syntax, such as GOTO.
    - FILEDNC main programs support the start block function (after 2.2.5), such as O888800, O999900 and Pr.3851=0.
    - When M98, G65, and G66 are used to call a subprogram, the subprograms must be copied into the controller first.

Folder Manager

- Path
    - 5 key: F2 Program → F4 File Manager → Next → F2 Folder Manager
    - 8 key: F2 Program → F8 File Manager → F7 Folder Manager
    - 10+8 key: G2 Program → F10 File Manager → F7 Folder Manager / G4 Monitor → F8 File Manager → F7 Folder Manager

- Explanation
  - This button can switch to the next fenubar layer to add/delete/rename the folder.
  - Supporting version: 10.116.36, 10.117.36

- Sub-Function key
  - New Folder: Add sub-folders in NcFiles.
  - Delete Folder: Move the cursor to the sub-folder to delete the sub-folder and all the files within.
  - Rename Folder: Move the cursor to the sub-folder to rename it.

- Attention
  - Users can add new sub-folder under NcFiles, but cannot add sub-folder under sub-folder.
  - Restrictions of folder name
    - The length cannot be more than 31 characters.
    - Special symbols are not allowed. Ex: ", "<", ">", "|", ":", "*", "?", "\", "/", " ".
    - Two folders with the same name is not allowed.
    - Folder with the same name as existing files is not allowed.
  - Cannot use "Delete Folder" function to delete a file.
  - Cannot use "Rename Folder" function to rename a file.
  - If the folder is in use, it'll be unable to delete or rename the folder.
    - When a file in the folder is selected for machining or editing, the folder will be considered in use.
  - If the main program and subprogram belong to different folders
    - The system will automatically find the subprogram in other folder. The level of the subprogram must be equal to or higher than the one of main program.
      - EX: When the main program belongs to NcFiles folder
        - Subprograms under NcFiles and Macro folder will be executed normally.
        - Subprograms under sub-folder will not be executed normally.
    - The folder level from low to high are respectively: sub-folder, NcFiles folder and Macro folder
  - MDIBlock file is always in NcFiles folder and is not supported in sub-folders.
  - When the main programs with same name exists in different folders, the one in chosen folder will be executed first.
  - Record the directory of the last operation. (10.116.36O, 10.116.45, 10.117.45)
    - The operation actions include adding new file, selecting file and leaving file manager page.
    - Default directory after booting is NCFILES.

Appendix

- Note 1 : The specification of【6- ByName (Sort by number)】in the filelist sorting type are as following:
  - Sorted by letter (0~9~A~Z)
    A0
    AA
    AB
    CDE
  - Assume that the first half in English is the same, then it'll be sorted by number.
    A1
    A9
    A10
    A19
    A20
    A102

- Assume that the first half in English is the same, and sorted by number, if the values of the number are also the same, it will be sorted by the 0s.

  A1
  A01
  A001

- Assuming that the first half in English is the same, and sorted by number, if the values of the number are the same, it will be sorted by 0.

  A1
  A01
  A9

- If the letters and numbers in the first half are the same, the second half will be sorted by the same rules listed above.

  A1   //There are no characters after 1, so it is listed before A01B3, just need to compare with A001
  A001   //There are no characters after 1, so it is listed before A01B3, just need to compare with A1.
  A01B3 //The letters and values of A01 and A1 are the same, so just compare B3 with B4 in the second half.
  A1B4   //The letters and values of A01 and A1 are the same, so just compare B4 with B3 in the second half.

- If the letters and numbers in the first half are the same, the second half will be sorted by the same rules listed above.

  A1B2   //The values of A1 and A01 are the same, so just compare B2 with B3, B4 and C in the second half.
  A01B3 //The values of A01 and A1 are the same, so just compare B3 with B2, B4 and C in the second half.
  A1B4   //The values of A1 and A01 are the same, so just compare B4 with B2, B3 and C in the second half.
  A1C   //The values of A1 and A01 are the same, so just compare C with B2, B3 and B4 in the second half.

2.4.9 Goto Line

- Path
  - 5 key：F2 Program → Next → F4 Goto Line
  - 8 key：F2 Program → Next → F1 Goto Line
  - 10+8 key：G2 Program → F8 Goto Line

- Explanation
  - Move the cursor to the specified line number of the NC file, it's suitable for editing or checking large NC files.

- Operation Method
  a. Input the line number and press "OK".

2.5 Offset/Setting

- Path
  - 5 key, 8 Key： F3 Offset/Setting
  - 10+8 key： G3 Offset/Setting

- Explanation
  - Offset/Setting function setup are under this fenubar group
  - Able to switch between pages in the group with hot key【Offset/Setting】

2.5.1 Workpiece Coord.

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

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

						Relative	
						X	0.000
						Y	0.000
						Z	0.000
						A	4.158

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

	●Ready	Auto		Alarm

- Path
    - 5 key, 8key： F3 Offset/Setting→F1 Workpiece Coord.
    - 10+8 key：G3 Offset/Setting→ F1 Workpiece Coord.

- Explanation
    - Switch to "Workpiece Coord." screen with this fenubar
    - System default will be G54 if there are no specific declaration with G54~G59.10 in the NC program.
    - External Coordinate Shift：Effective in all workpiece coordinates (G54~G59.10)

- Operation Method:
    - Move the cursor with【↑】【↓】【←】【→】.
    - Use【PageUp】【PageDown】to switch the pages.

- Note
    - After setting the workpiece coordinate, please set the tool length compensation again.

- Modification Timing
    - External Shift：
        - i. Able to modify：Block stop(ex:M00/M01), ready or not ready not caused by C40
        - ii. Not able to modify：Block stop or feedhold caused by C40 during machining or pre-interpretation

    - Workpiece Coordinate (G54P1(G54), G54P2(G55), ... G54P100)：
        - i. Able to modify：
            1. Block stop, ready or not ready not caused by C40;
            2. Block stop or feedhold caused by C40, but not modifying the using machining coordinate.

        - ii. Not able to modify：
            1. During machining or pre-interpretation;

2. Block stop or feedhold caused by C40 and modifying the using workpiece coordinate

Mach. Coord. Teach

- Path
    - 5 key, 8key：F3 Offset/Setting → F1 Workpiece Coord. → F1 Mach. Coord. Teach
    - 10+8 key：G3 Offset/Setting → F1 Workpiece Coord. → Mach. Coord. Teach

- Explanation
    - Set current cursor located workpiece coordinate value into the corresponding machine coordinate value

- Operation Method
    a. Move machine to the target location.
    b. Move cursor to the modifying workpiece coordinate
    c. Press "Mach. Coord. Teach"
    d. Original value of workpiece coordinate will then be replaced by the corresponding machine coordinate value

- Example
    a. Current machine coordinate of X axis is 5.000
    b. Current G54 X axis value is 0.000
    c. Move cursor to the G54 X axis coordinate
    d. Press "Mach. Coord. Teach"
    e. The G54 X axis value will then become 5.000

Rel. Coord. Teach

- Path
    - 5 key, 8 key：F3 Offset/Setting → F1 Workpiece Coord. → F2 Rel. Coord. Teach
    - 10+8 key：G3 Offset/Setting → Workpiece Coord. → F2 Rel. Coord. Teach

- Explanation
    - Set current cursor located workpiece coordinate value into the corresponding relative coordinate value

- Operation Method
    a. Move machine to the target location.
    b. Move cursor to the modifying workpiece coordinate
    c. Press "Rel. Coord. Teach"
    d. Original value of workpiece coordinate will then be replaced by the corresponding relative coordinate value

- Example
    a. Current relative coordinate of X axis is 5.000
    b. Current G54 X axis value is 0.000
    c. Move cursor to the G54 X axis coordinate
    d. Press "Rel. Coord. Teach"
    e. The G54 X axis value will then become 5.000

Aux. Coord. Teach

- Path
    - 5 key, 8 key：F3 Offset/Setting → F1 Workpiece Coord. → F3 Aux. Coord. Teach
    - 10+8 key：G3 Offset/Setting → F1 Workpiece Coord. → F3 Aux. Coord. Teach

- Explanation

- Set the current cursor located workpiece coordinate value into the corresponding auxiliary coordinate value.
- The Aux. coordinate value only appears after applying middle function.

- Operation Method
  a. Apply the middle function to calculate the aux. coordinate value.
  b. Move cursor to the modifying workpiece coordinate
  c. Press「Aux. Coord. Teach」
  d. Original value of workpiece coordinate will then be replaced by the corresponding Aux. coordinate value.

- Example
  a. Current Aux. coordinate of X axis is 5.000
  b. Current G54 X axis value is 0.000
  c. Move cursor to the G54 X axis coordinate
  d. Press "Aux. Coord. Teach"
  e. The G54 X axis value will then become 5.000

Inc. Input

- Path
  - 5 key, 8 key：F3 Offset/Setting→F1 Workpiece Coord.→F4 Inc. Input
  - 10+8 key：G3 Offset/Setting→ F1 Workpiece Coord.→ Inc. Input

- Explanation
  - Change the cursor located workpiece coordinate value into (cursor located coordinate value + keyed in teaching value)

- Operation Method
  a. Move cursor to the modifying workpiece coordinate.
  b. Input the teaching value.
  c. Press "Inc. Input"
  d. The workpiece coordinate value will then become the value of cursor located coordinate value + keyed in teaching value.

- Example
  a. Current G54 X axis value is 0.000
  b. Move cursor to the G54 X axis coordinate
  c. Input 10.000
  d. Press "Inc. Input"
  e. The G54 X axis value will then become 15.000

Mach. Coord. Inc. Teach

- Path
  - 5 key：F3 Offset/Setting→F1 Workpiece Coord.→Next→F1 Mach. Coord. Inc. Teach
  - 8 key：F3 Offset/Setting→F1 Workpiece Coord.→F6 Mach. Coord. Inc. Teach
  - 10+8 key：G3 Offset/Setting→F1 Workpiece Coord. →F6 Mach. Coord. Inc. Teach

- Explanation
  - Change the current cursor located workpiece coordinate value into (corresponding machine coordinate value + keyed in teaching value)

- Operation Method
  a. Move cursor to the modifying workpiece coordinate
  b. Input the teaching value
  c. Press "Mach. Inc. Teach"

    d. The workpiece coordinate value will then become the value of its corresponding machine coordinate value + keyed in teaching value.

- Example
  - a. Current machine coordinate of X axis is 0.000
  - b. Move cursor to the G54 X axis coordinate
  - c. Input 10.000
  - d. Press "Mach. Inc. Input"
  - e. The G54 X axis value will then become 15.000

Middle Function

- Path
  - 5 key, 8key：F3 Offset/Setting → F1 Workpiece Coord. → F5 Middle Func.
  - 10+8 key：F3 Offset/Setting → Workpiece Coord. → Middle Func.

- Explanation
  - This function helps to calibrate the middle point of workpiece before machining.
  - The value calculated by middle function will be input to G54~G59 workpiece coordinates for coordinate setups.
  - Middle function provides 2 modes, manual mode and auto mode.

Manual Middle Function

- Explanation
  - Set middle function to mode 0
  - User controls the machine with MPG and moves the machine to the end point of workpiece in X, Y axis direction respectively to record the machine location. The system will automatically calculate the X, Y axis value of workpiece center point.

- Operation method
  - a. Move the tool with MPG, reach Px1 point in the figure and press "Px1 Set". The current location will be set as the machine coordinate of Px1 and be recorded on the screen. The machine coordinate of workpiece middle point in X axis direction will be calculated using Px1 and Px2, the result will be shown in the Pxm column and Aux. Coord. X column.
  - b. Move the tool with MPG, reach Px2 point in the figure and press "Px2 Set". The current location will be set as the machine coordinate of Px2 and be recorded on the screen. The machine coordinate of workpiece middle point in X axis direction will be calculated using Px1 and Px2, the result will be shown in the Pxm column and Aux. Coord. X column.
  - c. Move the tool with MPG, reach Py1 point in the figure and press "Py1 Set". The current location will be set as the machine coordinate of Py1 and be recorded on the screen. The machine coordinate of workpiece middle point in Y axis direction will be calculated using Py1 and Py2, the result will be shown in the Pym column and Aux. Coord. Y column.
  - d. Move the tool with MPG, reach Py2 point in the figure and press "Py2 Set". The current location will be set as the machine coordinate of Py2 and be recorded on the screen. The machine coordinate of workpiece middle point in Y axis direction will be calculated using Py1 and Py2, the result will be shown in the Pym column and Aux. Coord. Y column.
  - e. The value of Pxm and Pym is the coordinate of workpiece middle point.
  - f. For controller software versions before 10.116.24x(included), first switch to workpiece coordinate setting screen, then move cursor to the required workpiece coordinate system and press F3 "Aux. Coord. Teach", the system will input the Aux. Coord. of the cursor located axis to the column.
  - g. For controller software versions above 10.116.34(included), first move cursor to "Workpiece Coord. No. P" on middle function screen, then input the required workpiece coordinate number and press F5 "Aux. Coord. Auto Setting", the system will input the Aux. Coord. in X, Y axis direction to the corresponding workpiece coordinate.

Auto Middle Function

- Explanation
    - Set middle function to mode 1
    - Different from the manual middle function, user only needs to enter the workpiece dimension and set the boundary coordinate. After leading the tool to the starting point and press start, the system will search for the workpiece middle point automatically.

- Parameter description
    - Length I: Workpiece length in X axis direction
    - Width J: Workpiece width in Y axis direction
    - Safety Distance H: When the middle function starts, the initial position will be located near the center of workpiece, while moving from the center to the outer edge of workpiece, descending to Z axis coordinate P2 and searching for the border, a safety distance is required to avid collision between the measuring tool and workpiece.
    - Feedrate F: Detecting speed of auto middle function
    - Z axis Coordinate P2: This Z axis coordinate is to ensure that the measuring tool is able to contact the border of workpiece when running auto middle function. It can be set by leading the tool to P2 and press "Z axis position teach".

- Operation method
    a. After setting up following the descriptions, place the tool above workpiece upper surface and near the center point, distance between the tool and the upper surface should allow the tool to move freely and not touching the surface. The point will be the starting point of auto middle function.
    b. Switch to auto mode and press [Start Auto Middle Function] to begin
    c. The system will move the tool and contact the workpiece border according to the set data, the coordinate values will be shown on the screen. X, Y coordinate of the workpiece center point will then be calculated.
    d. For controller software versions before 10.116.24x(included), first switch to workpiece coordinate setting screen, then move cursor to the required workpiece coordinate system and press F3 "Aux. Coord. Teach", the system will input the Aux. Coord. of the cursor located axis to the column.
    e. For controller software versions above 10.116.34(included), first move cursor to "Workpiece Coord. No. P" on middle function screen, then input the required workpiece coordinate number and press F5 "Aux. Coord. Auto Setting", the system will input the Aux. Coord. in X, Y axis direction to the corresponding workpiece coordinate.

2.5.2 Tool Set

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

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

Absolute

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

Machine
X	0.000
Y	0.000
Z	0.000
A	4.158

Dist. To Go
X	0.000
Y	0.000
Z	0.000
A	0.000

Relative
X	0.000
Y	0.000
Z	0.000
A	4.158

●Ready   Auto   Alarm

- Path
    - 5 key, 8 key: F3 Offset/Setting → F2 Tool Set
    - 10+8 key: G3 Offset/Setting → F2 Tool Set

- Function
    - Press the button to switch to the compensation setting page.
    - Actual G41/G42 compensation value = Tool nose + Tool radius wearing compensation
    - Actual G43/G44 compensation value = Tool length compensation + Tool length wearing compensation
    - Provides 200 sets of tool compensation setups (only 96 sets for versions before 10.116.36D)

- Parameter Explanation
    - Diameter Geometry: G41/G42 Cutter radius Dn compensation (please input radius, not diameter)
    - Diameter Wear: Minor modification of cutter radius
    - Length Geometry: G43/G44 Tool length Hn compensation.
    - Length wear: Minor modification of cutter length

- Operation method
    a. Move the cursor with arrow keys【↑】【↓】【←】【→】
    b. Use【PageUp】【PageDown】to switch the page.
    c. Key in method:
        - Normally applies absolute mode for tool nose or tool length compensation
        - Normally applies incremental mode for cutter radius wearing compensation or tool length wearing compensation

    d. Absolute input:
        - Press "A" and press "Enter"
        - The value of where the cursor is located will be set as the input value.

e. Incremental input:
- Press "I" and press "Enter".
- The value of where the cursor is located will be set as input value + current cursor value.

f. Measure input:
i. Press "Z" and press "Enter", the value of where the cursor is located will be set as the current Z axis "relative coordinate" value.
ii. Press F2 Set Tool Mach. Coord., the tool length compensation value of where the cursor is located will be set as current Z axis "machine coordinate" value.(valid version from 2.2.0)
iii. Press F3 Set Tool Rel. Coord., the tool length compensation value of where the cursor is located will be set as current Z axis "relative coordinate" value. (valid version from 2.2.0)

Clear Z Coord.

- Path
  - 5 key, 8 key: F3 Offset/Setting → F2 Tool Set → F1 Clear Z Coord.
  - 10+8 key: G3 Offset/Setting → F2 Tool Set → F1 Clear Coord.

- Function
  - Clear the current Z axis relative coordinate value.

Tool No.

- Path
  - 5 key, 8 key: F3 Offset/Setting → F2 Tool Set → F7 Tool No.
  - 10+8 key: G3 Offset/Setting → F2 Tool Set → F7 Tool No,

- Function
  - Please refer to chapter 3.4.4 for details

Tool Life Manag.

- Path
  - F3 Offset/Setting → F2 Tool Set → F8 Tool Life Manag.

- Function
  - Please refer to chapter 3.4.5 for details

2.5.3 Tool Tip Measure

- Command
  - F3 Offset/Setting→F3 Tool Tip Measure
  - G3 Offset/Setting→F3 Tool Tip Measure

- Function
  - When the machine is equipped with tool presetting and measuring machine, users can move the machine table to the measuring machine location to measure tool tip through parameter setting.
  - Under different conditions, here are four methods
    -Single tool, single workpiece
    -Single tool, multi workpiece
    -multi tool, multi workpiece
    -5-axis (need to turn on Option12 or Option13)
    -Refer to chapter 3.4.3

2.5.4 User Param.

- Command
    - 5 key, 8 key: F3 Offset/Setting → F4 User Param.
    - 10+8 key: G3 Offset/Setting → F4 User Param.

- Function
    - SYNTEC controller provided users to set the required related machining parameter.

- Parameters
    - Please refer to "Mill Parameter Manual" for further details.

2.6 Monitor
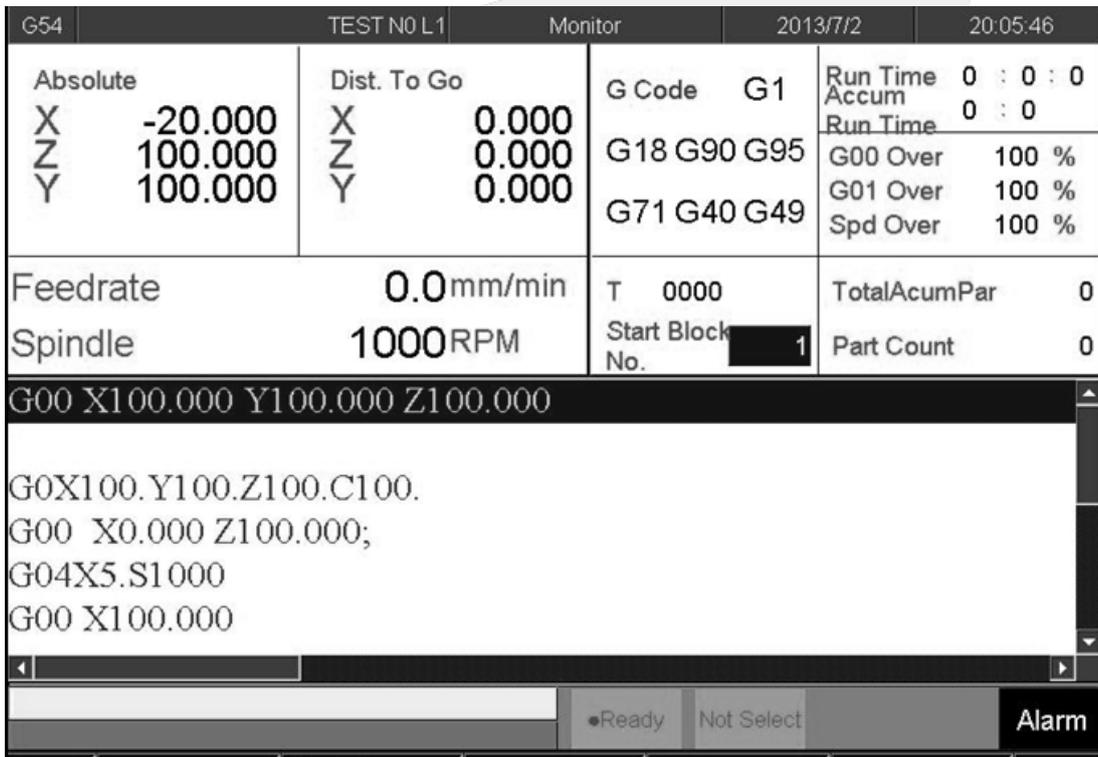
- Path
    - 5 key, 8 key：F4 Monitor
    - 10+8 key：G4 Monitor
- Function
    - This fenubar group provides the necessary information which needs to be monitored during machining process.

Fenubar Introduction

- F1：Edit
Explanation：Change the screen to "Program Editor" and load current NC file to edit.

- F2：Simulation Switch
Explanation：Display graphic simulation components.

    F3：MDI Input

Explanation：Edit MDI program under MDI state.

- F4：Param. Set
  Function：Used to switch between "Information" and "Setting"
  - Screen 1：Line No. Col. No., Parameter Set, Part Count

工件数设定			
总工件数	0	工件数	0
需求工件数	0		

加工参数设定		
进给速率	1000.000	mm/min
主轴转速	1000	RPM

G120.1 P0	断点行号	0
	断点序号	0

  - Screen 2：G code, Time, Override

G码状态　G1	加工时间　0：0：0
G17 G90 G94	累计加工　0：0：0
G71 G40 G49	G00 倍率　50 %
	G01 倍率　100 %
	主轴倍率　100 %
总工件数　0	T　0 D　0 H　0
工件数　0	起始单节　　1

  - Screen 3：Spindle Speed, Override, Loading (according to the number of supporting axes)

	转速	倍率	负载率
S1	1000 RPM	100 %	%
S2	0 RPM	100 %	%
S3	0 RPM	100 %	%
S4	0 RPM	100 %	%
S5	0 RPM	100 %	%
S6	0 RPM	100 %	%
S7	0 RPM	100 %	%
S8	0 RPM	100 %	%
S9	0 RPM	100 %	%
S10	0 RPM	100 %	%

  (support more than 8 axes)

	转速	倍率	负载率	
S1	1000 RPM	100 %		%
S2	0 RPM	100 %		%
S3	0 RPM	100 %		%
S4	0 RPM	100 %		%
S5	0 RPM	100 %		%
S6	0 RPM	100 %		%
S7	0 RPM	100 %		%
S8	0 RPM	100 %		%

(support more than 6 axes)

	转速	倍率	负载率	
S1	1000 RPM	100 %		%
S2	0 RPM	100 %		%
S3	0 RPM	100 %		%
S4	0 RPM	100 %		%
S5	0 RPM	100 %		%
S6	0 RPM	100 %		%

- F5：Tool Wear Set
  Function：Show the tool wear set screen
  ARM machine：Tool Wear Set and Information are at the same fenubar layer, the tool wear set page can be opened by pressing the fenubar and closed by pressing again (switch back to the Information page)

- F6：Start MPG Shift
  Function：Enable the MPG Shift function

- F7：
  - 5 key, 8 key：Work Record
    Function：Show the work record
  - 10+8 key：Graph Adjust
    Function：

- F8：
  - 5 key, 8 key：Clear Accum. Cycle Time
  - Function：Clear the accumulated machining time to zero
  - 10+8 key： File Manager
    Function：

- F9：
  - 5 key, 8 key：N/A
    Function：N/A
  - 10+8 key：Work Record
    Function：Show the work record

- F10：
  - 5 key, 8 key：N/A
    Function：N/A
  - 10+8 key：Clear Accum. Cycle Time
  - Function：Clear the accumulated machining time to zero

2.6.1 Screen Description

Machine State Monitor

- This area displays current machine state.
    - Absolute coordinate
    - Remaining distance to go during machining.
    - Feedrate
    - Spindle speed

Program Content Monitor

- This area displays the program content being executed.
- The yellow cursor indicates the block which is executing.

Machining Information Monitor

- Explanation
    - This area is overlapped with "Machining Parameter Setting".
    - Use "Parameter Set" to switch screens.

- Screen Description
    - a. G Code
        - Displays G code state in execution.

    - b. Run Time
        - Machining time for single workpiece.
        - The counting will reset when machining starts.

    - c. Accumulated Time
        - Accumulated machining time.

    - d. Override
        - G00 override
        - G01 override
        - Spindle override

    - e. Accumulated part
        - The total number of workpiece that have been machined.
        - Won't reset automatically.

- To manually reset, press "Parameter Set" to switch to Screen 2, and set "Accumulated Part" to 0.

   f. Part Count
   - Displays the number of workpiece completed for single NC program executed currently.
   - The number will reset when the executed NC program changes.

   g. Start Block
   - To set the start block of machining.
   - Operation Method
      - n: Specify the start block as the nth line. (Ex: 20)
      - L+n: Specify the start block as the nth line. (Ex: L20)
      - N+n: Search for the line where N+n is located and specify it as the start block. (Ex:N3)
      - T+n: Search for the line where T+n is located and specify it as the start block. (Ex:T01)
      - If the specified line number exceeds the maximum line number of NC program, the last line number will be specified as the start block.
      - Please refer to chapter "Program Preparation and Cycle Start" for the detail information of Start Block.

   h. Tool Number
   - 4 digit number
   - The first 2 are the tool number currently used.
   - The last 2 are the tool compensation number currently used.

Machining Parameter Display

- Explanation
   - This area overlaps with "Machining Information Monitor".
   - Use "Parameter Set" to switch between these displays.

- Screen Description
   a. Line number of break point
   - Show the line number (L) when the machining was interrupted last time.

   b. Serial number of break point
   - Show the serial number (N) when the machining was interrupted last time.

   c. Feed Rate
   - Set the feed rate of machining.
   - Can be set during machining.
   - The new feed rate will take effect when the pre-loaded blocks are all executed.

   d. Spindle speed
   - Set the speed of the spindle.
   - Can be set during machining and take effect instantly.

   e. Accumulated Count
   - Set the total number of workpiece machined.
   - It can only be reset to zero manually by the user.

   f. Part Count
   - Set the current number of workpiece machined.
   - The Part Count will be reset to zero when the executing NC file is changed.
   - When the specified M code is executed (set by Pr3804), Part Count will automatically add 1, and Run Time will reset.
   - When it reaches the number of required parts, the controller will automatically switch to B-Stop state.

g. Required Part Counting
- Set the upper limit of the workpiece quantity.
- When the required quantity is reached, a hint window will pop up and the controller will switch to B-Stop state.

Simulation Display

- Explanation
  - This area shows the tool path of the current machining file.
  - Refer to "Simulation Setting" for more details.
  - Use "Simulation Switch" to display or hide it.

2.6.2 Edit

- Path
  - 5 key, 8 key: F4 Monitor → F1 Edit
  - 10+8 key: G4 Monitor → F1 Edit

- Explanation
  - Load current machining file into the program editor and switch the screen to "Program".

- Note
  - The screen will be switched to "Program" if this button is pressed during machining, but it's unable to edit the current executing file.

2.6.3 Simulation Switch

- Path
  - 5 key, 8 key: F4 Monitor → F2 Simulation Switch
  - 10+8 key: G4 Monitor → F2 Simulation Switch

- Explanation
  - Show/Hide the simulation area.
  - The button "Graph Adjust" will only be enabled when simulation area is displayed on the screen.

2.6.4 MDI Input

- Path
  - 5 key, 8 key: F4 Monitor → F3 MDI Input
  - 10+8 key: G4 Monitor → F3 MDI Input

- Explanation
  - Edit the MDI executing program

- Operation Method
  - Switch to MDI mode.
  - An editing block will appear after selected "MDI Input"
  - Edit the program and press OK to save the content in MDIBLOCK.
  - Press Cycle Start to execute the MDI program.

- Note
  - This function is only enabled in MDI mode.

2.6.5 Param. Set

- Path
    - 5 key, 8 key: F4 Monitor → F4 Param. Set
    - 10+8 key: G4 Monitor → F4 Param. Set

- Explanation
    - Switch the screen between "Machining Parameter" and "Machining Information"

2.6.6 Tool Wear Set

- Path
    - 5 key, 8 key: F4 Monitor → F5 Tool Wear Set
    - 10+8 key: G4 Monitor → F5 Tool Wear Set

- Description
    - Set the tool wear value
    - Actual Tool length= Tool length +Tool wear

- Parameters
    - Tool Wear Set: Subtle tool length modification.

- Note
    - The tool wear set value will be cleared after the corresponding tool length is set.

2.6.7 Start MPG Shift

- Path
    - 5 key, 8 key：F4 Monitor → F6 Start MPG Shift
    - 10+8 key：G4 Monitor → F6 Start MPG Shift

- Explanation
    - The function is used to adjust the coordinate system by adding a shift with MPG to one of the blocks in the machining. The shifting amount will be retained for all the other blocks in the remaining process till the end of machining. It's usually applied when there is tool presetting and measuring error or tool wear, which cause too deep or ineffective cuts during the machining.
    [This function is provided for versions above 10.114]
    When rotating the MPG, the shifting amount will be added to the MPG coordinate and the program coordinate will remain the same. Press Esc to disable the function.
        - Applying Condition
            - In "MDI" or "Auto" mode

        - Operation
            1. Press "Start MPG Shift" in "MDI or "Auto" mode
            2. The MPG shift window will show up
            3. Switch the assigned MPG axis
            4. Rotate the MPG, the shifting amount will be shown in the MPG shift window

    Attention
    - The MPG shift window won't show up when there is another pop-up hint message window
    - Other enabling method
        - Besides enabling the function with HMI buttons, when in "MDI" or "Auto" mode, the rising-edge-trigger which makes R606.0 changes from 0 to 1 can also call out the MPG shift window.

    - Only MPG commands will be accepted, JOG and INJOG commands are invalid
    - When pressing "Start MPG Shift" in not "MDI" or not "Auto" mode.

- A reminder window will pop-up
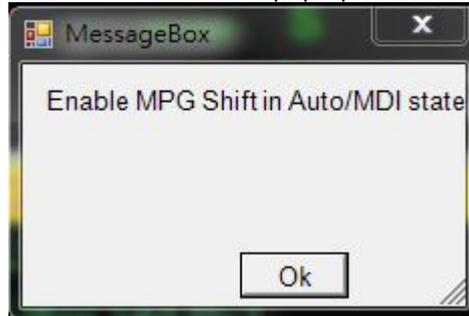
MPG shift window will be disabled when：
- Pressing ESC when the MPG shift window is shown
- The MPG window is the topmost window and R606.0 changes from 1 to 0
- "MDI" or "Auto" mode is switched to the other modes
- Alarms triggered

2.6.8 Work Record

- Path
  - 5 key: F4 Monitor → Next → F2 Work Record
  - 8 key: F4 Monitor → F7 Work Record
  - 10+8 key: G4 Monitor → F9 Work Record

- Explanation
  - Displaying Screen

File Name	N/A		File Comment	N/A
Require Part	1		Start DateTime	N/A
Part Count	0		Total Part Count	100
Cycle Time	0:00:00		Total Time	0:12:09

No.	User	Program	Start DateTime		Total Time	Total Part Count	Comment
1	DEFAULT	test	1/20/2020	10:14 AM	0:11:12	63	G91
2	DEFAULT	test	1/20/2020	9:58 AM	0:00:47	4	G91
3	U	test	1/16/2020	6:12 PM	0:00:03	2	G91
4	U	qqq	1/16/2020	6:07 PM	0:00:05	3	G91
5	U	test	1/16/2020	6:05 PM	0:00:17	5	
6	- End -						

- The current machining record can be checked with this table, the records can be exported to external storage devices. It can record up to 500 sets of machining files.
- Item in the table
  - No., User (authority), Program, Start DateTime, Total Time, Total Part Count and Comment.

- New saving timing for work record
  - When the accumulated part counts is not recorded and the following conditions are satisfied:
    1. Change machining NC program
       Whether reached the target part count or not, the current machining data will be recorded when changing the machining NC program.
    2. Reached the required parts count
       After reaching the required part count, the machining data will be recorded.
    3. Change operator (please refer to Account Management Application Manual. for operator login/logout)
       When the operator logs out, the machining data of current user will be recorded.

a. If no one has logged in before, the user name will be recorded as "DEFAULT".
b. The machining data will be recorded after every log out, but the required part count and the machined parts will not be affected.
c. Example:
    i. Set the required part count as 100 pieces. When worker 1 logged out after completed 30 pieces, a data set of worker 1 with 30 machined parts will be recorded.
    ii. Then worker 2 logged in and completed the remaining pieces (70), another data set of worker 2 with 70 machined pieces will be saved.
    iii. At last there will be 2 records on the table, "Worker 1, 30 parts" and "Worker 2, 70 parts".

- After new data is saved, the cursor will be on the first data row, which shows the latest recorded data.
- The working record of a machined part belongs to the worker who finishes the machining. For example, if the worker is changed during the machining. the machined part will belong to the latter worker.

2.6.9 Clear Accum. Cycle Time

- Path
    - 5 key：F4 Monitor → Next → F3 Clear Accum. Cycle Time
    - 8 key：F4 Monitor → F8 Clear Accum. Cycle Time
    - 10+8 key：G4 Monitor → F10 Clear Accum. Cycle Time

- Explanation
    - Clear the accumulated machining time.

2.6.10 Graph Adjust

- Path
    - 5 key: F4 Monitor → Next → F4 Graph Adjust
    - 8 key: F4 Monitor → Next → F1 Graph Adjust
    - 10+8 key: G4 Monitor → F7 Graph Adjust

- Explanation
    - Zoom in/out the simulation graph, enabled when the simulation area is displayed.

- Operating Introduction
    - Please refer to Simu. Setting

2.7 Maintain

- Path
    - 5 key, 8 key: F5 Maintain
    - 10+8 key: Maintain

- Explanation
    - This function page displays system alarms, network settings, fast diagnosis, PLC parameter settings, and controller software information.

2.7.1 Alarm

- Path
    - 5 key, 8 key: F5 Maintain → F1 Alarm

- 10+8 key: G5 Maintain → F1 Alarm
- Explanation
  - Display alarm messages on the screen.

Pending Alarm

- Path
  - 5 key, 8 key: F5 Maintain → F1 Alarm → F1 Pending Alarm
  - 10+8 key: G5 Maintain → F1 Alarm → Pending Alarm
- Explanation
  - Display current system alarm.

History Alarm

- Path
  - 5 key, 8 key: F5 Maintain → F1 Alarm → F2 History Alarm
  - 10+8 key: G5 Maintain → F1 Alarm → F2 History Alarm
- Explanation
  - Display the alarm history of the system.
- Attention
  - Some alarm types will not be recorded here, ex: MACRO alarm

Save Alarm

- Path
  - 5 key, 8 key: F5 Maintain → F1 Alarm → F3 Save Alarm
  - 10+8 key: G5 Maintain → F1 Alarm → F3 Save Alarm
- Explanation
  - Export the alarm contents of current screen to external device.
  - Ex: the screen is showing "Pending Alarm", the contents of current alarm can be exported to external device with this function.
  - Default name of the exported files:
    - Pending Alarm: Actalm.txt
    - History Alarm: Histalm.txt

2.7.2 Set Kernel Server

G54		TEST N0 L1	Set Kernel Server	2020/1/22	11:42:20	DEFAULT

IP Address Parameter   1 / 1

IP Address Setting   Specify an IP Address ▼

IP Address   Name Server Parameter

Subnet Mask   Primary DNS

Default Gateway   Primary WINS

Network DiskRemote Host Path

PC Name   Dir Name

User Name   Password

Net Status   Code：-1

Resource Shared

Shared Folder Path

●Ready   Auto   **Alarm**

- Path
    - 5 key, 8 key: F5 Maintain → F2 Set Kernel Server
    - 10+8 key: G5 Maintain → F2 Set Kernel Server

- Explanation
    - Set up the system network

- Parameters
    a. IP Address Setting
        - Select "Specify an IP Address" when using a crossover cable
        - Select "Obtain an IP Address via DHCP" when using a normal cable, "IP Address" and "Subnet Mask" can then be skipped

    b. IP Address
        - Enter the applicable IP address in the domain

    c. Subnet Mask
        - Enter the subnet mask of the IP address
        - Should be the same as the setting at PC end

    d. PC Name
        - Name of the connecting PC
        - Should be the same as the setting at PC end

    e. Dir Name
        - Name of the shared folder at PC end (should be the same as the setting at PC end).

    f. User Name
        - Can be skipped if no account and password is set to protect the folder shared by the Internet disk; if do, set the corresponding account and password.

    g. Password
        - Same situation as "User Name"

Set Kernel Server
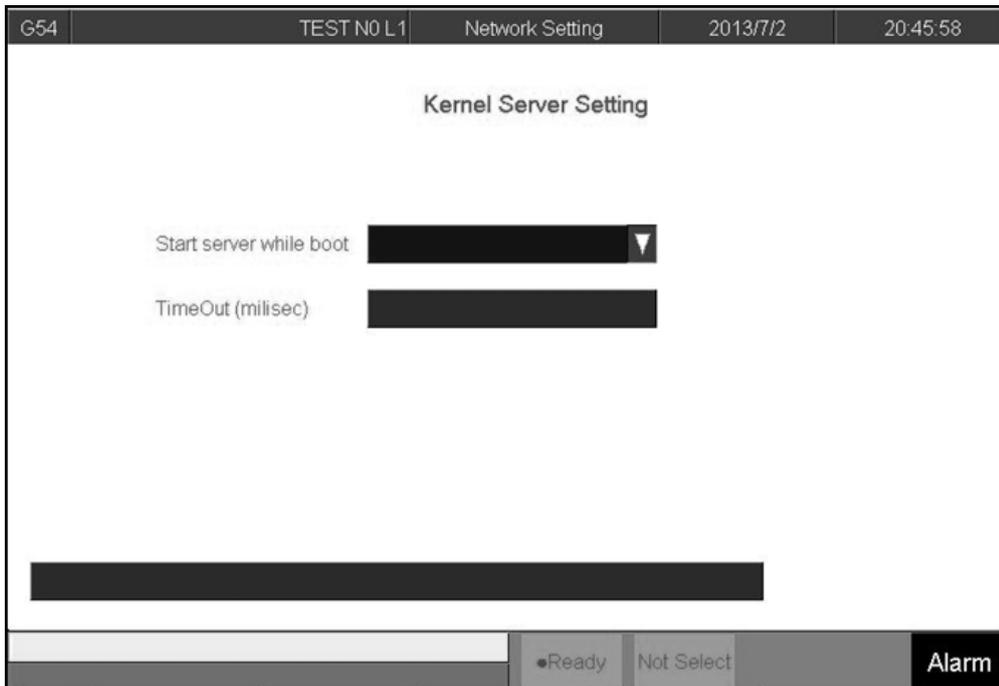
| G54 | | TEST N0 L1 | Network Setting | 2013/7/2 | 20:45:58 |

Kernel Server Setting

Start server while boot

TimeOut (milisec)

●Ready Not Select Alarm

- Path
    - F5 → Maintain → F2 Set Kernel Server → F5 Set Kernel Server

- Explanation
    - Set the related function of kernel server.

- Parameters
    - Start server while boot
        - Start the server after booting or not

    - Timeout(ms)
        - Set the acceptable timeout for connection failure

Start Server

- Path
    - F5 Maintain → F2 Set Kernel Server → F5 Set Kernel Server → F1 Start Server

- Explanation
    - Start the kernel server immediately.

2.7.3 Fast Diagnosis

- Path
    - 5 key, 8 key: F5 Maintain → F3 Fast Diagnosis
    - 10+8 key: G5 Maintain → F3 Fast Diagnosis

- Explanation
    - Display basic system information for diagnosis.

System Data

G54		TEST N0 L1	Fast Diag.	2020/1/22	13:36:38	DEFAULT

System Data

0.HMI Exe. Times	85145	23.Interpolation %	99	39.CPU Temp.(C)	-1
1.Motion Intrp. Times	852420	29.MPG Abs. Pos.	0	54.Intrp. Timeout Times	0
2.PLC San Times	852419	12.DA Voltage	2000	55. Fine Intrp. Timeout Times	0
3.I/O Scan Times	852419	44.Spindle Cmd.	1000	68.Axis Card Sync. Fail Times	0
4.Motion Intrp. Run Time	10000	28.Spindle Angle	0	69.Encoder Fdbk. Abnml. Time	0
5.PLC Scan Run Time	10000	52.SPD Index Counter	567652000	70.Encoder Fdbk. Abnml. Value	0
6.SRAM Write Times	101	13.Tapping Max. Error	0	320.Number of Interpreted Serial Blocks	0
77.HMI Free Memory	1787379712	14.Tapping Dyn. Error	0	324.Number of Interpolated Serial Blocks	0
78.HW. Free Memory	4294967295	79.Software Ver.	10.118.14A	Controller Model	21A

	●Ready	Auto	Alarm

- Path
    - 55 key, 8 key: F5 Maintain → F3 Fast Diagnosis → F1 System Data
    - 10+8 key: G5 Maintain → F3 Fast Diagnosis → F1 System Data

- Explanation
    - Display basic system and spindle information for diagnosis.

Axes Data

Controller Axes Data	X	Y	Z
G54 TEST N0 L1 Diagnostic 2020/1/22 13:47:59 DEFAULT			
Absolute Coord.	0	0	0
Machine Coord.	0	0	0
Abs. Position Command	0	0	0
Abs. Position Feedback	0	0	0
Following Error	0	0	0
Reasonable Following Error	0	0	0
Max. Following Error	11111	11111	11111
Index Counter	0	0	0
Axis Limit Offset	0	0	0
Motor Kp Estimation	0	0	0
Double Loop Abs. Pos. Feedback	0	0	0
Double Loop Index Counter	0	0	0
Double Loop Pos. Error	0	0	0

●Ready   Auto   Alarm

- Path
    - 5 key, 8 key: F5 Maintain → F3 Fast Diagnosis → F2 Axes Data
    - 10+8 key: G5 Maintain → F3 Fast Diagnosis → F2 Axes Data

- Explanation
    - Display basic axes information for diagnosis.

2.7.4 PLC Parameter

G54		TEST N0 L1	Offset/ Setting		2020/1/22		13:55:21			DEFAULT								
Index	Item	Value	F	E	D	C	B	A	9	8	7	6	5	4	3	2	1	0
3401	MLC R81	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3402	MLC R82	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3403	MLC R83	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3404	MLC R84	0x0001	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	1
3405	MLC R85	0x0014	0	0	0	0	0	0	0	0	0	0	0	1	0	1	0	0
3406	MLC R86	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3407	MLC R87	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3408	MLC R88	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3409	MLC R89	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3410	MLC R90	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3411	MLC R91	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3412	MLC R92	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3413	MLC R93	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3414	MLC R94	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3415	MLC R95	0x003C	0	0	0	0	0	0	0	0	0	0	1	1	1	1	0	0
3416	MLC R96	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0

Extension Parameter Bit R81.0 | Be modified only when Pr3401~ setting is between 0~65535

●Ready   Auto   **Alarm**

- Path
  - 5 key, 8 key: F5 Maintain → F4 PLC Param.
  - 10+8 key: G5 Maintain → F4 PLC Param.

- Explanation
  - Syntec controllers provide R81~R100 total 20 registers for machinery to use, each register can be divided into 16 Bits.
  - With the 20 register sets, machinery can provide users the control flag of specific PLC functions.
  - Extended parameters(Pr3401~Pr3420) with value out of the range of 0~65535 are not able to set its corresponding register in this screen.

- Operation Method
  a. Move the cursor with arrow keys【↑】【↓】【←】【→】
  b. Use【PageUp】【PageDown】to switch pages.
  c. Users can only input 0 or 1.
  d. Users can add comments to each Bit.
  e. Corresponding file name for the comments is ParamExt_RBit_(L).xml
  f. (L) = COM/CHT/CHS/language

2.7.5 System Setting

G54	TEST N0 L1	Parameter	2020/1/22	15:41:36	DEFAULT

Item	Value
Input/Display Unit(0:mm, 1:inch)	0
System Date Setting (YYYY/MM/DD)	2020 / 1 / 22
System Time Setting (HH:MM:SS)	15 : 41 : 37

●Ready　　Auto　　　　　　**Alarm**

(0~1)

- Path
    - 5 key, 8 key: F5 Maintain → F5 Sys. Setting
    - 10+8 key: G5 Maintain → F5 Sys. Setting

- Explanation
    - This page is used for system environment setting.

- Operation Method
    - Move the cursor with【↑】【↓】【←】【→】
    - Switch pages with【PageUp】【PageDown】

Input/Display Unit

- Explanation
    - Set the unit of input and display.
        - 0: SI Units
        - 1: Imperial Units

System Date/Time Setting

- Explanation
    - Date format: YYYY/MM/DD
        - YYYY: year
        - MM: month
        - DD: day

    - Time format: HH/MM/SS
        - HH: hour
        - MM: minute
        - SS: second

Set MMIConfig

- Path
  - 5 key, 8 key: F5 Maintain → F5 Sys. Setting → F8 Set MMIConfig
  - 10+8 key: G5 Maintain → F5 Sys. Setting → F8 Set MMIConfig

- Explanation
  - This page is used for other system environment setting. (9 pages in total)

- Operation Method
  - Move the cursor with【↑】【↓】【←】【→】
  - Switch pages with【PageUp】【PageDown】

Set Default

- Path
  - 5 key, 8 key: F5 Maintain → F5 Sys. Setting → F8 Set MMIConfig → F4 Set Default
  - 10+8 key: G5 Maintain → F5 Sys. Setting → F8 Set MMIConfig → F4 Set Default

- Explanation
  - Reset the value of where the cursor is located to default

- Attention
  - The setting only takes effect after reboot

Set Default All

- Path
  - 5 key, 8 key: F5 Maintain → F5 Sys. Setting → F8 Set MMIConfig → F5 Set Default All
  - 10+8 key: G5 Maintain → F5 Sys. Setting → F8 Set MMIConfig → F5 Set Default All

- Explanation
  - Reset all parameters to default.

- Note
  - The setting only takes effect after reboot

My Favor

G54		TEST N0 L1	Parameter	2020/1/22	16:21:48	DEFAULT

Default Setting   Set Kernel Server   Logout

invisible button   Fast Diag.   Authority Mgt.

File Manager   PLC Param.   About

Simulation   Sys. Setting

Start MPG Shift   Backup Sys.

Work Record   System Permissions

Alarm   Login

F 6 My Favorite : Default Setting   *Valid After Rebooting

●Ready   Auto   **Alarm**

- Path
  - 5 key, 8 key: F5 Maintain → F5 Sys. Setting → F8 Set MMIConfig → F6(F7, F8) My Favor
  - 10+8 key: G5 Maintain → F5 Sys. Setting → F8 Set MMIConfig → F6(F7, F8) My Favor

- Explanation
  - For 8 key system of ARM controller, provides user-define function to F6/F7/F8 of first fenubar layer

- Operation Method
  - Move the cursor with【↑】【↓】【←】【→】to choose the page to set as My Favor and press F1.

- Note
  - The setting only takes effect after reboot
  - Valid version: 11/21 series above 2.2.0
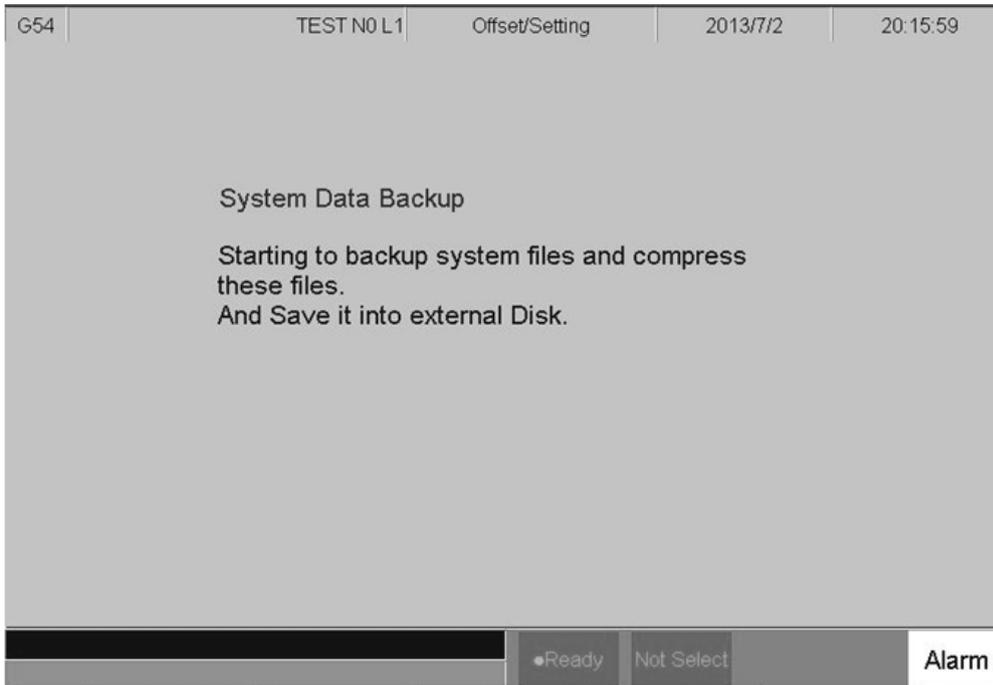
2.7.6 Backup System

- Path
  - 5 key, 8 key: F5 Maintain → Next → F1 Backup Sys.
  - 10+8 key: F5 Maintain → F6 Backup Sys.

- Explanation
  - Backup system data as a compressed file and saved into the assigned external device.

Online Backup

- Path
  - 5 key: F5 Maintain → Next → F1 Backup Sys. → F5 Online Backup
  - 8 key: F5 Maintain → F6 Backup Sys. → F8 Online Backup
  - 10+8 key: G5 Maintain → F6 Backup Sys. → F10 Online Backup

- Explanation
  - Upload the system backup data to the cloud for related technicians to help diagnose the problem.

2.7.7 System Permissions

- Path
  - 5 keys: F5 Maintain → Next → F3 Use Time Setting
  - 8 key: F5 Maintain → F8 Use Time Setting
  - 10+8 key: G5 Maintain → F10 Use Time Setting

- Explanation
  - Can be used to set the system use time
  - Please ask the original manufacturer for the setup password

2.7.8 Login

- Path
    - 5 key: F5 Maintain → Next → Next → F1 Login
    - 8 key: F5 Maintain → Next → F1 Login
    - 10+8 key: G5 Maintain → F7 Login

- Explanation
    - Enter user account and password to login the system.
    - Different permissions can be set for each user account
    - Permissions of each account can be set in "Authority Management" page.

2.7.9 Logout

- Path
    - 5 key: F5 Maintain → Next → Next → F2 Logout
    - 8 key: F5 Maintain → Next → F2 Logout
    - 10+8 key: G5 Maintain → F8 Logout

- Explanation
    - Logout the currently logged in account.

2.7.10 Forget Password

- Path
    - 5 key: F5 Maintain → Next → Next → F3 Forget Password
    - 8 key: F5 Maintain → Next → F3 Forget Password

- Explanation
    - For general users, please contact the administrator to reset the password.
    - For machinery or administrator, please contact Syntec to reset the password.

2.7.11 Authority Management

- Path
    - 5 key: F5 Maintain → Next → Next → F4 Authority Management
    - 8 key: F5 Maintain → Next → F4 Authority Management
    - 10+8 key: G5 Maintain → F9 Authority Management

- Explanation
    - Set user's access to each function.
    - Please refer to Account Management Application Manual. for further details

Change Password

- Path
    - 5 key: F5 Maintain → Next → Next → F4 Authority Management → F1 Change Password
    - 8 key: F5 Maintain → Next → F4 Authority Management → F1 Change Password
    - 10+8 key: G5 Maintain → F9 Authority Management → F1 Change Password

- Explanation
    - Change the password of the currently logged in user.
    - The original password is required

Default Authority

- Path
    - 5 key: F5 Maintain → Next → Next → F4 Authority Management → F2 Default Authority
    - 8 key: F5 Maintain → Next → F4 Authority Management → F2 Default Authority
    - 10+8 key: G5 Maintain → F9 Authority Management → F2 Default Authority

- Explanation
    - Set the authorities of each function when no user logs in.

User Authority

- Path
    - 5 key: F5 Maintain → Next → Next → F4 Authority Management → F3 User Authority
    - 8 key: F5 Maintain → Next → F4 Authority Management → F3 User Authority
    - 10+8 key: G5 Maintain → F9 Authority Management → F3 User Authority

- Explanation
    - The machinery or administrator can set the authorities of each user account.

2.7.12 Reset Password

- Path
    - 5 key: F5 Maintain → Next → Next → F5 Reset Password
    - 8 key: F5 Maintain → Next → F5 Reset Password
    - 10+8 key: G5 Maintain → Next → F1 Reset Password

- Explanation
    - Can be used to reset the password of the machinery

2.7.13 About

- Command:
    - F8 Maintain -> F8 About

- Function
    - Provide controller software version

3 Introduction of Control Panel

3.1 Operation Panel

- POWER ON
    - Turn on the controller power and ready for operation
- POWER OFF
    - Turn off the controller power after operation
- Emergency Stop
    - Press this button when the operator or the machine has any safety concern or unusual state, all electric control of the machine will be cut off. Except for the controller, all power including servo, spindle and machining fluid will be also cut off to ensure the safety of operator and machine.

- **Home Mode**

    - When CNC power is on, please apply home searching function in home mode

- **JOG Mode (Rapid JOG)**

    - Operators are able to control the axis movement with「X+/X-/Y+/Y-/Z+/Z-」on panel in this mode.

- **Incremental JOG (IN JOG)**

    - Operators are able to control axis movement with「X+/X-/Y+/Y-/Z+/Z-」on panel in this mode, the moving distance is decided by「x1/x10/x100」

- **MPG Mode**

    - Operators are able to move the machine by rotating the MPG (Manual Pulse Generator) in this mode

- **AUTO Mode**

    - Operators are able to auto execute NC file in this mode.

- **MDI Mode**

    - Operators can execute MDIblock programs instead of NC programs

- **MPG Simulation**

  - When this function is enabled in "AUTO" or "MDI" mode, the speed rate is depended on the MPG rotation speed. The machine will be moving faster as the MPG rotating faster; be stopped as the MPG stopped. This function is normally used to check NC files.

- **Single Block**

  - When this function is enabled in "AUTO" or "MDI" mode, the machine will hold (B-STOP) after finishing one block. Operators need to press "Cycle Start" to move on to the next block. This is function is normally used to check NC files.

- **Optional Stop**

  - Operators are able to decide whether to stop or not for the "M01" code in NC programs with this function

- **Optional Skip**

  - Operators are able to decide whether to skip or not for the "/" symbol with this function

- **Tool Magazine Clockwise Rotation**

  - Tool magazine clockwise rotation

- **Tool Magazine Counterclockwise Rotation**

  - Tool magazine counterclockwise rotation

- **Spindle Control**

	Spindle CW rotation
	Spindle stop
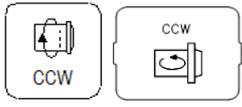
CCW	CCW	Spindle CCW rotation
-10% SPINDLE		Spindle rate deceleration: Spindle speed will decelerate by 10%.
100% SPINDLE		Spindle rate 100%: Spindle will rotate with 100% speed.
+10% SPINDLE		Spindle rate acceleration: Spindle speed will accelerate by 10%.

- **G01 Segments**
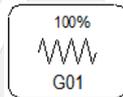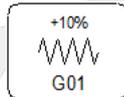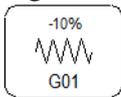
Decrease G01 rate by 10%   Increase G01 rate by 10%   G01 rate 100%

- **G00 Segments**
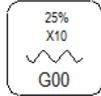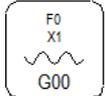
G00 rate x1   G00 rate x10   G00 rate x100   G00 rate x1000

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

- **Working LED**

  - Working LED On/Off

- **Chip Conveyor Machine Clockwise Rotation**

  - Switch chip conveyor machine to clockwise rotation

- **Chip Conveyor Machine Counterclockwise Rotation**

  - Switch chip conveyor machine to counterclockwise rotation

- **Cutting Fluid**

  [MANUAL icon]

  - Working liquid On/Off

- **Auto Cutting Fluid**

  [AUTO icon]
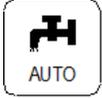
  - The cutting fluid M code of machining program will only be enabled after applying this function.

- **Blowing**

  [Blowing icon]

  - Blowing air On/Off

- **Expansion Key(User define)**

  | F1 | F2 | F3 | F4 | F5 | AUX1 | AUX2 | AUX3 | AUX |

  - Machinery expansion keys, combine with PLC ladder function.

- **Axis Direction & Axes CW/CCW Rotation & Fast Travel**

Z+	↑ Y	4+
←X	RAPID	X→
Z-	↓ Y	4-
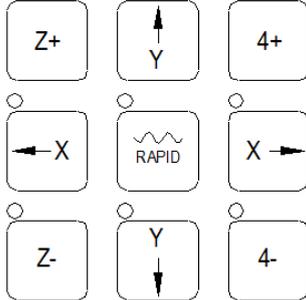
  - Control the axis movements, able to fast travel when pressing the rapid button with arrow keys at the same time.

3.2 Text Key

A ~ Z	:	Position keys, total 26 English character keys, used for command positions
0 ~ 9	:	Numerical keys, used for data input

DEL	:	Used to delete characters while editing programs
INS	:	Used to insert /replace characters while editing programs
SHIFT	:	Used to switch to another function of the key
SPACE	:	Space key, the corresponding position is blank
BACK SPACE	:	Used to modify the data when there are mistakes or errors
RESET	:	Used to reset the program settings
ENTER	:	Used to input current data
HELP	:	Provide function guide / message on screen after pressing
/	:	Used for optional skip blocks while editing programs
;	:	Used to represent the end of block in programs
.	:	Used for decimal fraction
PROG	:	Change to program edit page.
MONI	:	Change to monitor page.
TEACH	:	Change to customize teach page. (key disabled with no customize option)
INSERT	:	Used to insert /replace characters while editing programs

	:	Used to insert /replace characters while editing programs
	:	PAGE UP
	:	PAGE DOWN

"(", ")", "[", "]", "\|", "!", "&", "$", "#", "<", ">", "=", "%", "@", "*", ":", ";", "+", "-"

All above symbols are for "Program Edit" mode.
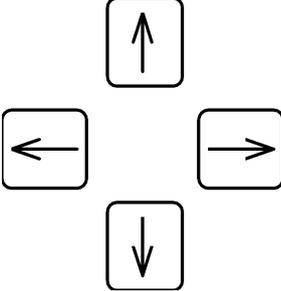
	:	Page Up /Page Down
	:	Cursor control

4 **Operation**

4.1 System State

Syntec controller displays different state under different situations, the following content will introduce the triggering conditions of each state.

(For multi-system interface, the controller will display the state of the system with smallest number.)

4.1.1 Not Ready

System cannot implement any operation under Not Ready state.

Conditions:

- Emergency stop button is pressed
- Serious alarm happens

System will switch to "Not Ready" state from any state and stop all actions to ensure the safety of users and machine.

4.1.2 Ready

System can implement all operations under Ready state.

Conditions:

- "Emergency stop" button is released and no alarm exists, system will switch state from "Not Ready" to "Ready".
- Under "Busy"/ "FeedHold"/ "B-stop" state, if user presses "Reset", system will switch to "Ready" state.

4.1.3 Busy

System is executing NC program.

Conditions:

- Under "Ready"/ "FeedHold"/ "B-stop" state, if Cycle Start function is triggered; system will switch to "Busy" state.

4.1.4 Feed Hold

System pauses the motion of the machine while executing NC program.

Conditions

- Press Stop (or FeedHold) button under "Busy" state, system will switch to "Feedhold" state.

Note:

- Spindle will keep rotating under "Feedhold" state.

4.1.5 B-Stop

System pauses the motion of the machine at the end of the block in NC program.

Conditions:

- Executing M00 under "Busy" statue.
- Single block function is triggered under "Busy" state.

Note:

- Under "B-Stop" state, spindle will keep rotating.

4.2 Machine Preparation

4.2.1 Manual Function

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

JOG

- Function:
    - Control the axis movement toward the selected direction
    - Can control more than one axis at the same time

- Condition:
    - System is on "Ready" state
    - 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%

Rapid JOG

- Function:
    - Control axis movement toward the selected direction with G00 speed
    - Can control more than one axis at the same time

- Condition:
    - System is on "Ready" state
    - JOG mode is selected

- Operation:
    - Press rapid key " ~ ", then press axis direction key (X+,X-,Y+,Y-,Z+…) 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.

INC JOG (incremental JOG)

- Function:
    - Control axis movement toward the selected direction with fixed distance(incremental distance)

- Condition:
    - System is on "Ready" state
    - INC JOG mode is selected

- Operation:
    - Press axis direction key (X+, X-,Y+,Y-,Z+…) and machine moves toward the selected direction.
    - 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
        - These percentage movements are shared with MPG mode

MPG

- Function:
    - Control axis movement toward the selected direction

- Condition:
    - System is on "Ready" state
    - MPG mode is selected

- Operation:
    - Press axis direction key (X+, X-,Y+,Y-,Z+…) or turn the direction switch on the manual pulse generator for the direction to move.
    - Turn the dial on manual pulse generator clockwise / counterclockwise for forward / reverse direction.
    - 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
        - These percentage movements are shared with INC JOG mode

4.2.2 Machining Process

Auto

- Description:
    - Execute "Machining Program"

- Condition:
    - System is on "Ready" state
    - AUTO mode is selected

- Operation:
    - Press "CYCLE START" button
    - System will machine the current machining program
    - System state will be switched from "Ready" to "Busy" when machining is started and back to "Ready" when finished

Single Block

- Description:
    - Execute each single block in program

- Condition:
    - System is on "Ready" state
    - Single block mode is selected

- Operation:
    - Press "CYCLE START" button
    - System will execute the current program in single block
    - System state will be switched from "Ready" to "Busy" when machining is started and back to "Ready" when finished

4.2.3 HOME

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

- Condition:
    - System is on "Ready" state
    - "Home" mode is selected

- Operation:
    - Release emergency stop button, CNC state 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.

- Warning:
    - Stroke protection in software is activated only after Homing is finished, so please make sure the machine does not run too fast before Homing.

4.3 Workpiece Preparation

4.3.1 Workpiece Coord. set

The setting of Workpiece Coordinate will be introduced on this section.

- Conditions
    - N/A

- Operation Procedure

a. Path: F3 Offset/Setting → F1 Workpiece Coord.

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

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

Relative

X 0.000
Y 0.000
Z 0.000
A 4.158

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

●Ready   Auto   **Alarm**

b. Move cursor to the required coordinate with ↑↓←→ and page ↑ , page ↓
c. Able to directly enter the target value
d. Able to enter the current machine coordinate with "Apply Mach. Coord."
e. Able to enter the current relative coordinate with "Apply Rel. Coord."
f. Able to enter the current auxiliary coordinate with "Apply Aux. Coord."
g. Press "Apply Mach. Coord. Inc." after entered the increasing value, the system will add the entered value to machine coordinate and fill in to workpiece coordinate automatically.
h. Press "Inc. Input" after entered the increasing value, the system will add the entered value to workpiece coordinate automatically.
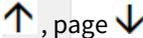
4.3.2 Manual Center Function

• Function:
Middle function is used to find the middle point of the workpiece for users to set the center of workpiece as the starting point of machining. Thus, the function requires to reach and contact the workpiece edge with detector or tool for the coordinate of workpiece edge, the controller will then automatically calculate the middle point. Users can later set the obtained middle point coordinate to the corresponding workpiece coordinate (G54~G59.9) as the starting point of machining program. Please notice that the auto middle function can only be applied when there is an auto tool presetting and measuring machine.

• Operation:
In 8-key system, user can enter the function by pressing F3 Offset/Setting → F1 Workpeice Coord. → F6

Middle Func.

| G54 | | N0 L1 | Offset/Setting | 2013/8/22 | 18:03:28 |

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

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

Machine Coord
Px1   0.000
Px2   0.000
Pxm   0.000
Py1   0.000
Py2   0.000
Pym   0.000

Machine
X   0.000
Y   0.000
Z   0.000
A   4.158

Relative
X   0.000
Y   0.000
Z   0.000
A   4.158

Aux. Coord
X   0.000
Y   0.000
Z   0.000

Manual Center Operation St

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

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

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

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

Step5：back to Workpiece screen to do Auxiliary

     point set

●Ready   Auto   Alarm

Fig. A

Manual 4Pts Center

- Operation
    a. Set Middle Function (first line) to mode 0, "Manual Center" will show in the top left corner of the screen.
    b. Set the Middle Function (second line) to mode 0, the Manual 4Pts Center page (Fig. A) will show up.
    c. Move the detector or tool to the left of workpiece (point Px1 in Fig. A)
    d. Press "Set Px1", the system will fill the current X axis machine coordinate into the Px1 column on the screen.
    e. Move the detector or tool to the right of workpiece (point Px2 in Fig. A)
    f. Press "Set Px2", the system will fill the current X axis machine coordinate into the Px2 column on the screen.
    g. Now can see "Pxm" is shown on the screen. It's the middle function result calculated by the controller.
    h. Move the detector or tool to the left of workpiece (point Py1 in Fig. A)
    i. Press "Set Py1", the system will fill the current Y axis machine coordinate into the Py1 column on the screen.
    j. Move the detector or tool to the left of workpiece (point Py2 in Fig. A)
    k. Press "Set Py2", the system will fill the current Y axis machine coordinate into the Py2 column on the screen.
    l. Now can see "Pym" is shown on the screen. It's the middle function result calculated by the controller.
    m. For "Auxiliary Coordinate" in the bottom right corner, the X, Y coordinate will now be the value of "Pxm" and "Pym".

n. Move the cursor to "Workpiece Coordinate P" and input the assigned workpiece coordinate number, press F5 "Auto Set Aux. Coord," (Valid version：10.116.34)
System confirmation window will show up [Apply Aux. Coord. to the assigned workpiece coordinate?], press [Yes] and the system will set the X, Y axis auxiliary coordinate value into the assigned workpiece coordinate.

o. It's able to return to the "Set Workpiece Coordinate" screen with left arrow key.

p. Move cursor to the X coordinate of the workpiece coordinate requires setting and press "Apply Aux. Coord,".

q. System confirmation window will show up [Apply Aux. Coord,?], press [Yes] and the system will set the X coordinate of "Aux. Coord." to the workpiece coordinate which the cursor is at.

r. Move cursor to the Y coordinate of the workpiece coordinate requires setting and press "Apply Aux. Coord.".

s. System Confirmation window will show up [Apply Aux. Coord.?], press [Yes] and the system will set the Y coordinate of "Aux. Coord," to the workpiece coordinate which the cursor is at.

Manual 3Pts Center
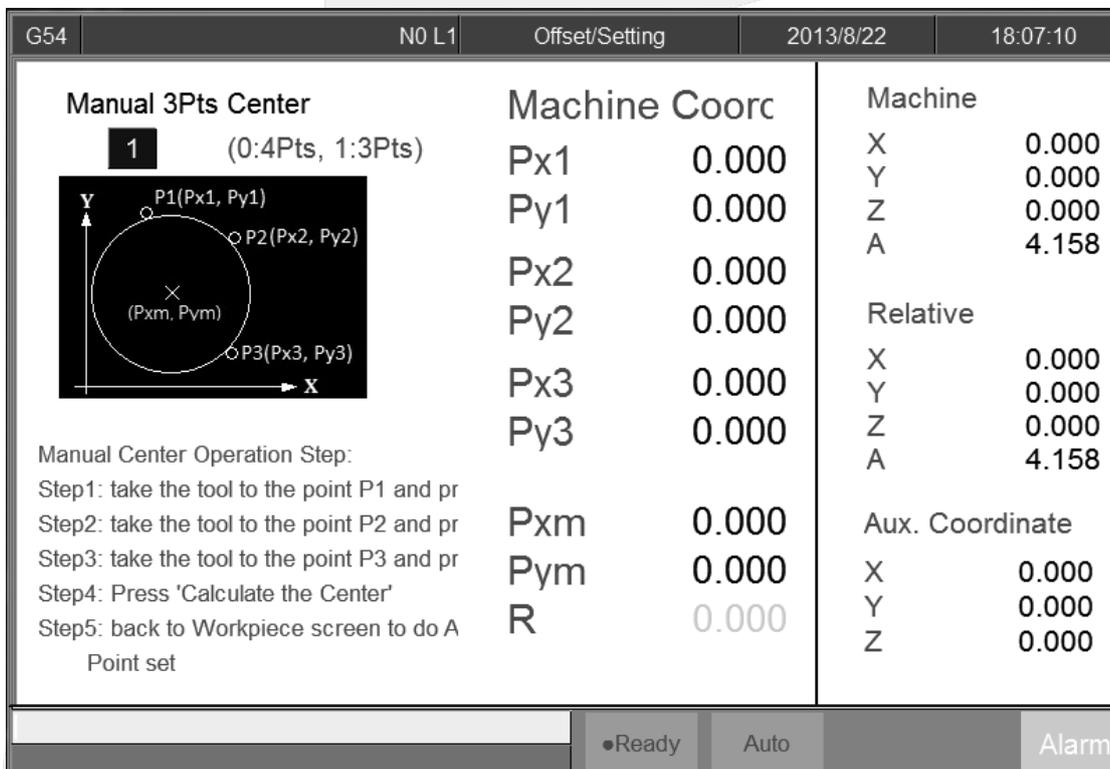
Fig. C

- Operation:
  a. Set Middle Function (first line) to mode 0, "Manual Center" will show in the top left corner of the screen.
  b. Set the Middle Function (second line) to mode 1, the Manual 3Pts Center page (Fig. A) will show up.
  c. Move the detector or tool to any point on the circumference of round workpiece (point P1 in Fig. C)
  d. Press "Set P1", the system will fill the current X and Y axis machine coordinate into the "Px1" and "Py1" columns on the screen.
  e. Move the detector or tool to any point (not the same point as P1) on the circumference of round workpiece (point P2 in Fig. C)

   f. Press "Set P2", the system will fill the current X and Y axis machine coordinate into the "Px2" and "Py2" columns on the screen.

   g. Move the detector or tool to any point (not the same point as P1 or P2) on the circumference of round workpiece (point P3 in Fig. C)

   h. Press "Set P3", the system will fill the current X and Y axis machine coordinate into the "Px3" and "Py3" columns on the screen.

   i. After finished the setting of 3 points on circumference, press "Calculate Center Coordinate". The system will calculate the machine coordinate of center and also the radius, the results will be shown in the "Pxm", "Pym" and "R" columns.

   j. For "Auxiliary Coordinate" in the bottom right corner, the X, Y coordinate will now be the value of "Pxm" and "Pym".

   k. If the settings are wrong or the 3 points are repeated, the alarm will be triggered.

   l. Move the cursor to "Workpiece Coordinate P" and input the assigned workpiece coordinate number, press F5 "Auto Set Aux. Coord," (Valid version：10.116.34)
System confirmation window will show up [Apply Aux. Coord. to the assigned workpiece coordinate?], press [Yes] and the system will set the X, Y axis auxiliary coordinate value into the assigned workpiece coordinate.

   m. It's able to return to the "Set Workpiece Coordinate" screen with left arrow key.

   n. Move cursor to the X coordinate of the workpiece coordinate requires setting and press "Apply Aux. Coord,".

   o. System confirmation window will show up [Apply Aux. Coord,?], press [Yes] and the system will set the X coordinate of "Aux. Coord." to the workpiece coordinate which the cursor is at.

   p. Move cursor to the Y coordinate of the workpiece coordinate requires setting and press "Apply Aux. Coord.".

   q. System Confirmation window will show up [Apply Aux. Coord.?], press [Yes] and the system will set the Y coordinate of "Aux. Coord," to the workpiece coordinate which the cursor is at.

4.3.3 Auto Center Function

OUTAuto Center

Fig. D

- Operation
  a. Set Middle Function (first line) to mode 1, "Auto Center" will show in the top left corner of the screen.
  b. Set the Middle Function (second line) to mode 0 (OUTAutoCenter), the OUTAutoCenter page (Fig. D) will show up.
  c. Input the actual length and width of the workpiece to "Workpiece Length I" and "Workpiece Width J" respectively.
  d. Set "Z axis coordinate P2" then bring the tool to P2 and press "Apply Z axis position". This Z axis position is set to ensure the measuring tool is able to reach the plane of workpiece edge when running the auto measuring.
  e. Set "Safe Distance H". Since the starting position of auto middle function is only approximately set at the center of workpiece, when the measuring tool moves from the starting point to the out border and begins to search for the workpiece edge after descending to Z axis position P2, the safe distance is required to avoid collision.
  f. Set "Feedrate F". The feedrate is the moving speed of measuring tool during auto middle function.
  g. After finished the settings above, move the tool to position P1, where is above the workpiece surface and near the workpiece center position. The tool should be moving above the workpiece and have no contact with the workpiece, the P1 position will then be the starting point of auto middle function.
  h. Please switch to auto mode and press "Start Auto Middle Function" to start.
  i. After auto middle function is complete, the system will automatically input the edge coordinate in X axis direction into "Px1" and "Px2"; Y axis direction into "Py1" and "Py2", the center coordinate will be filled into "Pxm" and "Pym".
  j. For "Auxiliary Coordinate", the X, Y coordinate will now be the value of "Pxm" and "Pym".
  k. Move the cursor to "Workpiece Coordinate P" and input the assigned workpiece coordinate number, press F4 "Auto Set Aux. Coord," (Valid version：10.116.34)
     System confirmation window will show up [Apply Aux. Coord. to the assigned workpiece coordinate?], press [Yes] and the system will set the X, Y axis auxiliary coordinate value into the assigned workpiece coordinate.
  l. It's able to return to the "Set Workpiece Coordinate" screen with left arrow key.
  m. Move cursor to the X coordinate of the workpiece coordinate requires setting and press "Apply Aux. Coord,".
  n. System confirmation window will show up [Apply Aux. Coord,?], press [Yes] and the system will set the X coordinate of "Aux. Coord." to the workpiece coordinate which the cursor is at.
  o. Move cursor to the Y coordinate of the workpiece coordinate requires setting and press "Apply Aux. Coord.".
  p. System Confirmation window will show up [Apply Aux. Coord.?], press [Yes] and the system will set the Y coordinate of "Aux. Coord," to the workpiece coordinate which the cursor is at.
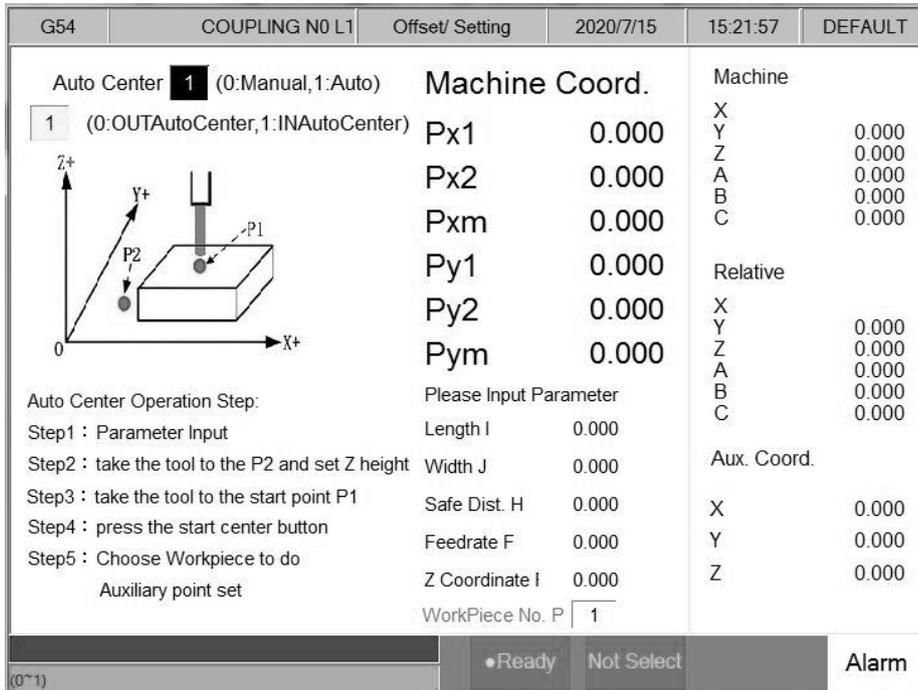
INAutoCenter

Fig. E

- Operation
    a. Set Middle Function (first line) to mode 1, "Auto Center" will show in the top left corner of the screen.
    b. Set the Middle Function (second line) to mode 1 (INAutoCenter), the INAutoCenter page (Fig. E) will show up.
    c. Input the actual length and width of the workpiece to "Workpiece Length I" and "Workpiece Width J" respectively.
    d. Set "Z axis coordinate P2" then bring the tool to P2 and press "Apply Z axis position". This Z axis position is set to ensure the measuring tool is able to reach the plane of workpiece edge when running the auto measuring.
    e. Set "Safe Distance H". Since the starting position of auto middle function is only approximately set at the center of workpiece, when the measuring tool moves from the starting point to the out border and begins to search for the workpiece edge after descending to Z axis position P2, the safe distance is required to avoid collision.
    f. Set "Feedrate F". The feedrate is the moving speed of measuring tool during auto middle function.
    g. After finished the settings above, move the tool to position P1, where is above the workpiece surface and near the workpiece center position. The tool should be moving above the workpiece and have no contact with the workpiece, the P1 position will then be the starting point of auto middle function.
    h. Please switch to auto mode and press "Start Auto Middle Function" to start.

4.4 Tool Preparation

Tool setting function will be introduced in this section

4.4.1 Tool Set

- Description
    - Able to manually set the compensation and wear value of tool length and cutter radius

- Condition
    - Enabled in both Manual and Auto modes
    - The increase limit of wear value is 1.0, alarm will be triggered if input excessive value
- Operation
    a. Press "Tool Set"
    b. Control the cursor with ↑↓←→ and switch pages with ↑ ↓ , move the cursor to the modifying content and change the input form
    c. Enter A, I or Z to change the input form
        - A：Absolute input (normally used for tool tip or tool length compensation)
        - I：Incremental input (normally used for cutter radius wear or tool length wear)
        - Z：Measurement input (normally used for lathe tool tip measure)
        - The input form will be displayed in the top right corner of the screen
- Tool Tip + Cutter Radius Wear = Actual G41/42 compensation value
- Tool Length Compensation + Tool Length Wear = Actual G43/44 compensation value
- Attention
    - Do not set the tool length or cutter radius compensation during machining

4.4.2 Manual Tool Tip Measure

- Introduction
    - This section will show how to manually execute tool tip measure, tool set and workpiece coordinate set for machining tool preparation.

- Explanation
    - When there is only one single tool for the machine, we often input the tool length and offset value to external shift instead of tool length.
    - When there are multiple tools for the machine, the tool length of each should be set individually. The setting method could be "Relative Tool Length" or "Absolute Tool Length".

- Condition
    - In manual mode

- Operation Procedure
    - Relative Tool Length
        i. Move the reference tool to the Z axis program zero point of workpiece (usually be the workpiece surface) with manual function, then input the current Z axis machine coordinate to G54 or other workpiece coordinate.
        ii. Clear the relative coordinate value
        iii. After changing to the tool which needs to be measured, move the tool to the same position as in step 1 with manual function.
        iv. The relative coordinate value at this moment will be the relative difference between reference tool and the measured tool, which is the tool length.
        v. Input the relative difference value to the corresponding tool number.
        vi. When the tool is worn out during machining and changed, the tool tip measure needs to be done again. If the reference plane of Z axis program zero point is gone, apply the standard tool and touch any point of the machine with its tool tip, then touch the same point with the new tool after clearing the relative coordinate value, thus the relative coordinate value at the moment will be the tool length.

    - Absolute Tool Length
        i. Move the tool which needs to be measured to a fixed standard position (usually be the surface of working table, or added the distance of tool outer radius as the gap) and input the current Z axis machine coordinate to the tool length.

ii. Repeat the step above to measure the tool length of each tool.
iii. Move the machining tool to the standard position as in step 1 and clear the relative coordinate value.
iv. Move the machining tool to the Z axis program zero point of workpiece (usually be the workpiece surface) with manual function, the current relative coordinate value of Z axis will then be the value of G54 or other workpiece coordinate
v. When the tool is worn out during machining and changed, the tool tip measure needs to be done again. Touch the standard position as in step 1 with the new tool, the current Z axis machine coordinate will be the tool length.

4.4.3 Tool Tip Measure

- Explanation
Tool tip measure function measures the tool tip Z axis coordinate value of each tool with the tool presetting and measuring machine on the device. Since the distance from tool presetting and measuring machine to workpiece datum plane is fixed, after the tool tip measure is done, the controller can automatically calculate the Z axis program home of tool for the corresponding workpiece. Please make sure tool presetting and measuring machine is installed before applying the function
There are 3 tool tip measure modes for now:
   a. Single tool Single workpiece: Select a workpiecec number, the tool length will be saved after tool tip measure. Then move the tool to workpiece datum plane and press "Z Delta Set", the value will be saved into external coordinate offset. Z axis program home can thus be calculated.
   b. Single tool Multi-workpiece: The tool length will be saved into external coordinate offset after tool tip measure. Then select a workpiece number and press "Z Delta Set", the value will be saved into workpiece coordinate. Z axis program home can thus be calculated.
   c. Multi-tool Multi-workpiece: Select a tool number, the tool length will be saved into tool length compensation table after tool tip measure. Then select a workpiece number and press "Z Delta Set", the value will be saved into workpiece coordinate. Z axis program home can thus be calculated.

- Path: Offset/Setting → Tool Tip Measure

- Screen Description:
   - Measuring mode: 1: Single tool Single workpiece; 2: Single tool Multi-workpiece; 3: Multi-tool Multi-workpiece; 4：5-axis
   - Workpiece No. P: Decide which coordinate will the tool tip measure result value be saved to (for mode 1)
   - Tool No. T: Set the measuring tool number, or fill the measured value into the assigned tool length compensation number (for mode 3)
   - Feedrate F: Set the feedrate and retract speed of each tool tip measure movement
   - Use reference: Set whether move to the reference point before tool tip measure or not, suggested to enable the function if the tool presetting and measuring machine has a fixed position.
      - Set 0: Disable the reference coordinate, run the tool tip measure from current location
      - Set 1: Enable the reference coordinate. Z axis will move back to the origin first and X, Y axis move to the reference point, then Z axis descends to the reference coordinate and start the tool tip measure.
   - Ref Coord. X: Set the X axis reference point, normally will be the X axis coordinate of tool presetting and measuring machine, need to be set with "Apply XY Ref. Coord.
   - Ref Coord. Y: Set the Y axis reference point, normally will be the Y axis coordinate of tool presetting and measuring machine, need to be set with "Apply XY Ref. Coord.
   - Ref Coord. Z: Set the Z axis starting point of tool tip measure. The Z axis will start moving in measuring speed after descended to this point. Can be set by entering the coordinate value directly or be set with "Apply Z Mach. Coord."

- Min. Z Mach. H: Set the lower limit of Z axis coordinate during tool tip measure, normally will be the position where the spindle won't collide with the tool presetting and measuring machine. Can be set by entering the coordinate value directly or be set with "Apply Z Mach. Coord.
- Safe Z after measure: The target location of Z axis after the tool tip measure is done. Delta Z set (delta set No.):
  - Mode 1: Move the tool tip to workpiece datum plane after the tool tip measure is done. The drop can be set as external shift with "Z Delta Set".
  - Mode 2 or Mode 3: Set the workpiece coordinate number and move the tool tip to workpiece datum plane after the tool tip measure is done. The drop can be set to the corresponding workpiece coordinate with "Z Delta Set".
- Gauge air blow (On/Off): Before tool tip measure starts, it sometimes requires air blow to clean the surface of tool presetting and measuring machine, this function provides setting the M code of gauge air blow. The enabling timing should between Z axis zero point to Ref Coord. Z.

- Action Explain
  a. Z axis moving to machine zero point in G00 speed (reference point enabled)
  b. X, Y axis moving to the reference point in G00 speed (reference point enabled)
  c. If the gauge air blow M code is set, the air low starts now
  d. Z axis moving to Ref Coor. Z in the speed of 7000mm/min (reference point enabled)
  e. f the gauge air blow M code is set, the air blow stops now
  f. Z axis moving towards the direction of Min. Z Mach. with the set feedrate F. The action will be stopped immediately when Z axis reaches the tool presetting and measuring machine during the descending process. Please note that if Min. Z Mach is set incorrect, Z axis might be moving upwards
  g. Z axis climbs 5mm with the set feedrate
  h. Z axis second descending, descends 6mm in the speed of 50mm/min. The action will be stopped immediately when Z axis reaches the tool presetting and measuring machine during the descending process, the Z axis machine coordinate of the moment will be recorded
  i. Z axis climbs 3mm with the set feedrate

j. Z axis third descending, descends 4mm in the speed of 50mm/min. The action will be stopped immediately when Z axis reaches the tool presetting and measuring machine during the descending process, the Z axis machine coordinate of the moment will be recorded

k. If the difference between two recorded Z axis coordinate is within 0.01mm, the average value will be calculated and input to the assigned workpiece coordinate or tool length compensation table

l. Z axis climbs to safe Z after measure in G00 speed

- Limitation and Protection
    a. During the first descending, if Z axis reaches the Min. Z Mach. before reaching the tool presetting and measuring machine, alarm **MACRO-330_Min. Z Mach. setting error** will be sent by the controller
    b. During the second descending, if Z axis does not reach the tool presetting and measuring machine within 6mm, alarm **MACRO-331_Tool Tip Measure Error** will be sent by the controller
    c. During the third descending, if Z axis does not reach the tool presetting and measuring machine within 4mm, alarm **MACRO-331_Tool Tip Measure Error** will be sent by the controller
    d. If the difference between two recorded coordinate value exceeds 0.01 mm, alarm **MACRO-332_2nd and 3rd Measuring difference exceeds setting range** will be sent by the controller
    e. If Z axis reaches the tool presetting and measuring machine before reaching the reference point, alarm **MACRO-333_Z axis staring point error** will be sent by the controller
    f. If Delta Z set is set inappropriately, alarm **MACRO-401_Incorrect workpiece coordinate setting** will be sent by the controller
    g. If tool number is set inappropriately. alarm **MACRO-402_Incorrect tool setting** will be sent by the controller
    h. If measuring mode is set over 3, alarm **MACRO-403_Incorrect tool tip measure mode** will be sent by the controller

Operation of Single Tool Single Workpiece

- Explanation
    a. Set the "Mode" in top left corner of tool tip measure screen to 1

b. Set the "WorkPiece No. P" to the workpiece coordinate which the workpiece belongs to

c. Workpiece coordinate number P: 1: G54, 2: G55, 3: G56, 4: G57, 5: G58, 6: G59, 7: G59.1 …

d. Set the feedrate and retract speed of tool tip measure with "Feedrate F"

e. Skip to step 6 and set Ref Coord. XY if the tool presetting and measuring machine has a fixed position on the machine. If the current location is at the location of tool presetting and measuring machine, please set "Use reference" to 0.

f. Skip this step if step 5 is already done. Set "Use reference" to 1 and move the XY axes, align the tool with the center of tool presetting and measuring machine and press "Apply XY Ref Coord.", choose "Yes" for the confirmation window, the current machine coordinate will be automatically filled into "Ref Coord. X" and "Ref Coord. Y"

g. Move the cursor to "Ref Coord. Z" and set the Z axis starting location of tool tip measure, or move Z axis to the stating point and press "Apply Z Mach. Coord." to fill the Z axis coordinate into "Ref Coord. Z"

h. Move the cursor to "Min. Z Mach. H" and directly set the lower limit of Z axis during tool tip measure, or set with "Apply Z Mach. Coord."

i. After finished the settings above, please switch the mode to auto mode and press "Start" to begin

j. If any emergencies happened during the process, press "Reset", emergency stop or reset to stop the process

k. After tool tip measure is done, the Z axis coordinate of the point where the tool reaches tool presetting and measuring machine will be saved into auxiliary coordinate and the assigned workpiece coordinate automatically

l. Move the tool tip to the workpiece surface manually and press "Z Delta Set", the drop between tool presetting and measuring machine and workpiece surface will be filled into "Delta Z set" and external shift.

m. Tool tip measure of Single Tool Single Workpiece is done

Operation of Single Tool Multi-Workpiece

- Explanation

a. Set the "Mode" in top left corner of tool tip measure screen to 2
b. Set the feedrate and retract speed of tool tip measure with "Feedrate F"
c. Skip to step 4 and set Ref Coord. XY if the tool presetting and measuring machine has a fixed position on the machine. If the current location is at the location of tool presetting and measuring machine, please set "Use reference" to 0.
d. Skip this step if step 3 is already done. Set "Use reference" to 1 and move the XY axes, align the tool with the center of tool presetting and measuring machine and press "Apply XY Ref Coord.", choose "Yes" for the confirmation window, the current machine coordinate will be automatically filled into "Ref Coord. X" and "Ref Coord. Y"
e. Move the cursor to "Ref Coord. Z" and set the Z axis starting location of tool tip measure, or move Z axis to the stating point and press "Apply Z Mach. Coord." to fill the Z axis coordinate into "Ref Coord. Z"
f. Move the cursor to "Min. Z Mach. H" and directly set the lower limit of Z axis during tool tip measure, or set with "Apply Z Mach. Coord."
g. After finished the settings above, please switch the mode to auto mode and press "Start" to begin
h. If any emergencies happened during the process, press "Reset", emergency stop or reset to stop the process
i. After tool tip measure is done, the Z axis coordinate of the point where the tool reaches tool presetting and measuring machine will be saved into auxiliary coordinate and the assigned workpiece coordinate automatically
j. Move the tool tip to the workpiece surface manually and press "Z Delta Set", the drop between tool presetting and measuring machine and workpiece surface will be filled into "Delta Z set" and external shift
k. WorkPiece No. : 1: G54, 2: G55, 3：G56, 4: G57, 5: G58, 6:G59, 7: G59.1 …
l. If there are other workpieces require setting, repeat step 10 to complete tool tip measure of Single Tool Mullti-Workpiece

Operation of Multi-Tool Multi-Workpiece

- Explanation
    a. Set the "Mode" in top left corner of tool tip measure screen to 3
    b. Enter the measuring tool number in "Tool No. T"
    c. Set the feedrate and retract speed of tool tip measure with "Feedrate F"
    d. Skip to step 5 and set Ref Coord. XY if the tool presetting and measuring machine has a fixed position on the machine. If the current location is at the location of tool presetting and measuring machine, please set "Use reference" to 0
    e. Skip this step if step 4 is already done. Set "Use reference" to 1 and move the XY axes, align the tool with the center of tool presetting and measuring machine and press "Apply XY Ref Coord.", choose "Yes" for the confirmation window, the current machine coordinate will be automatically filled into "Ref Coord. X" and "Ref Coord. Y"
    f. Move the cursor to "Ref Coord. Z" and set the Z axis starting location of tool tip measure, or move Z axis to the stating point and press "Apply Z Mach. Coord." to fill the Z axis coordinate into "Ref Coord. Z"
    g. Move the cursor to "Min. Z Mach. H" and directly set the lower limit of Z axis during tool tip measure, or set with "Apply Z Mach. Coord."
    h. After finished the settings above, please switch the mode to auto mode and press "Start" to begin
    i. If any emergencies happened during the process, press "Reset", emergency stop or reset to stop the process
    j. After tool tip measure is done, the Z axis coordinate of the point where the tool reaches tool presetting and measuring machine will be saved into auxiliary coordinate and the assigned workpiece coordinate automatically
    k. Move the tool tip to the workpiece surface manually and press "Z Delta Set", the drop between tool presetting and measuring machine and workpiece surface will be filled into "Delta Z set" and external shift
    l. WorkPiece No. : 1: G54, 2: G55, 3：G56, 4: G57, 5: G58, 6:G59, 7: G59.1 …
    m. If there are other workpieces require setting, repeat step 2 to step 12 to complete tool tip measure of Single Tool Mullti-Workpiece

Operation of 5-Axis Tool (Need to turn on Option12 or Option13)

There are 2 steps : (1) Operation of tool presetting and measuring machine coordinate (2) Tool tip measure
Unless the machine or the tool presetting and measuring machine has changed, users only need to conduct (2) after changing tools.

(1) Operation of tool presetting and measuring machine coordinate :

1. Set the "Mode" to 4 on the tool tip measure screen
2. Fill in "Tool No. T"
3. Fill in "Feedrate F"
4. Skip to step 5 and set Ref Coord. XY if the tool presetting and measuring machine has a fixed position on the machine. If the current location is at the location of tool presetting and measuring machine, please set "Use reference" to 0
5. Skip this step if step 4 is already done. Set "Use reference" to 1 and move the XY axes, align the tool with the center of tool presetting and measuring machine and press "Apply XY Ref Coord.", choose "Yes" for the confirmation window, the current machine coordinate will be automatically filled into "Ref Coord. X" and "Ref Coord. Y"
6. Move the cursor to "Ref Coord. Z" and set the Z axis starting location of tool tip measure, or move Z axis to the stating point and press "Apply Z Mach. Coord." to fill the Z axis coordinate into "Ref Coord. Z"
7. Move the cursor to "Min. Z Mach. H" and directly set the lower limit of Z axis during tool tip measure, or set with "Apply Z Mach. Coord."
8. Switch the mode to auto mode. Before installing tools, press F7 to view the operation of tool presetting and measuring machine coordinate

9. If any emergencies happened during the process, press "Reset", emergency stop or reset to stop the process
10. Tool tip measure result will be automatically filled into Z axis machine coordinate of the tool presetting and measuring machine.

G54		1 N0 L1	Offset/ Setting	2020/7/15	15:48:04	Admin

Auto Tool Function

Mode　　　4

1:Single tool Single workpiece
2:Single tool Multi-workpiece
3:Multi-tool Multi-workpiece
4:Five axes

Tool No. T	0
Feedrate F	0.000
Use Reference	0
Ref Coord. X	0.000
Ref Coord. Y	0.000
Ref Coord. Z	0.000
Min. Z Mach. H	0.000
Safe Z After Measure	0.000
Gauge Z Mach. Coord.	0.000

1.Please choose using Ref. Point or taking tool tip to upper of measurement
2.Set all measure parameters
3.Press F7 Gauge Z Coord. Teach Start(maker);Press F1 Measure Start(user)
Remark:Please execute F7 again, if spindle side mechanism is adjusted

Gauge Air Blow: ON: M　　　0　　OFF: M　　　0

Machine
X	0.000
Y	0.000
Z	0.000
A	0.000
C	0.000

Relative
X	0.000
Y	0.000
Z	0.000
A	0.000
C	0.000

Aux. Coord.
X	0.000
Y	0.000
Z	0.000

| ●Ready | Not Select | | Alarm |

(2) Tool tip measure :

1. Please make sure (1) is well known
2. Set the "Mode" to 4 on the tool tip measure screen
3. Please refer to step 2 to step 7 in (1)
4. Switch the mode to auto mode and press F1 to launch tool tip measure. The system will alter the tool into the measuring tool automatically and start to measure
5. If any emergencies happened during the process, press "Reset", emergency stop or reset to stop the process
6. After the measurement, tool length will be saved into the tool length compensation form

4.4.4 Tool No.

This section will introduce Tool No. setting and Tool Life Management functions

- Explanation
    - When applying arm type tool magazines, the relation between tool No. and MG should be confirmed. Thus it requires the correct tool No. to operate the tool change properly.

- Condition
    - Applicable in both auto and manual mode

- Operation Procedure
    - a. Click "Tool No."
    - b. The screen shows the MG and Tool No. T Table

c. MG5—T = 7 means tool No.7 is in MG No.5 (as shown below)

| G54 | | 1 N0 L1 | Offset/ Setting | 2020/7/15 | 16:23:00 | DEFAULT |

MG And Tool No. T Table

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

Turrent No   0

Spindle No   0

●Ready   Not Select   Alarm

d. When tool change action is executed, this table will be updated according to the current relation between tool No, and MG.

- Modification Timing
  - The default setting of the machinery
  - When the tools are messed up, set the tool No. after manually checked the relation between tool No. and MG.

4.4.5 Tool Manager Function

- Explanation

- Provides tool life management screen and display. Since the actual function requires PLC settings, please contact the machinery for further specification details

| G54 | | | 1 N0 L1 | Offset/ Setting | 2020/7/15 | 17:14:18 | DEFAULT |

Tool Manager Function

No	Turret	Group	Information	Current Life	Max. Life	Notice	Status
No	0	0	U N C -	0	0	0	Not Managed
No	0	0	U N C -	0	0	0	Not Managed
No	0	0	U N C -	0	0	0	Not Managed
No	0	0	U N C -	0	0	0	Not Managed
No	0	0	U N C -	0	0	0	Not Managed
No	0	0	U N C -	0	0	0	Not Managed
No	0	0	U N C -	0	0	0	Not Managed
No	0	0	U N C -	0	0	0	Not Managed
No	0	0	U N C -	0	0	0	Not Managed
No	0	0	U N C -	0	0	0	Not Managed
No	0	0	U N C -	0	0	0	Not Managed
No	0	0	U N C -	0	0	0	Not Managed

| | | | •Ready | Not Select | | | Alarm |

- Condition
  - Applicable in both auto and manual modes

- Operation Procedure
  - Click "Tool Life Manag." (set enable/disable by Pr.3228)

- Description
  a. Turret
     - The current MG No. of the tool

  b. Group
     - The same kind of tools will be in the same group. If the first tool of the group is locked or reached the maximum life, it'll be skipped when operating the tool change with T code and the second tool will be selected.

  c. Information
     - L – Lock / U – Unlock
       - The tool will be disabled in lock mode and be skipped for T code tool change

     - B – Big Radius Tool / N – Normal Tool
       - The MG next to big radius tool should be empty (only for display)

     - T – Use Time / C – Use Count
       - Determines displaying unit of current life, maximum life, and notice to be time or count.

     - R – Calculation Enabled / – Calculation Disabled
       - Whether the tool is using tool management function

  d. Current Life
     - Current state of the tool

e. Max. Life
- Maximum life of the tool

f. Notice
- Notification will be sent when the tool life exceeds the value of Notice

g. Status
- (0)Not managed：Invalid setting value
- (1)Not used：Tool life = 0
- (2)Usable： 0 < Tool Life < Notice
- (3)End notice：Notice < Tool Life < Max. Life
- (4)Life end：Max. Life < Tool Life
- (5)Broken

4.5 Program Preparation and Execute Machining

4.5.1 Select NC Program

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

4.5.2 Simu. Setting

SYNTEC's controller provides simulation program, after editing machining program, users can easily simulate the path of machining process. This feature also includes checking function that help user quickly find out the syntax error in machining program or unreasonable actions. We suggest user make good use of this feature after programming.

- Condition
    - Except single block mode
- Operation
    - In the "File Management" page, select the program you want to edit after programming, and press F7 "Simulation"
    - Screen will switch to the page "graphic simulation" and scan the content of the program
- Description
    a. In simulation screen
        - The solid line represents the cutting path
        - The dashed line represents the moving path

b. During the checking, if there is any syntax or error of content, they will be displayed on the screen with corresponding line number
c. F1(~~S~~tep): Check NC files once a line
d. F2(Continue): Simulate the machining process
e. F3(Zoom): To zoom in/out the simulation. Users can use the arrow key " ← ", "↑", " → ", "↓" to move the frame to the expected area, use "PageUp" "PageDn" to zoom in/out this area. After selecting zoom scales, press "enter" to finish.
f. F5(Simu. Setting): Set simulation parameter

4.5.3 Machining Test

MPG Simulation

This chapter is going to introduce how to do machining test with MPG

- Condition:
  - Only for single block(MDI) and auto mode

- Operation:
  a. Press MPG simulation button on operation panel
  b. "Cycle Start"
  c. Turn MPG dial to execute machining
  d. If MPG is turned in CW direction, program will be run from current NC line down to the next NC line
  e. If MPG is turned in CCW direction, program will be run from current NC line up to the last NC line

- Confirm: User could check whether MPG simulation is activated successfully or not with two methods
  - No "Cycle Start."
    After activate MPG function, the G01 percentage in "Monitor" page is zero/nonzero while user does not/does turning MPG dial
  - Already "Cycle Start."
    After activate MPG function, machine starts decelerating immediately until 0, unless user turn MPG dial or cancel MPG function.

Single Block

This chapter is going to introduce how to do machining test with single block

- Condition:
  - Only for single block(MDI) and auto mode

- Operation:
  a. Press single block button on operation panel
  b. "Cycle Start"
  c. After executing a single block, machine decelerates to 0, state of system is switched to B-stop
  d. Press "CYCLE START" again
  e. After completing next single block in NC file, state of system is in B-stop again

MPG Shift

This chapter is going to introduce the MPG shift function

- Condition
  - Only for single block(MDI) and auto mode

- Operation
  a. Press "MPG Shift – Enable" in "MDI" or "auto" mode

b. MPG shift widow will be displayed
c. Switch to MPG axis assign
d. Rotate the MPG and the value will be shown in the MPG shift window

- Confirmation
  - Switch to coordinate system screen and make sure the value in MPG shift column is correct.

- Related Parameters
  a. Pr.3201 set lathe, MPG shift function can't be enabled when the parameter is not 0
  b. When the path of axes Pr.701~716 has axes belonging to different path, the MPG shift function can;t be enabled.

- Notifications
  - Other enabling method
    - Besides enabling with the MDI button, the MPG shift window can also be called by the rising-edge trigger which makes R606.0 changes to 1 from 0 in "MDI" or "Auto" mode.
    - Only accepts MPG command, JOG and INJOG commands are invalid

  - Press "MPG Shift – Enable" in non-"MDI" or non-""Auto" mode"
    - The reminder window will pop up
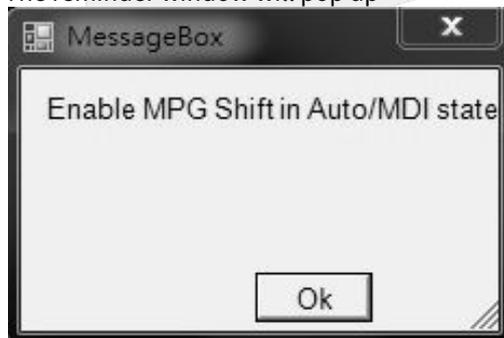
  - Close the MPG shift window：
    - Press ESC when the window is displayed
    - When R606.0 turns to 0 from 1
    - Being switched to other modes except for "MDI" or "Auto" mode.

- Able to be operated when the machine is locked
  - Dismiss timing
    i. Save the MPG shift coordinate when power-on
    ii. Skip clear after changing the coordinate (G54/G55)
    iii. Skip clear while changing the machining process
    iv. Skip clear after home searching
    v. Skip clear after G28/G29/G30 return to reference point actions

- Limitation
  - When MPG shift function is enabled under MPG simulation, the commands will be sent to MPG shift function when rotating the MPG. After the MPG shift function is done, the commands will then be sent to MPG simulation function.

4.5.4 Mid-Program Restart

Timing to use

1. Tool fractures during machining process.
2. The power is cut off abnormally and rebooting the system is necessary.
3. User want to execute part of machining program separately.

4.5.5 **Related Setting**

Controller Parameter Pr3851：

> The way to return fractured tool (0 or 888800 or 999900 or Customize Macro)

Operation

1. Push "Reset" button to clear alarm and "Busy" state.
2. After tool fractures, move tool aside in MPG mode.
   After replacing tool, in MPG mode, move tool to the neighborhood of X-axis and Y-axis position which is specified by the first line in machining program.
3. Switch to 'Machining Execution' screen. Input program Serial number (N) and Row number (L), which is the position of program restarting. (Note two)
4. Start MPG simulation function, and "Cycle Start". Slowly rotate MPG and check whether the starting point of machining is correct or not.
5. If user need to slightly adjust tool length, please enter 'Tool Setting' and change 'Tool length Wear'. Wear amount is able to be adjusted during machining, and it is enabled immediately.
6. If user need to adjust Machining Speed (F) and Spindle Speed (S), please enter 'Machine Parameter Setting'. (Note Four)
7. If every movement is normal, please cancel MPG simulation setting, and system will return to normal machining.

Note one：

If user want to use the broken tool return function, please use absolute coordinate (G90) to do program coding, and it will be easier to check.

Note Two：

How to assign middle starting position in a program(N/L)

EX：

- Input N10 means executing program from N10. System will automatically translate N10 into row number and display on screen.
- Input L10 means executing from the tenth row.
- Input 10 (the same as L10) means executing from the tenth row.

Note Three：

Press 'Tool Setting' and change tool wearing value (be able change it at any time during machining).

Note Four：

There must be corresponding spindle speed and machining speed to start execution from the middle. User can input spindle speed and machining speed, which should be same as that at row number of restarting program, again in screen 'Machine Parameter Setting.'

4.5.6 Fractured Tool Return Function Precautions

1. Pr3851 Tool break handle program No: number of fractured tool process program (will be effective after 9.118 version)
2. The serial number for restarting in the middle of program must be bigger than zero.
3. After setting serial number and "Cycle Start"（executing machining）, Start Block will show "1."

4. If Pr3851 equals to "888800", user shouldn't click Cycle Start during scanning. If so, there will be alarm that file does not exist.
5. Pr3851 = "888800" do not support the framework of dipole.

4.5.7 Machining Monitor

This section will introduce the function of workpiece counting and work record.

Part Count Manager

- Condition:
    - None

- Description:
    a. Accumulated Part Count
        - The total workpiece number machined by CNC.

    b. Required Part Count
        - The specified number from a machining program. If system runs machining program with M99, CNC stops once the required number of part is reached and informs user that the required number is reached.

    c. Part Count
        - Once CNC runs machining program with M99, this number will be accumulated until reset
        - Condition to reset part count(clear to 0)
            1. Change machining files
            2. Required part count is reached(clear to 0 while "Cycle Start" again)
            3. Modify the required part count, and the required part count is smaller than part count(clear to 0 while "Cycle Start" again)

Work Record Function

- Condition:
    - None

- Description:
    - Once CNC continues machining when it meets M99, work record function will automatically record it.

- Recording timing:
    - Required part count is reached
    - If 0 < part count < required part count, at the time of changing machining files
    - At the moment of modifying the required part count which must be smaller than the part count.

4.6 Alarm Processing

To avoid wrong operation causes danger to human and damage to machine, the system and PLC are equipped with much of protection. When conditions are reached, system issue warning or alarm to user. This section introduces how to inspect and troubleshooting alarm.

4.6.1 Emergency Stop

Machine failure or unexpected movements may cause unsafety for human and machine. Pressing emergency stop button, you can immediately stop the machine. The button is locked after it is pressed down. There may be

difference between the method of unlocking button of emergency stop. Normally, turning the button should be able to unlock it.

Because emergency stop button would stop machine movement, please trouble shooting before you unlock the button.

4.6.2 Alarm Display

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

Pending Alarm

- The current state of system alarm
- Once an alarm occurs, an alarming window will pop up on screen with current alarm content
- Press ESC to jump out of that window
- If the alarm is not removed, press reset button, and alarm window will be displayed again
- Press F5 "Maintain" to display pending alarm contents automatically

History Alarm

Accessing to this page enables user to see all system alarms which have occurred to find out the causes.

- Path: F5 "Maintain" => F1 "Alarm" =>F2 "History alarm", and controller will display history alarm on screen
- In the No. column, the smaller number it is, the later the alarm occurs

4.6.3 Save Alarm

In case user need support from machinery for repairing once alarm occurs. User can export the alarm to an external storage device, and send it to machinery. In this way, machinery could do trouble shooting first.

- Operation:
  - Connect the external storage device to controller, or set the corresponding network folder
  - Switch to "Alarm" page (F5 "Maintain" => F1 "Alarm")
  - To export the pending alarm, press F1 "Pending alarm" => F3 "Save alarm"
  - To export the history alarm, press F2 "History alarm" => F3 "Save alarm"
  - External storage device will be displayed on screen, select the destination folder
  - Select OK to complete alarm exporting

- File name:
  - Pending alarm：Actalm.txt
  - History alarm：Histalm.txt

4.7 Network Setting

| G54 | | TEST N0 L1 | Set Kernel Server | 2020/1/22 | 11:42:20 | DEFAULT |

```
                    IP Address Parameter                1 / 1

IP Address Setting        │         Specify an IP Address          ▼│

IP Address          │          │   Name Server Parameter
Subnet Mask         │          │   Primary DNS       │          │
Default Gateway     │          │   Primary WINS      │          │

                 Network DiskRemote Host Path

PC Name             │          │   Dir Name          │          │
User Name           │          │   Password          │          │
Net Status          │ Code：-1                                  │

                      Resource Shared
Shared Folder Path  │                                           │
```

| ●Ready | Auto | | **Alarm** |

- Path
    - 5 key, 8 key: F5 Maintain → F2 Set Kernel Server
    - 10+8 key: G5 Maintain → F2 Set Kernel Server

- Explanation
    - Set up the system network

- Parameters
    a. IP Address Setting
        - Select "Specify an IP Address" when using a crossover cable
        - Select "Obtain an IP Address via DHCP" when using a normal cable, "IP Address" and "Subnet Mask" can then be skipped

    b. IP Address
        - Enter the applicable IP address in the domain

    c. Subnet Mask
        - Enter the subnet mask of the IP address
        - Should be the same as the setting at PC end

    d. PC Name
        - Name of the connecting PC
        - Should be the same as the setting at PC end

    e. Dir Name
        - Name of the shared folder at PC end (should be the same as the setting at PC end).

    f. User Name
        - Can be skipped if no account and password is set to protect the folder shared by the Internet disk; if do, set the corresponding account and password.

    g. Password
        - Same situation as "User Name"

4.7.1 Set Kernel Server
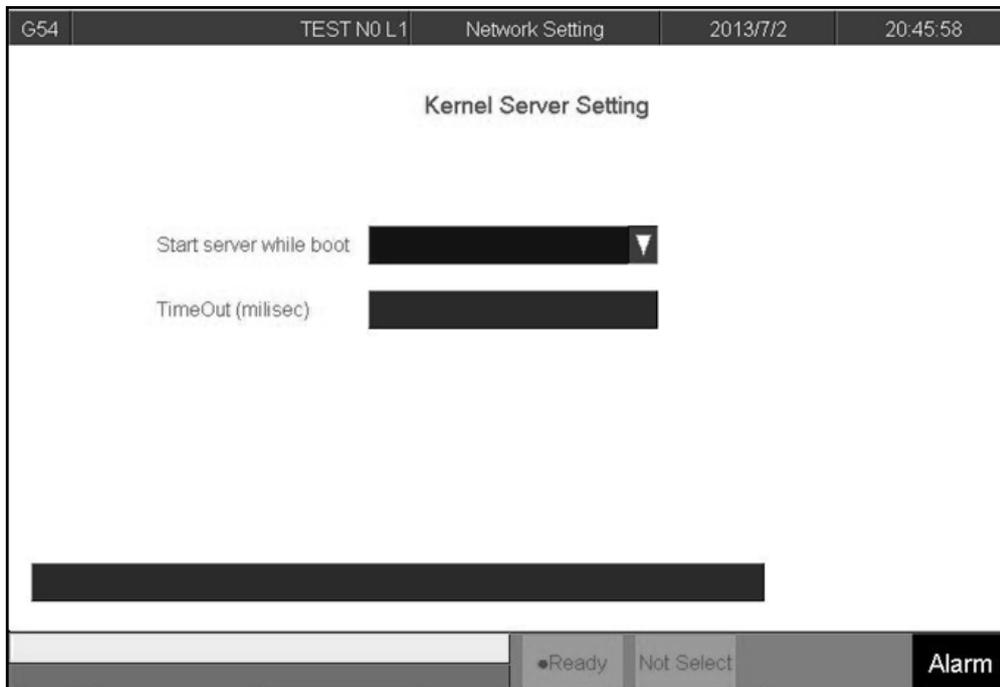
- Path
    - F5 → Maintain → F2 Set Kernel Server → F5 Set Kernel Server

- Explanation
    - Set the related function of kernel server.

- Parameters
    - Start server while boot
        - Start the server after booting or not

    - Timeout(ms)
        - Set the acceptable timeout for connection failure

Start Server

- Path
    - F5 Maintain → F2 Set Kernel Server → F5 Set Kernel Server → F1 Start Server

- Explanation
    - Start the kernel server immediately.
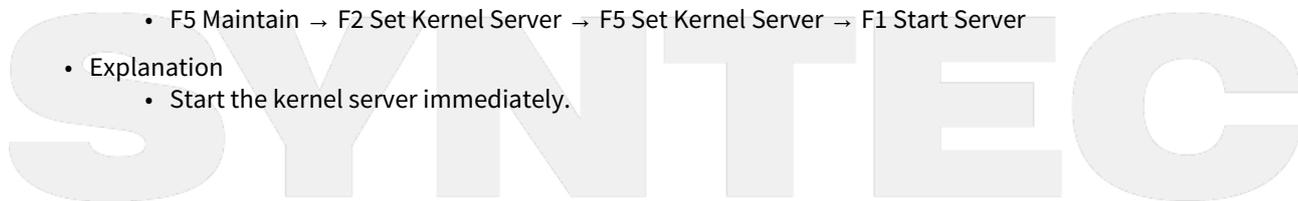
4.8 PC Setting

4.8.1 XP

1. Guest account setting
   Log in as Administrator and select "start" => "control panel" => "user account" , enable "Guest" account

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"
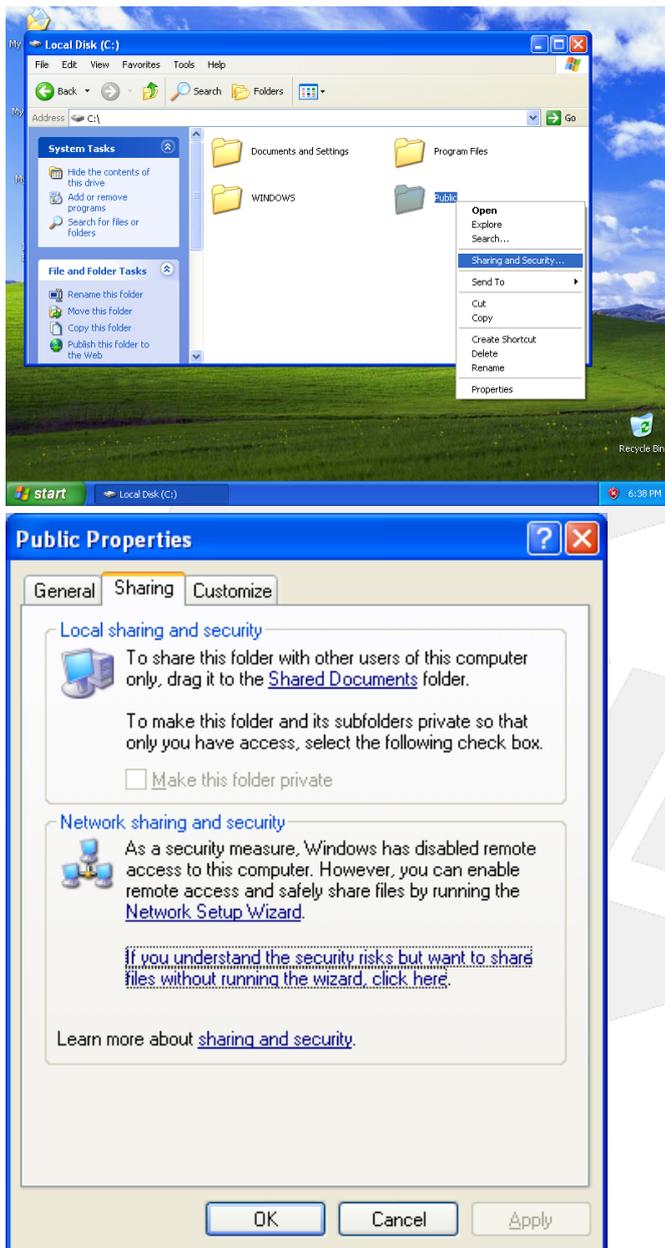
3. Click "OK" to confirm sharing setting; Select "Share this folder on the network", and "Allow network users to change my files".
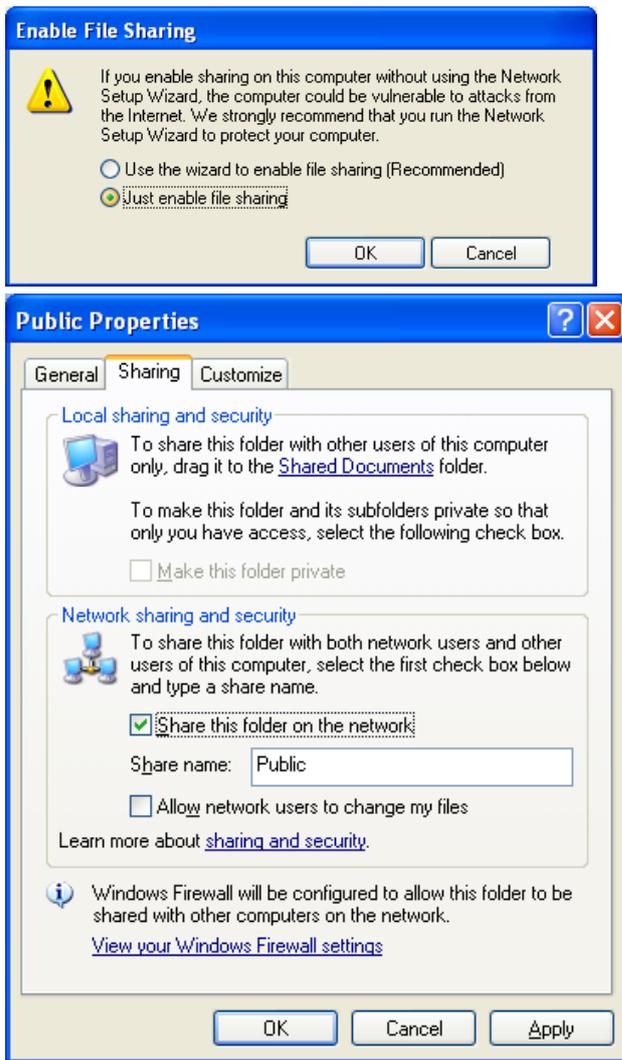
4. Setting PC name and workgroup
   "Start" => "control panel" => "System" => "change" to set "Computer Name" and "Workgroup", and

remember these setting contents. You will use these later when setting controller.
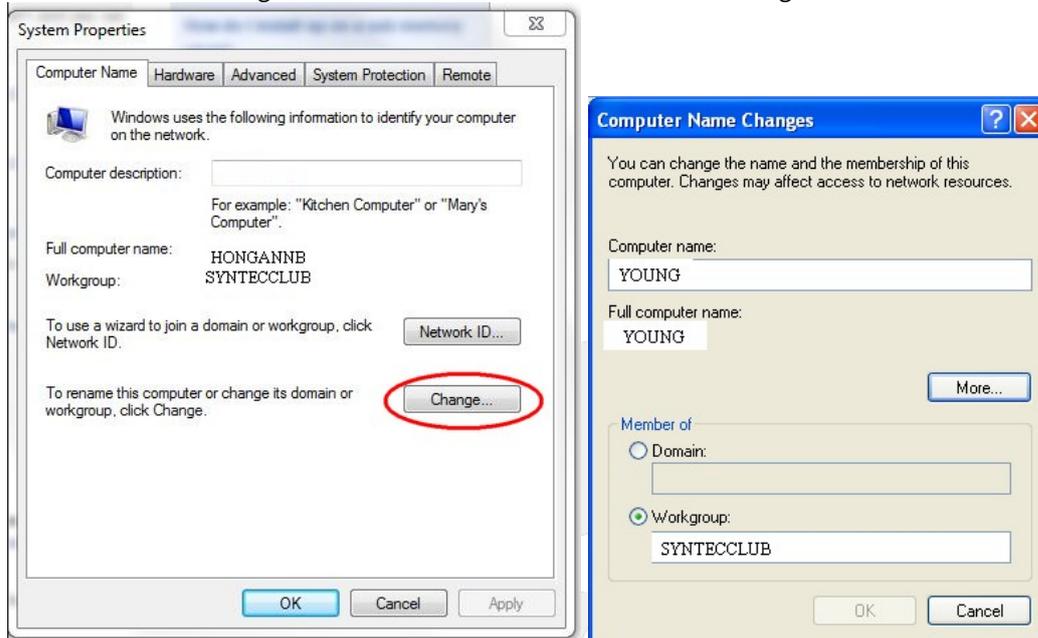
5. TCP/IP setting
   "Start" => "Setting" => "Network connections" and right click on "Properties", and select "Internet Protocol [TCP/IP]"
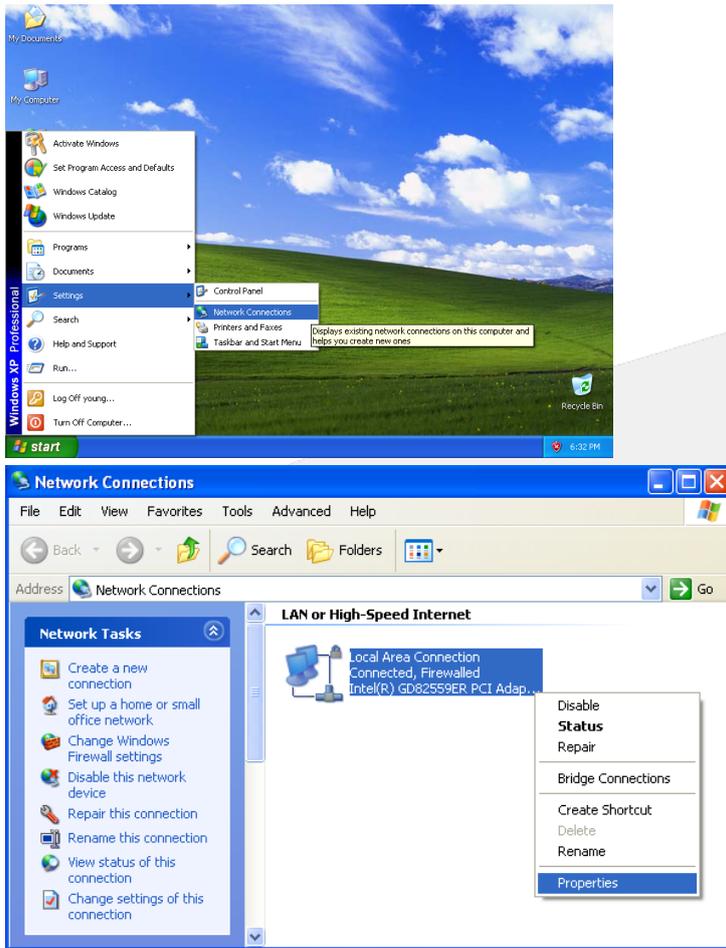
- Jumper cable (without HUB), select "use the following IP address" and enter IP address (the forth number must be different from controller setting) and Subnet mask (same with controller setting)

- Network cable (with HUB), select "Obtain an IP address automatically"
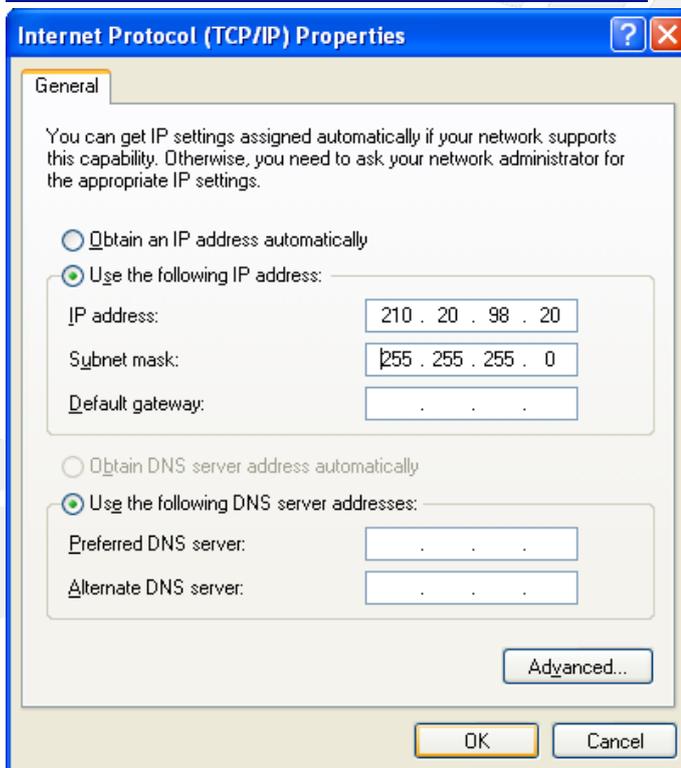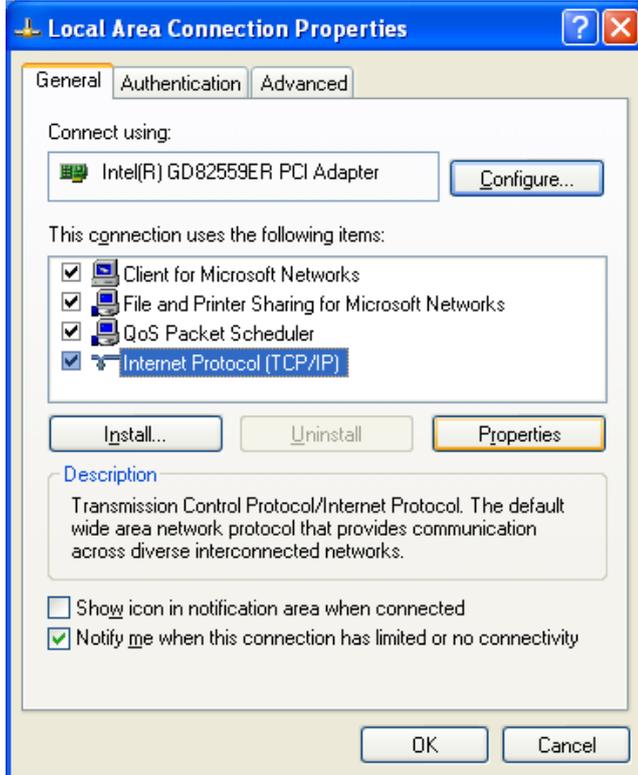
4.8.2 VISTA

1. Guest account setting
   Log in as Administrator and select "start" => "control panel" => "user account" => "Guest" account\
2. Sharing resource setting
   Create a sharing folder, and change this folder's setting to offer controller to use, as the below pictures.
   - Right click on folder and click "Properties", switch to tab "Sharing", and click "advanced sharing"
   - Click "share this forlder"
   - Click "Permissions"
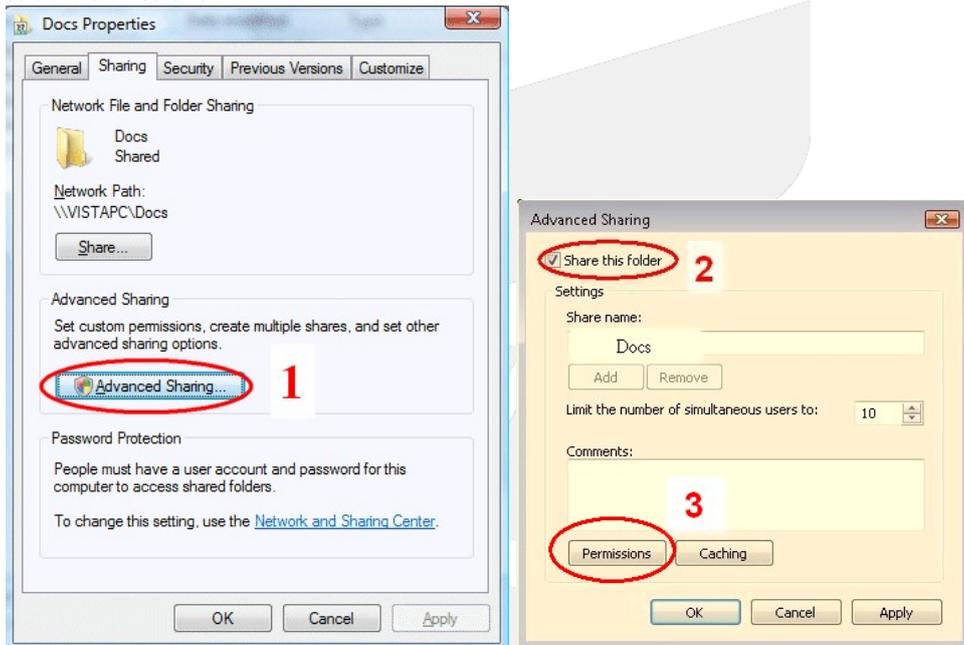
   - Click "add"
   - Enter "GUEST" as the new group name, click "OK" to complete setting

3. Security setting
   Right click folder => click "Properties" => "Security" => "Edit" => add "Guest" as a new group, and maximize permission for "Guest" group.
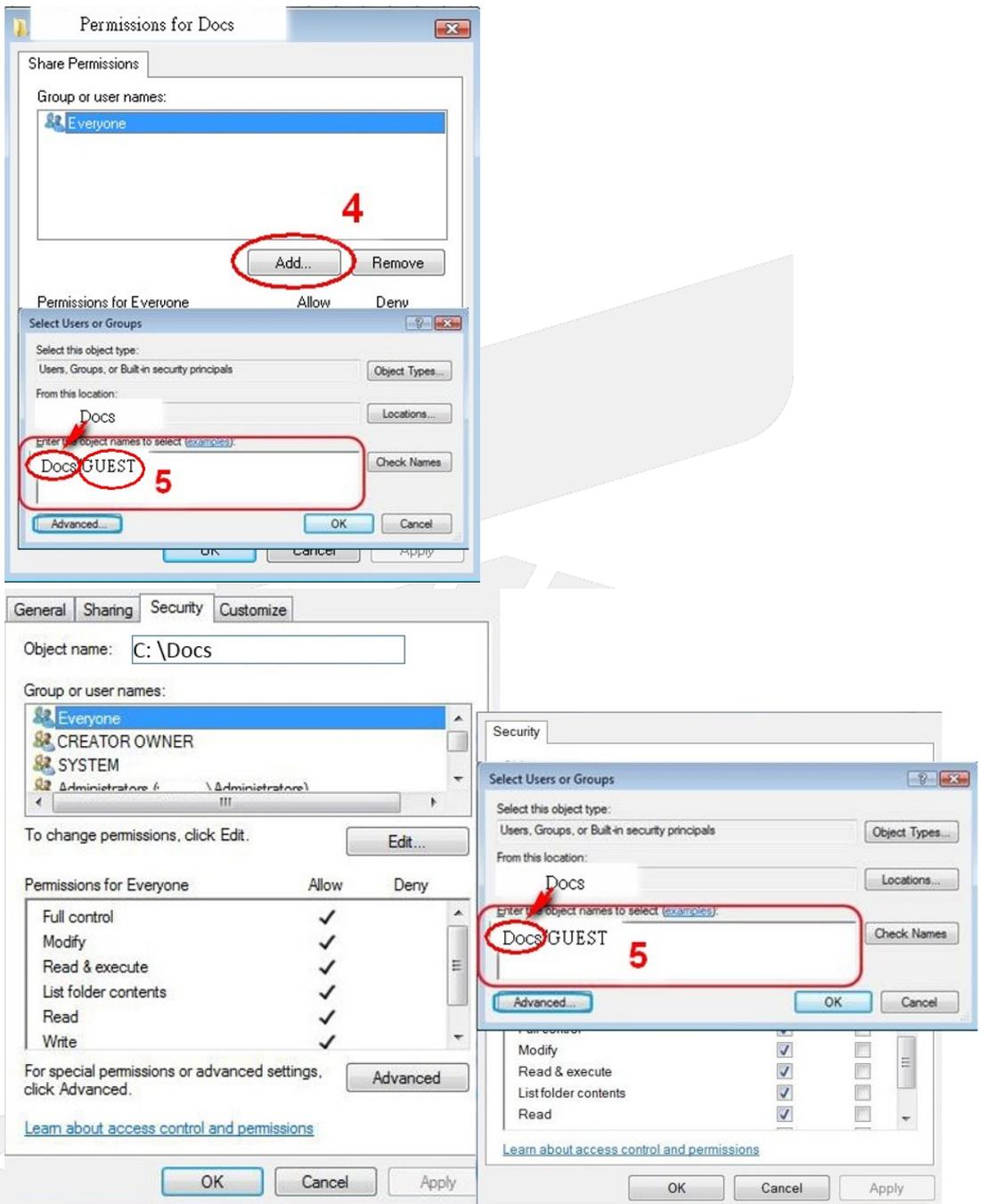
- Set workgroup and name of PC
- "start" => "control panel" => "System" => "change." Then you should see "Computer Name" and "Workgroup." Remember them.

4. TCP/IP Setting

- Log in as Administrator and do the followings in sequence. "start" => "control panel" => "network and sharing center" => "network connection manager" => "properties"
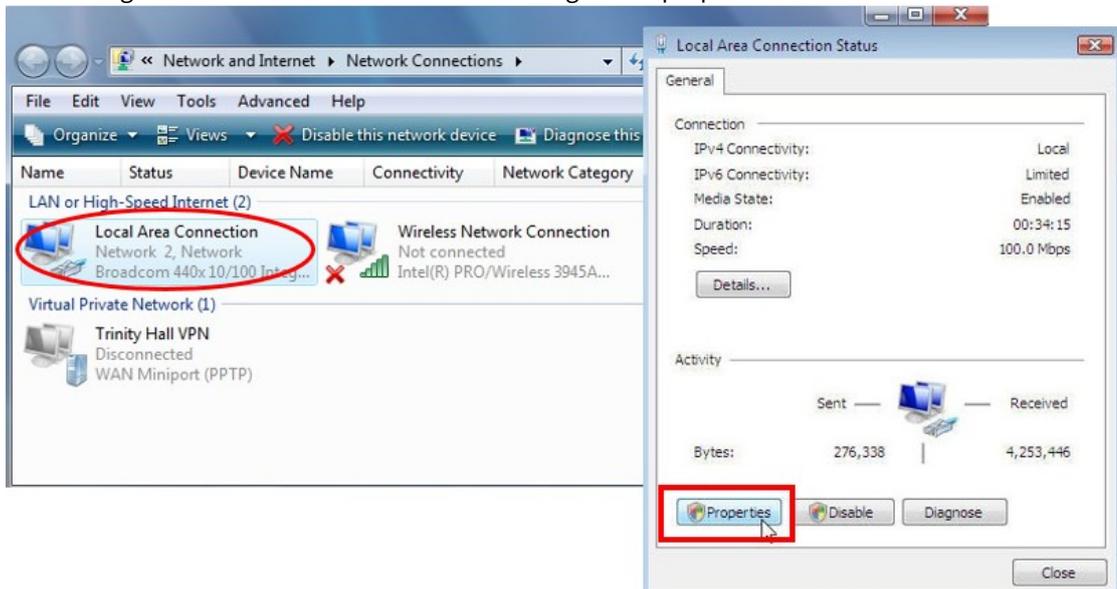
- Select "internet protocol(TCP/IPv4)":
    - Jumper cable (without HUB), select "use the following IP address" and enter IP address (the forth number is different from controller setting) and Subnet mask (same with controller setting)
    - Network cable (with HUB), select"Obtain an IP address automatically"
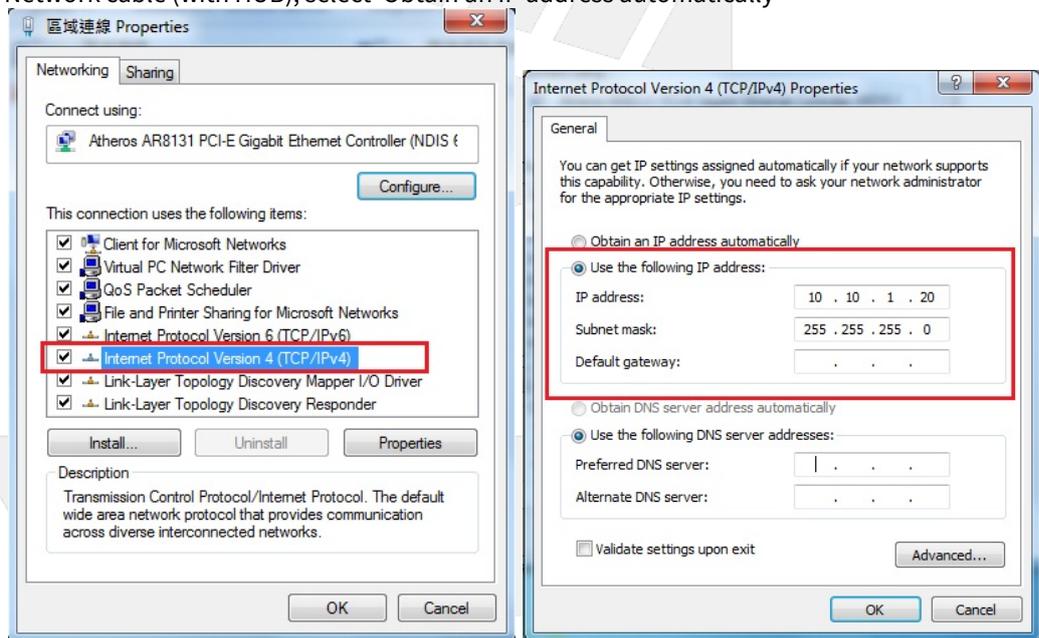
4.8.3 Win7

- Sharing resource setting
Create a sharing folder, and change this folder's setting to offer controller to use, as the below pictures.

1. Right click folder, click "share with" and "specific people"

2. Share this folder to "Everyone", and then click "Share" as below. Set "access level" as write/read
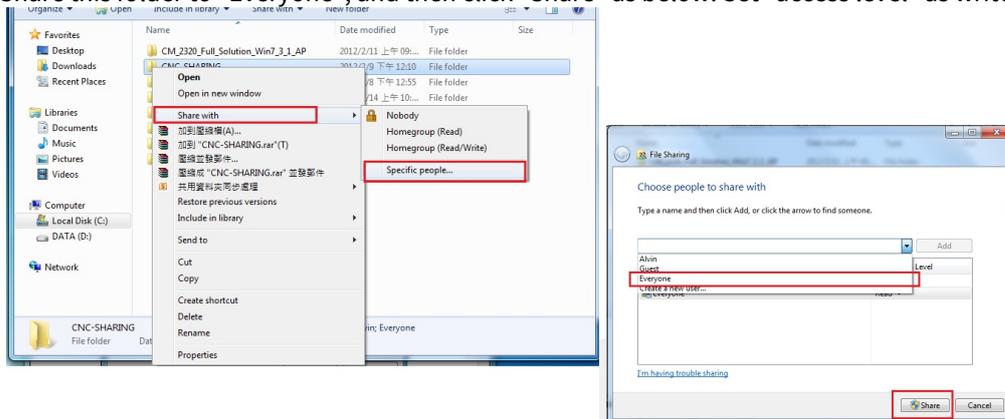
3. Click "advanced sharing" and select "share this forlder"
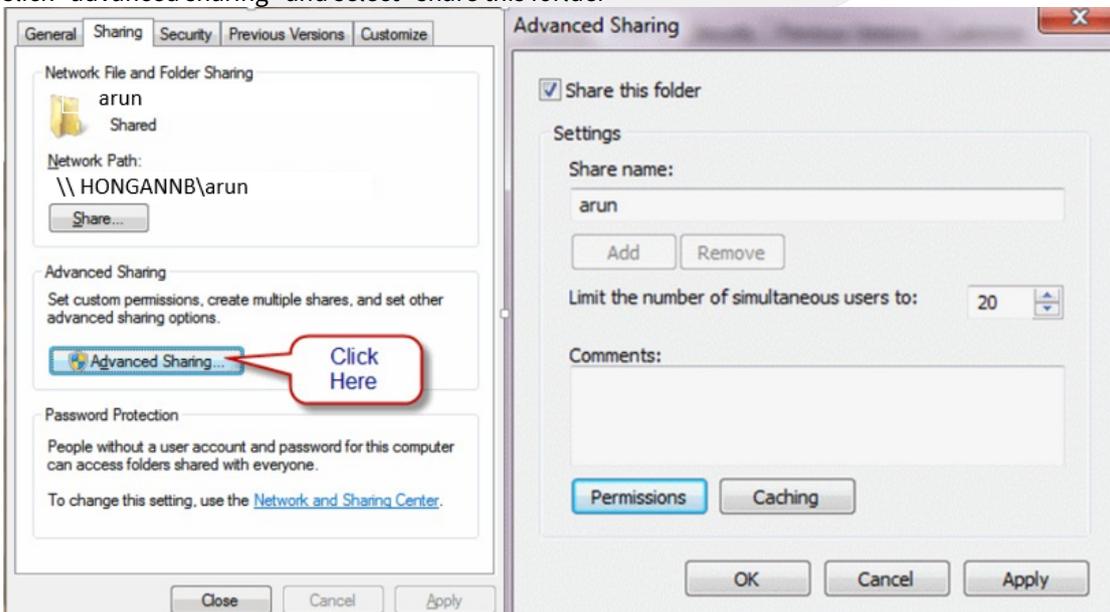
4. Select "permission" and select "full control" "only read"and "change"
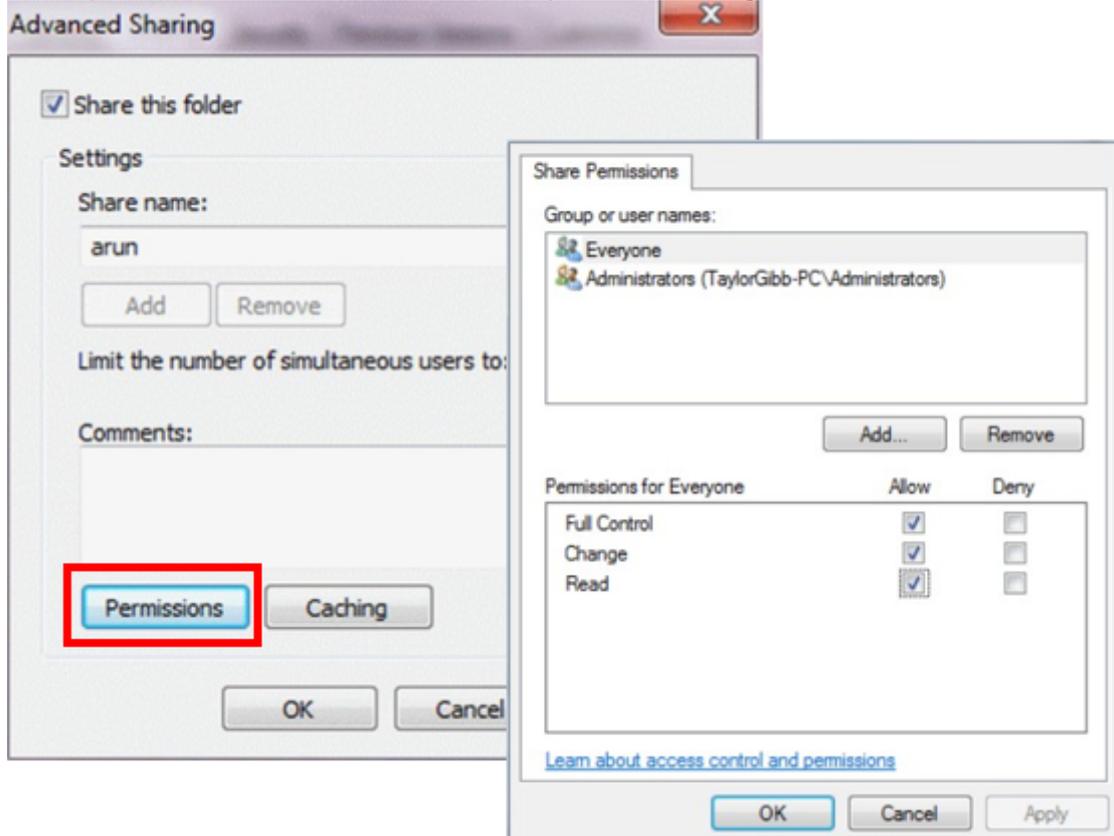
5. Open"network and sharing center", select "Turn off Public folder ...." and "Turn off password protected sharing"
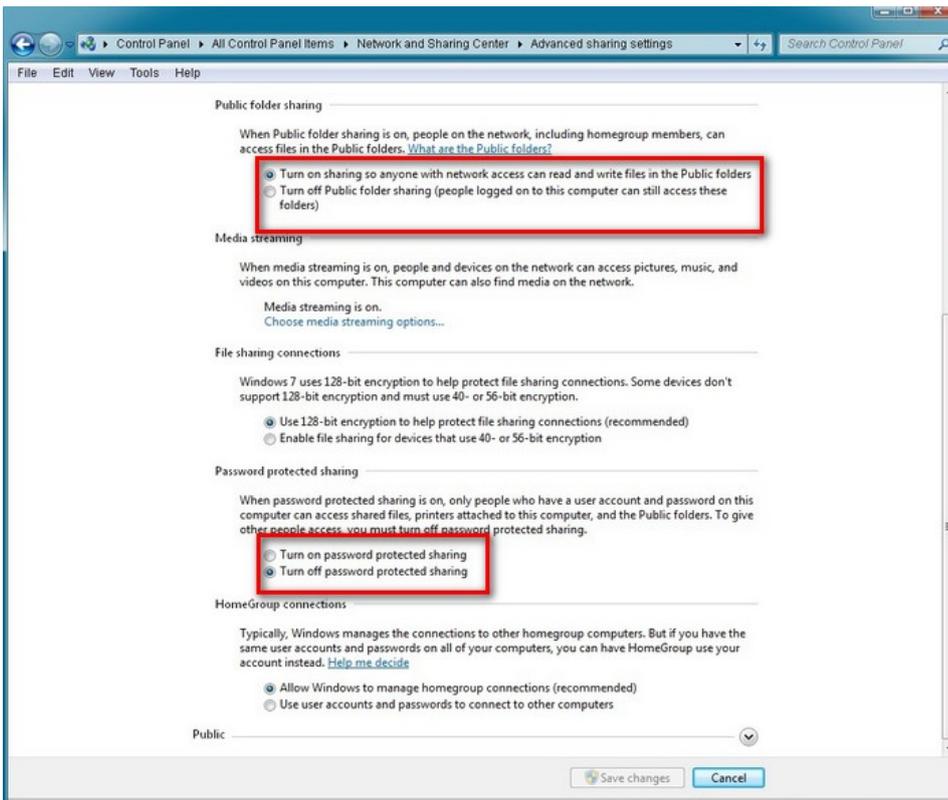
- Set workgroup and PC name

"Start" => "control panel" => "system and security" => "System" => "Change settings " to see "Computer Name" and "Workgroup." Remember them.
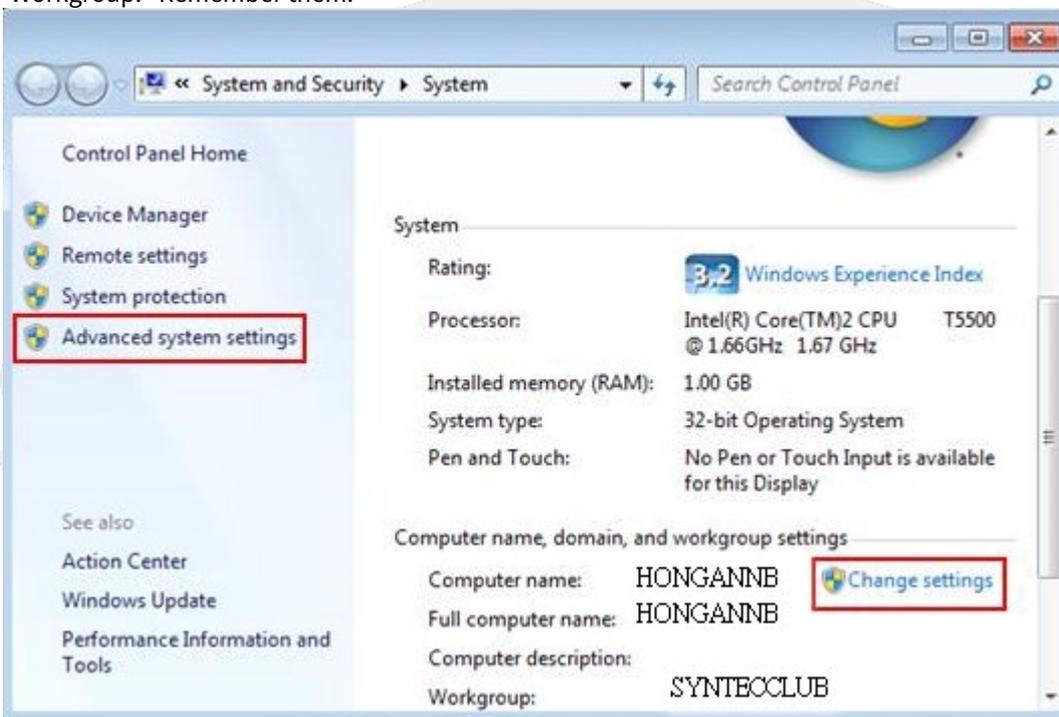
- TCP/IP Setting)
    a. Path: "Start" => "Control panel" => "Network and sharing center" => "network connection manager" => "properties"
    b. Double click "Internet Protocol Version 4 (TCP/IPv4)"
        i. Jumper cable(without HUB), select "use the following IP address" and enter IP address(the forth number is different from controller setting) and Subnet mask(same with controller setting)
        ii. Network cable(with HUB), select "Obtain an IP address automatically"
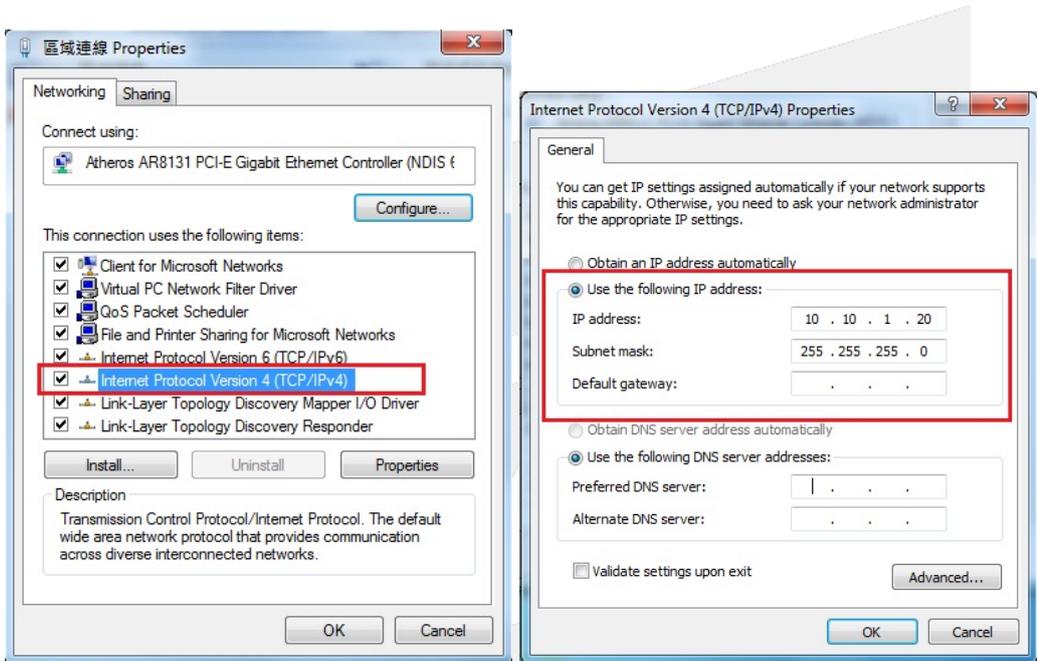
4.9 File Transfer

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.

4.9.1 File Import

- Condition:
    - None.

- Operation:
    a. Path: F2 "Program" => F8 "File Manager" => F4 "File Transfer" => F1 "File Import"
    b. A window will show up on screen, press F5 "Device Select", and move cursor to the desired external device
    c. Press "Enter" to access to the desired external device. If the device icon has red cross, it means that there is no connection between CNC and this external device.
    d. Press F2 "Select" to choose file. The box will be checked. Press F2 again to cancel it.
    e. Press F3 "Select All" to select all files.
    f. Press F4 "Cancel Select" to cancel all chosen files.
    g. If the file is in the folder, move cursor on the folder and press "Enter" to enter it.
    h. Move cursor on the "..." folder and press "Enter" or just press "Backspace" to back to the parent folder.

i. Press F1 "Copy" to complete file importing.
j. After complete file transfer, press "Exit" or ESC to leave this window
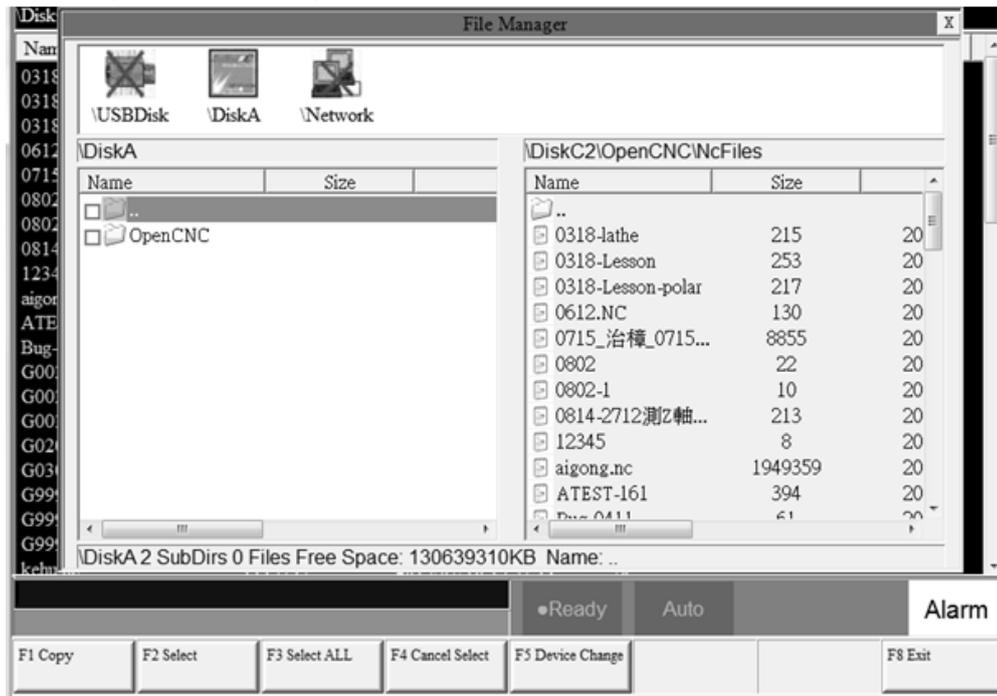
4.9.2 File Export

- Condition:
    - None

- Operation:
    a. Path: F2 "Program" => F8 "File Manager" => F4 "File Transfer" => F2 "File Export"
    b. A window will show up on screen, press F5 "Device Select", and move cursor to the desired external device
    c. Press "Enter" to access the desired external device. With red cross on the device icon, it means that there is no connection between CNC and this external device.
    d. Press F2 "Select" to choose file. The box will be checked. Press F2 again to cancel it.
    e. Press F3 "Select All" to select all files.
    f. Press F4 "Cancel Select" to cancel all chosen files.
    g. If the file is in the folder, move cursor on the folder and press "Enter" to enter it.
    h. Move cursor on the "..." folder and press "Enter" or just press "Backspace" to back to the parent folder.
    i. Press F1 "Copy" to complete file exporting.
    j. After complete file transfer, press "Exit" or ESC to leave this window
    k. If user would like to change the destination external device, press F6 "Window Select". Cursor will move to external device, which is the right window.

- Note:

- While user try to enter a unconnected external device, the warning window shows as below.
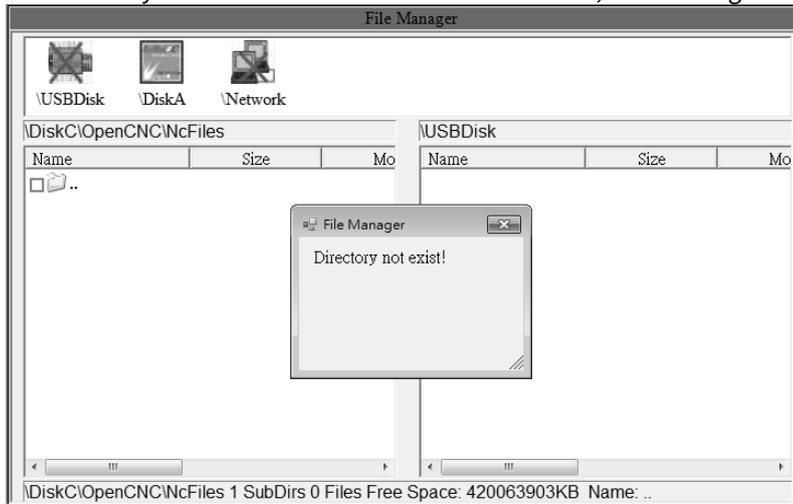
5 **Multi-Program Controller Screen Operation**

5.1 Intro

For controller software version above 10.116.35, 10.117.35 (included), HMI operation screen for multi-path is provided. The coordinates, information etc. of each path can be shown individually, the operation and display will be introduced below.

5.2 Multi-Program Screen

As shown below

Dual Programs

(1)			(2)	(3)	(4)
(5)	(6)	(7)	(5')	(6')	(7')

Screen

(8)	(9)	(10)	(11)	(12)	(13)
Input	HintT				Alarm
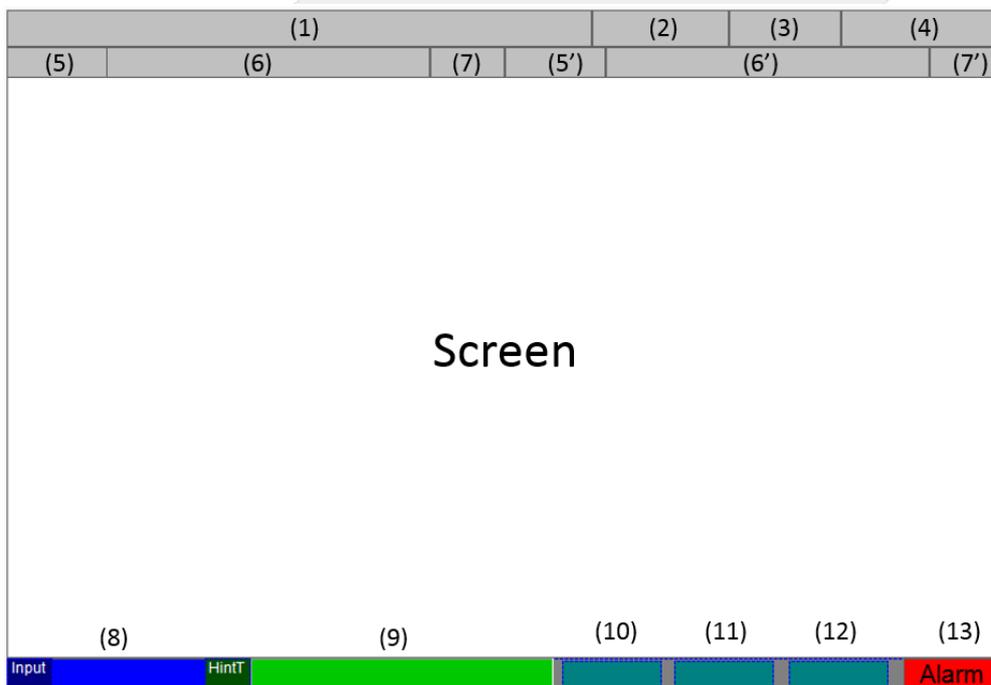
Screen Description

- (1) Page Title
- (2) Time
- (3) Date
- (4) User
- (5) Program No.
- (6) Machining Process / Line
- (7) Path
- (8) Input Column
- (9) Input Hint
- (10) State
- (11) Mode
- (12) Dipole Internet Connection Hint
- (13) Alarm

5.3 Multi-Program Main Function Screen

The figure below shows the main function screen of CNC lathe controller. The fenubar F1~F8 below is used for the operation of CNC lathe controller, users can enter the corresponding sub function by simply pressing F1~F8 on the operation panel:

Dual Programs

Screen Description：

- Screen section：Left- shows the information of 1st path, Right- shows the information of second path
- Machine Coordinate：Coordinate related to the machine
- Feedrate (F)：Assign the tool moving speed for each cutting, unit: mm/min
- Spindle Speed (S)：RPM value of spindle rotation speed
- Override：Spindle override information
- Note：For axes belong to non-main system paths (third, fourth path), the information will be displayed in $1 coordinate object (valid version: 10.116.36R, 10.116.47, 10.117.47)

5.3.1 F1：Information

The coordinate systems showing current location can also be used to reset the relative coordinate. Press **"Information"** on the main function screen

Dual Programs

- F1：Switch Coordinate.
  Function：Switch the displaying coordinate system.
  Operation：In "Machine Setting" screen, users can switch to show the data of 4 coordinates by pressing F1 anytime

- F2：Half Coordinate
  Function：Divide the relative coordinate value by 2 and apply with Clear Rel. Coord. to calculate the middle point
  Operation：Key in the calculating axis and press the button.

- F3：Zero Relative Coordinate
  Function：Clear the relative coordinate value to 0.
  Operation：Key in the calculating axis and press the button.

- F4：Zero all relative coordinates
  Function：Clear all relative coordinate values to 0.
  Operation：Press "F4 Clear all relative coordinates" on the "Machine Setting" screen.

5.3.2 F2：Program

Syntec controllers provide program editing function, users are able to edit the program with this function
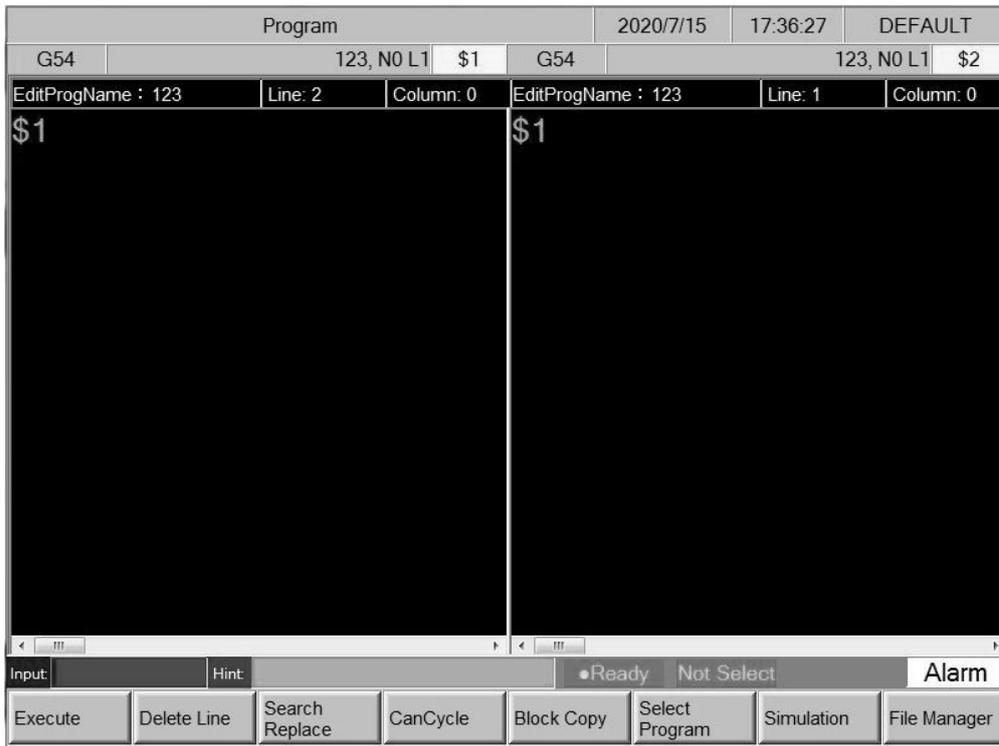
- F1：Execute
  Explanation：The button can assign the editing program as the machining program then switch the screen to "Monitor" screen.

- F2：Delete Line
  Explanation：Delete the line which the cursor is at.

- F3：Search / Replace
 Explanation：Able to search and replace the lines.

- F4：CanCycle
  Explanation：Since the system provides so many G codes for program editing and the definition of arguments of each G code are also different, CanCycle is able make the editing easier.

- F5：Block Copy
  Explanation：Able to copy the content of editing block.

- F6：Select Program
  Explanation：Switch the current focus of function.

- F7：Simulation
  Explanation：Able to preview the machining path of current editing program and check if there is any program error

- F8：File Manager
  Explanation：Able to manage all the machining files of the system

- Notification：
    - "Teach" function is not supported in multi-program HMI program

5.3.3 F4：Monitor

Shows the coordinate, parameters, simulation switch, feedrate, machining time, MDI and common machining information

Dual Programs
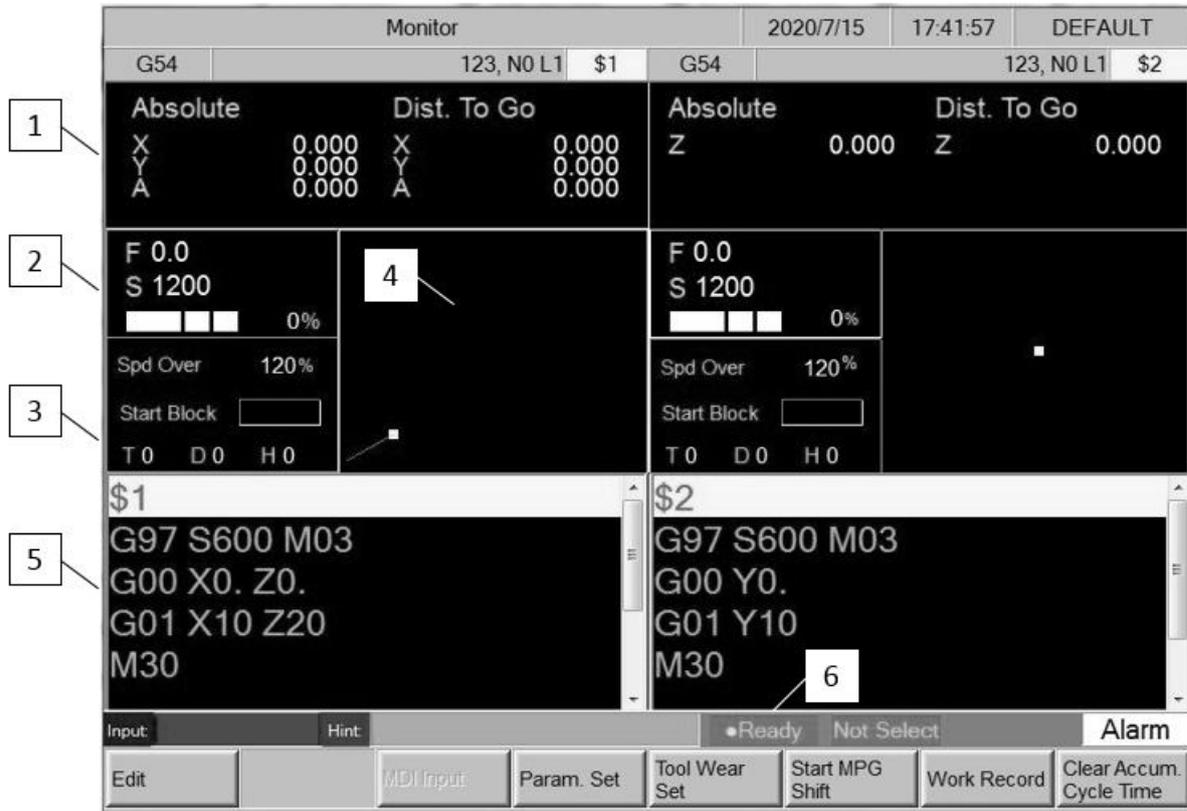
Screen Description

1. Coordinates
   • Note：For axes belong to non-main system paths (third, fourth path), the information will be displayed in $1 coordinate object (valid version: 10.116.36R, 10.116.47, 10.117.47)

2. Spindle Speed, Feedrate, Override
3. Spindle Ratio, Starting Block, T code
4. Simulation Switch
5. Param. Set
6. Multi-path mode, only shows the machining state of $1 (when running G04.1 waiting function, system see as busy)

Supporting Functions

1. The input cursor at starting block will be moving with the chosen path
   • Valid version: 10.116.36J, 10.116.42, 10.117.42
   • Description
       • When switching the chosen single path, the cursor at starting block will be automatically switched to the chosen path (ex: when the chosen path is switched from $1 to $2, the starting block cursor will move to $2)
       • When switching the chosen multi-path

- If the chosen paths include the original path, the cursor will remain at the same place (ex: the chosen path is switched from $1 to $1 and $2)
- If the chosen paths do not include the original path, the cursor will move to the path of smallest number (ex: the chosen path and cursor are at $2, when the chosen path is switched to $1 and $3, the starting block cursor will move to $1)

- After switching the screen, the starting block will save the cursor location of previous screen

2. Search
   - Description
     - The starting lock column will show the searching content
     - The searching of starting number (prefix N) is according to the current chosen paths
     - The searching of numbers, L numbers and strings are according to the current starting block cursor located path

3. Expansion input method of Search function
   - Valid version: 10.116.36P, 10.116.45, 10.117.45
   - Description
     - Separates the searching of N code, line number or characters in different path with "/" .
     - Example: N1/L2 means searching N1 in the first path program($1) and searching L2 in the second path program($2)
     - Enter /L2 means only searching L2 in the second path program
     - If skipping n paths, input n-1 "/". Does not affected by the chosen paths
     - The searching will follow the original rule if no "/" is entered.

   - Compatibility Changes
     - String searching with "/" is no longer supported
     - The "Can't Find" message will now pop up after searching all paths instead of showing one by one
     - The pop up hint message will now indicate which program does it belongs to

4. R21 related definitions and descriptions can refer to the R Register Interface chapter in CNC and PLC interface manual

Fenubar Description

- F1：Edit
  Explanation: Load the current machining file into program editor to edit and switch the screen to "Edit".

- F3：MDI Input
  Explanation: Edit the MDI executing program
     a. When R21.11 is off
        - R21=0, shows the dual program screen
        - R21=1: shows the 1st path program in full screen
        - R21=2: shows the 2nd path program in full screen
        - R21=3: shows the 3rd path program in full screen
        - R21=4: shows the 4th path program in full screen

     b. When R21.11 is on

        - R21.1 on: shows the 1st path program in full screen
        - R21.2 on: shows the 2nd path program in full screen
        - R21.3 on: shows the 3rd path program in full screen
        - R21.4 on: shows the 4th path program in full screen
        - The bits mentioned below are incompatible with each other. For other bit combinations: **always shows the dual program screen**
        - Note: If not chosen single path, **the MDI program editor will always shows the dual program screen**

- F4：Param. Set
  Explanation: Used to switch between "Information" and "Setting", can total switch 3 times including the machining screen.

    - After pressing F4 Param. Set

Changes after switching

  a. Spindle Ratio, G00 G01 Ratio, T code
  b. G code state

- After pressing F4 Pram. Set again

Changes after switching

a. Switched to common information screen, Machining time, breaking point line number, workpiece setting, feedrate, spindle speed and spindle state

- Press F4 Param. Set to return to the machining screen

• Compatibility Change:
  • 10.118.0C, 10.118.3：module 220MB, 220MB-5, 220TB can display the information of 8 spindles

• F5：Tool Wear Set
Explanation：Shows the tool wear setting screen
ARM type：Tool Wear Set and Information are at the same fenubar layer, the tool wear setting screen can be shown by pressing the fenubar and be closed by pressing the fenubar again (back to the Information screen)

• F7：Work Record
Explanation：Shows the working record table

• F8：Clear Accum. Cycle Time
Explanation：Clear the accumulated machining time to 0.

6 **Appendix**

6.1 Contact Window

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

Zhejiang-Wenling The Suzhou new generation of CNC Equipment Co., Ltd. Wenling Branch TEL：0576- 86138372 FAX：0576-86119106 Address： Room 1206, Department A, Zhenxing Plaza, Taiping Subdistrict, Wenling City	**Guangdong-Guangzhou** SuZhou SYNTEC Equipment CO.,Ltd. Guangzhou Branch TEL：+86-20-34583040 FAX：+86-20-34583220 Address： Room 403 Stairs1 Youyi Building!ALuojia Village 8,Fuyi Road,ShijiTown,Panyu Distrct,Guangzhou City,Guangdong Province, China
Guangdong-Dongguan SuZhou SYNTEC Equipment CO.,Ltd. Dongguan Branch TEL：+86-769-81660318 FAX：+86-769-81660328 Address： Room 705 JinGuo Business affairs centre Xin an District Chang an	**Shaanxi-Xi'an** The Suzhou new generation of CNC Equipment Co., Ltd. Xi'an Office TEL：029-88287423 FAX：029-88287423 Address： Room 2503,Unit 1,Rancho Santa Fe,No.36, Dianzi Third Road, Yanta District, Xi'an, Shaanxi
Shaanxi-Baoji The Suzhou new generation CNC Equipment Co., Ltd. Baoji offices TEL：18700712118 Address： Room 2401, Building 1, Xinjian Road, Weibin District, Baoji	**Shandong-Jinan** SuZhou SYNTEC Equipment CO.,Ltd. Jinan Branch TEL：+86-53185907208 FAX：+86-53185905708 Address： Room 1112, Unit A, Jiahui Global Plaza, No.548, Beiyuan Street, Tianqiao District, Jinan, China
Fujian-Xiamen SuZhou SYNTEC Equipment CO.,Ltd. Xiamen Branch TEL：+86-592-7191901 FAX：+86-592-7220536 Address： No. 100,B Chuang Chang Fang 3F East,Jin Fu Road,Tong An District,Xiamen City, Fujian Province, China	**Tianjin** The Suzhou new generation of CNC Equipment Co., Ltd. Tianjin Branch TEL：+86-22-87134111 FAX：022-87134111 Address： Tianjin Huayuan Industrial Zone, Rong Yuan Road No. 4 day science and Technology Park Building No. 1, 2 door 501
Henan-Luoyang The Suzhou new generation of CNC Equipment Co., Ltd. Luoyang Company TEL：0379-65110352 FAX：0379-65110352 Address： 601B, Runsheng Building, Cross of Sanshan Road & Heluo Road, Hi-and-New Tech Park of Luoyang, Henan	**Chongqing** The Suzhou new generation of CNC Equipment Co., Ltd. Chongqing Branch TEL：023-67913296 FAX：023-67634382 Address： Room 41-4, Building A, Hongding International, Guanyinqiao, Jiangbei District, Chongqing

Jiangsu-Nanjing The Suzhou new generation of CNC Equipment Co., Ltd. Chongqing Branch TEL：0512-69008860-300 Address： Room 505, Unit 3, Building 12,Wuyi Luzhou Guan Chu Yuan, No.99, Tianyuan Middle Road, Jiangning District, Nanjing	Hubei-Wuhan Suzhou new generation of CNC Equipment Co., Ltd. Wuhan Office TEL：027-87638876 FAX：027-87204137 Address： Room2003-2006,A#Guangguguoji,456#Luoyu Road,Donghukaifaqu,Wuhan City, China
Shenyang The Suzhou new generation CNC Equipment Co., Ltd. Shenyang Office TEL：024-25821398 FAX：024-25821398	Anhui-Hefei The new generation of CNC Equipment Co., Ltd. in Suzhou, Hefei Office TEL：15951989576 Address： Room 704, Building 13,Wanzhen Xiaoyao Garden Ⅳ, Taihu East Road, Hefei
Shenzhen Shenzhen branch District, Sham Chun City, Guandon Province TEL：0755-84584085 Address： Room 706, Hsin Toun Building, Toun Chung Road, Ai Lian Hsin Toun Village, Loggan	Thailand SIAM RADERMEN Co., Ltd. TEL：+66-27553536 FAX：+66-27575476 E-Mail： lee_cheahow@yahoo.com Address： 52/4,MOO5,THEPARAK RD. (KM11.5)BANGPLEEYAI,BANGPLEE, SAMUTPRAKRAN 10540
Malaysia SURE FIRST ENTERPRISE (M) SDN BHD. TEL：+603-61577632 FAX：+603-61574632 Address： 47000 NO 19.JALAN TSB 6,TAMAN INDUSTRI SUNGAI BULOH.47000 SELANGOR DARUL.EHSAN	Lebanon / Damascus Kesmeyan Group Co. Lebanon Beirut, Jdeideh - Ammaieh - Kessmeyan Group Build.DamascusTEL: +963 11 6731602Mob.: +963 988 211407E-Mail : info@rockwell.tw TEL：+961-76-883778 FAX：+961-1-901807 E-Mail： kesmeyan@rockwell.tw
France INTEGRATION CNC TEL：+33 2 35 06 07 83 FAX：+33 2 35 06 07 83 http://www.integrationcnc.fr/ E-Mail： vaque.j@online.fr Address： 6, Rue l'arché de Copigny, 76630 BAILLY, FRANCE	Turkey KASIKCIOGLU ELEKTROMEKANIK OTOMASYON TEL：+90-224-4434684 FAX：+90-224-4434685 E-Mail： murat@kasikcioglu.com Address： Besevler K.S.S. 17 BLOK NO:72 Nilufer/BURSA/TURKYIE

ISTANBUL/TURKEY AKSIS MAKINA TEL：+90 212 613 87 83 FAX：+90 212 613 87 85 http://www.aksismakina.com E-Mail： info@aksismakina.com Address： Yeni doğan mahallesi karakaş Sk. Emintaş Erciyes Sanayi Sitesi no:12/206	