



SYNTEC
TECHNOLOGY CO.,LTD.

60CA/60CB Simple-Type Wood Cutting Machine Operation Manual

匯出日期：2023-05-10

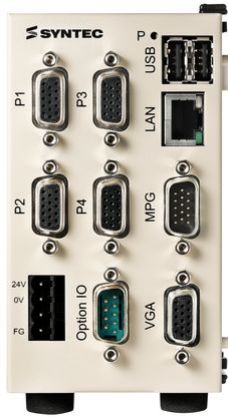
修改日期：2019-05-31

Manual



SYNTEC

1 Hardware



60CA (Pulse train controller)

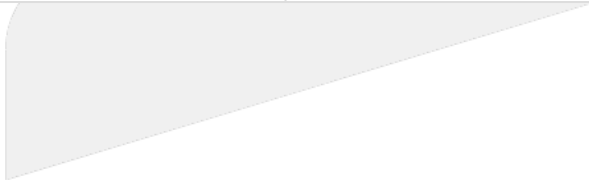


60CB (Serial-bus controller)

SYNTEC

2 Software Spec.

Item	Content
Axes(Standard / Option)	4 / 4
Spindle Number	6
Axes Group Number	4
Workpiece Coordinate Number	32
BPT	600
Software Version	10.116.54x, 10.118.12x, 10.118.x
Communication Protocol	Pulse: 60CA M3/EtherCAT: 60CB
Option	• HPCC : No • RTCP : No • ROT : No • AutoLabeling Function : 60CB



SYNTEC

Item	Content
Software Function Limit	• These G-code are not supported for 60CA/60CB - G1084: // fasttapping G code - G5: // HPCC mode - G5001: // path smooth mode - G5005: // disable learning control - G5006: // learning start - G5007: // enable parameter learning control - G6: // Parabolic Interpolation - G6002: // nurbs G code - G7001: // G07.1(G7001) cylinder programming - G22: // stored stroke check function on - G23: // stored stroke check function off - G33: // Constant Thread Cutting - G34: // Variable Thread Cutting - G41: // Cutter Compensation Left - G42: // Cutter Compensation Right - G45: // tool offset - G46: // tool offset - G47: // tool offset - G48: // tool offset - G50: // scaling cancel - G51: // scaling - G52: // local coordinate setting - G61: // exact stop mode - G62: // cutting mode - G62002: // S-Curve mode - G63002: // EDM mode - G74: // left-hand tapping - G76: // fine boring cycle - G81001: // chopping start - G84: // tapping cycle - G87: // accurate boring cycle - G88: // boring cycle - G89: // boring cycle with dwelling at bottom - G93: // 1 / time(min), inverse time feed - G96: // Constant Surface Speed - G97: // Revolution per minute - G900081: // fast drilling
Standard	• Support T-code and T0000 from T0~T16 • WorkingList • Plugin Enable (Option-4) • Pr4070 (Default Feedrate) • SRI setting screen for FC-IO

3 Manual Objective Machine

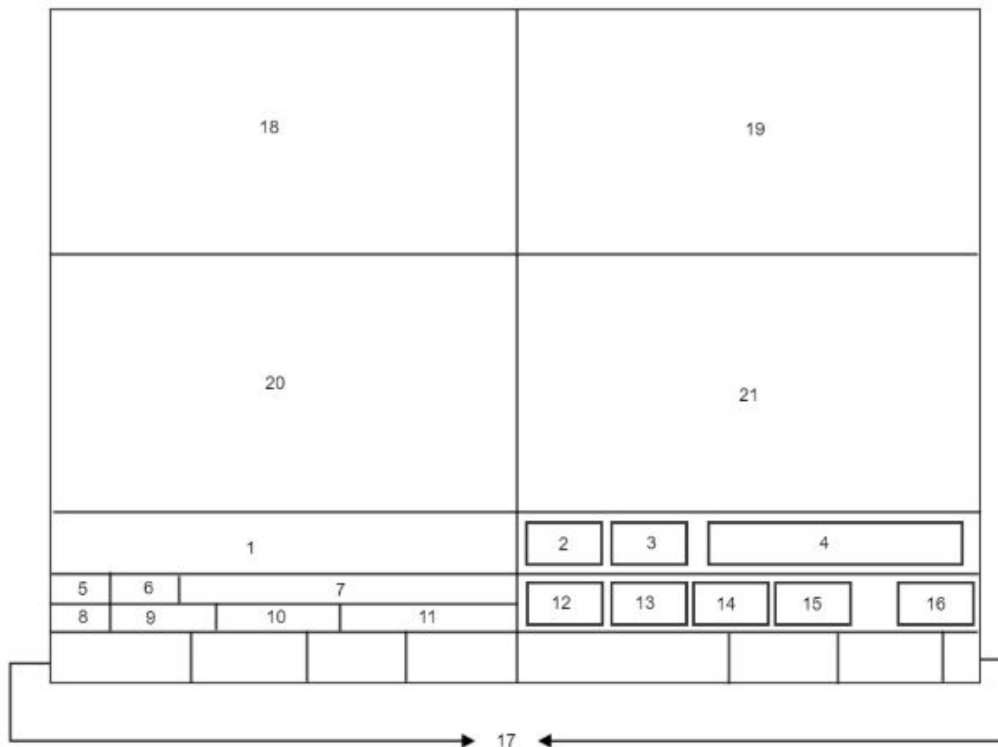
This Operation Manual is suitable for Syntec FC-60CA and FC-60CB controller.



SYNTEC

4 Fenu and System Introduction

4.1 Main Screen

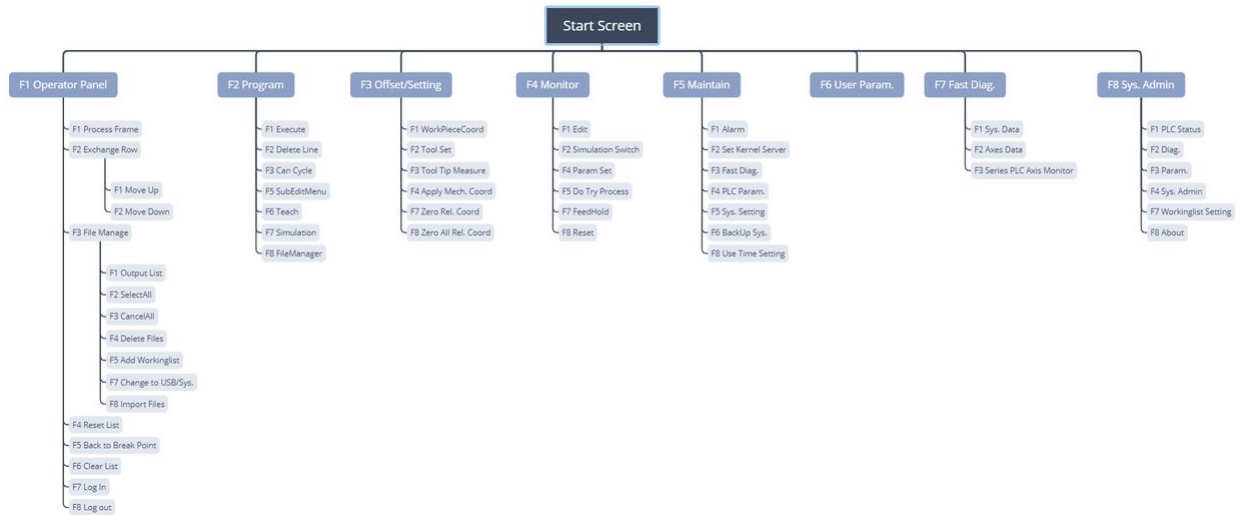


• Screen component description

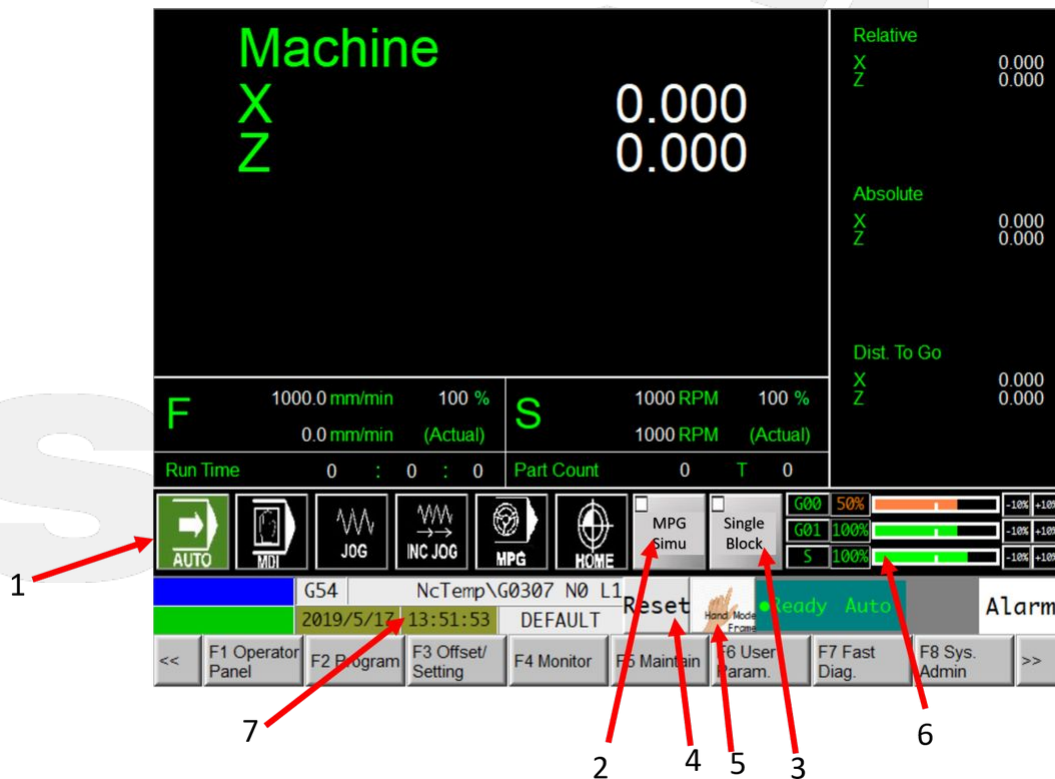
- a. **Mode Select**
- b. **HandWheel Simulation**
- c. **Single Block Execution**
- d. **Feedrate Adjustment**
- e. **Data Input**
- f. **Current Workpiece Coordinate**
- g. **Current Processing file and process line**
- h. **Hint**
- i. **Date**
- j. **Time**
- k. **Log in/out status**
- l. **Rest**
- m. **Hand frame panel**
- n. **Controller Status**
- o. **Mode**
- p. **Alarm**
- q. **Fenu Selection**
- r. **Process Simulation Previes**

- s. **G-Code status and Coordinate**
- t. **Workinglist**
- u. **Processing Record and Cycle start / Feed hold / Resret**

4.2 Function key Tree



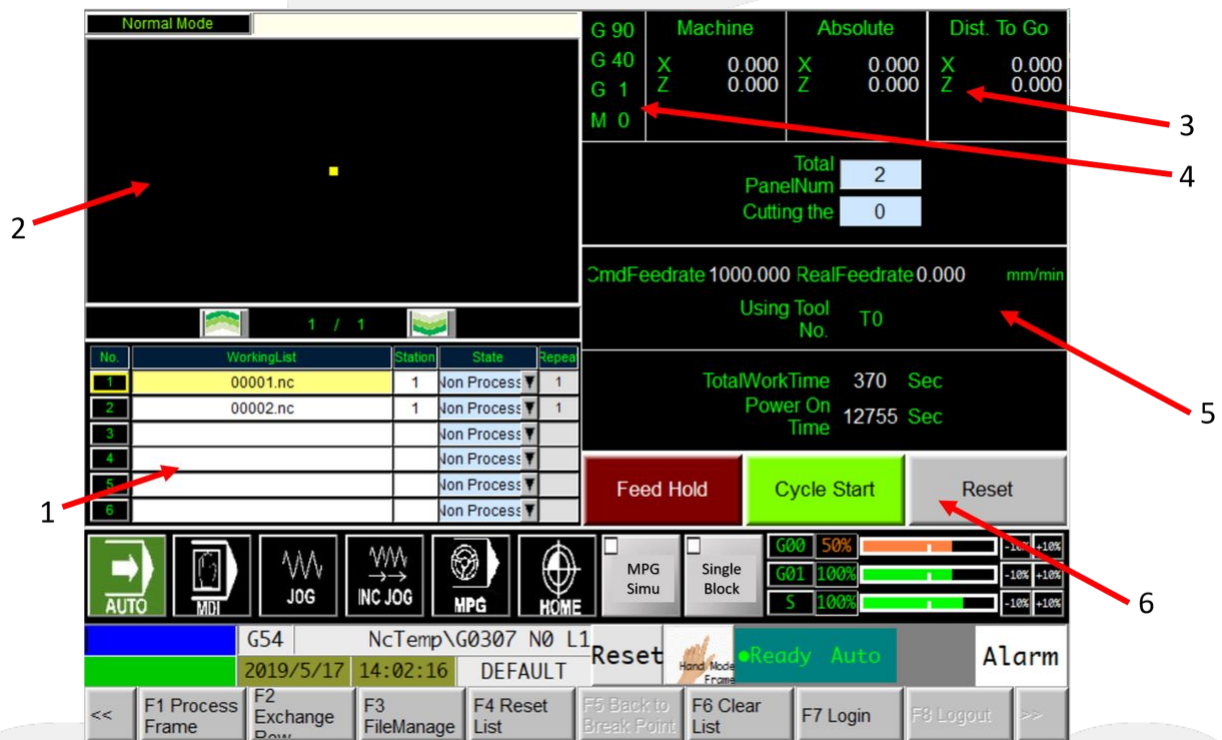
4.3 Start Screen



	Component	Those component is permently display on screen no matter how screen change
1	Mode Selection	- AUTO, Change to Auto Mode - MDI, Change to MDI Mode, and screen will change to F4 Monitor with MDI input popup - JOG, Change to JOG Mode - INJOG, Change to INJOG Mode - MPG, Change to MPG Mode - HOME, Change to HOME Mode
2	Handwheel Simulation	- Press the button and the box one the upper left be green, simulation mode enable - Press again the green box disappear, simulation mode disable.
3	Single Block Execute	- Press the button and the box one the upper left be green, single block mode enable - Press again the green box disappear, single block mode disable.
4	Reset	Action trigger the software be reset, no PLC interface will be triggered.
5	Hand Frame Panel	- Press the button, change screen to hand mode frame. This panel is provided to Syntec Technician to customize according to customer's demand - Axes jog, Peripheral devices, or other additional function. 

6	Feedrate Adjustment	- G00 rate adjust, 0%, 25%, 50%, 100% - G01 rate adjust, 0%, 10%, 20%,150% - Spindle rotation rate adjust, 0%, 10%, 20%,120%
7	Data	- Date - Log in User - Workpiece Coordinate, G54, G55... - Current Processing File - Current Processing Line

4.4 Operator Panel (MainFenu: F1)



4.4.1 Operator Panel-F1: Process Frame

- Devide this browser into different part, and function of each part is as below:

Part	Item
1	WorkingList
2	Process Path Simulation

Part	Item
3	Coordinate
4	G-code State
5	Process Information
6	Cycle Start/Feedhold/Reset

- Description
 - All the manipulation for operator can be executed in this page during processing
 - File Manage
 - Execute Process as arrange in workinglist
 - Adjust the sequence and work station (multiple-station machine) of NC files
 - Log in/out of operator
 - WorkingList process Monitor
 - Work Station default set **1 to G54, and 2 to G55**
 - Process state default built-in:
 - Undone
 - Aligning
 - Working
 - Printing
 - Done
 - Contact Syntec Technician if you have request for new process of the state.
 - Information of Workinglist shows as:

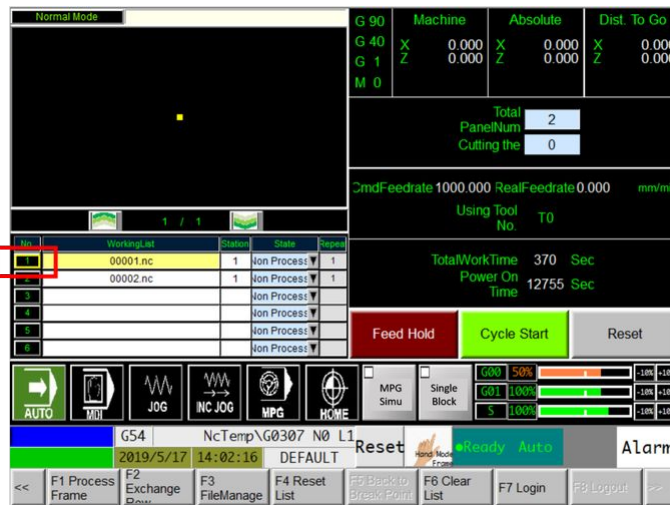
Sequence	NC files	Work Station	State	Repeat Times
1	00001.nc	1	Non Process	1
2	00002.nc	2	Non Process	1

4.4.2 Operator Panel-F2: Change Column

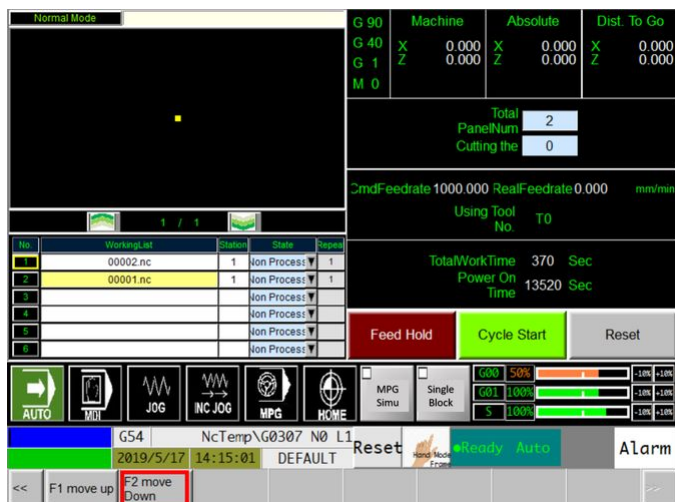
- Adjust the sequence of items in workinglist
- Select One Item by pressing the button on the left side of workinglist, and there are 2 ways to change the sequence.
 - Adjust selected item be forward, the sequence of selected item will be replaced by previous item.
 - Adjust selected item be later, the sequence of selected item will be replaced by the next item.

Adjust selected item be later

Before

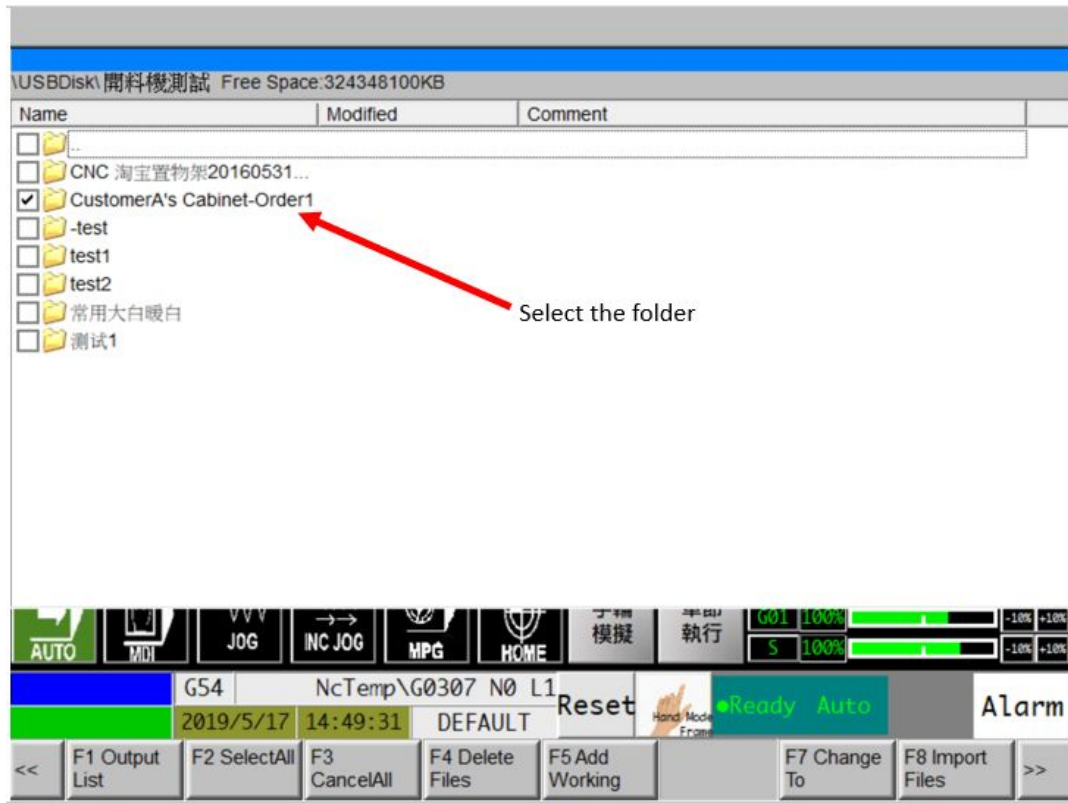


After



SYNTEC

4.4.3 Operator Panel-F3: File Manage

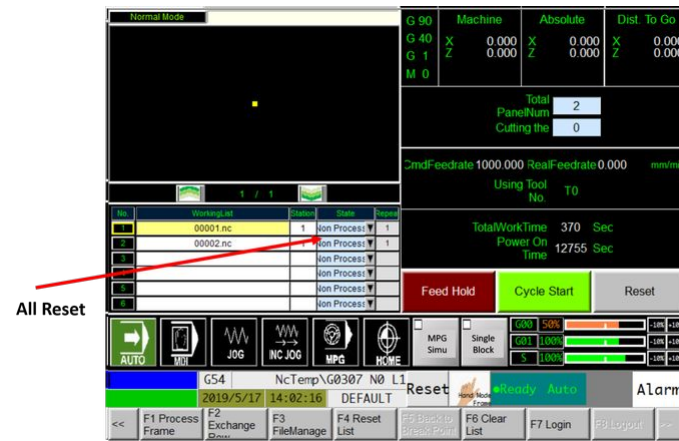


- F1 Output List
 - Generate a new workinglist according the files user selected
 - User can select multiple files or folders.
- F2 Select All
 - Selecte all items at current file page
 - Not include the folder. Only the files will be selected.
- F3 Cancel All
 - Cancel selected items
- F4 Delete Files
 - Delete selected files
- F5 Add Workinglist
 - Add working item to the end of current workinglist
 - For example, there are 5 items in Current Workinglist, after user chooses another 5 item and add to workinglist, there will be 10 items in workinglist.
- F7 Change To USB/Sys.
 - Change the file source between the controller and USB.
 - If user needs to import files from external device like USB, need to use the button change to USB source

- F8 Import Files
 - Import files or folder into controller, depends on what kind of item user select.
 - Just import, no workinglist will be generate after press this button.

4.4.4 Operator Panel-F4: Reset List

- Set all panel's status to be "Non Process"

	Reset List
Before	 The screenshot shows the CNC control interface. At the top, it says 'Normal Mode' and '00002.nc'. Below this is a large black area with a yellow dot. To the right, there are various status indicators: 'Machine', 'Absolute', 'Dist. To Go', 'Total PanelNum', 'Cutting the', 'CmdFeedrate', 'RealFeedrate', 'Using Tool No.', 'TotalWorkTime', and 'Power On Time'. At the bottom, there is a 'Working List' table with columns: No., Working List, Status, State, and Reset. The table shows two items: 00002.nc (completed) and 00001.nc (completed). Below the table are various function keys like 'F1 Process Frame', 'F2 Exchange Program', 'F3 File Manager', 'F4 Reset List', 'F5 Back to Break Point', 'F6 Clear List', 'F7 Login', and 'F8 Logout'.
After	 The screenshot shows the CNC control interface after the reset operation. The 'Working List' table now shows two items: 00001.nc (Non Process) and 00002.nc (Non Process). The 'Status' column for both is 'Non Process'. A red arrow points to the 'F4 Reset List' button with the label 'All Reset'.

4.4.5 Operator Panel-F5: Back to Break Point

Press F5, a pop-up show on screen	
- Press Yes, Change to Break point process mode - Check the focus of the file content is right, and execute.	
- After finishing this process, go back to operator panel - The status will be changed to next process and go on.	

- When an unexpected event happens that causes a break to the processing file and you don't want to do the whole process again
- Users can press this button and the system will go back to the block before the break happened, and continue the working.
- Hint: Please DO NOT press the button at normal condition, this might cause the panel process state to be wrong.**

4.4.6 Operator Panel-F6: Clear List

- Clear workinglist. No item will show on workinglist after use this function.

4.4.7 Operator Panel-F7: Login

- Enter username and password to login to the system
- login different users can have different permissions
- Permissions can be set in "Authority management"

4.4.8 Operator Panel-F8: LogOut

- logout the accessed account

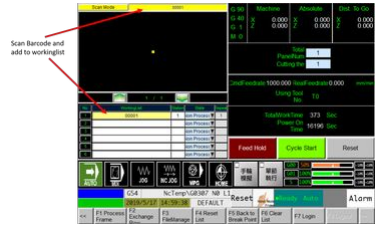
4.4.9 Workinglist Manipulation

- Normal Mode
 - **Select Files/Folder, Output Workinglist**

• Operator Frame(F1) → F3 File Manage • F7: Change to USB/Sys.	
• Select File/Folder which contains the NC files for all order. • Press F1:Output List	
• The items in the folder will show on workinglist • Check the sequence is corresponding to the panel.	
• Execute the process.	

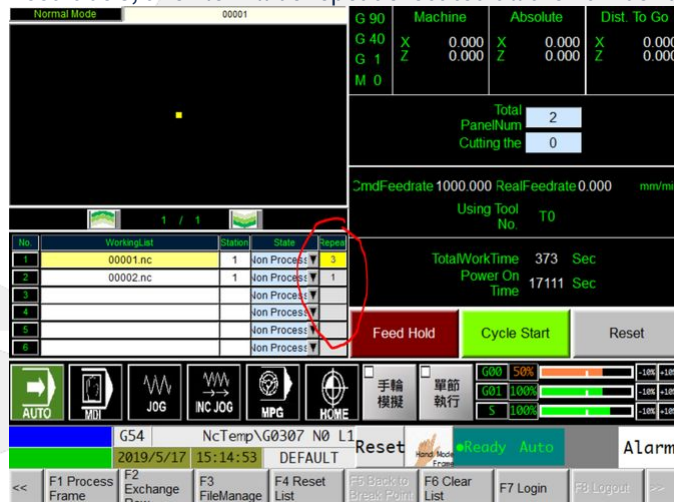
• BarCode Scanning Mode

- Press The "Normal Mode" button, change to "Scan Mode", Press again back to "Normal Mode".
- Scan the barcode and the scanned NCfile will be added to workinglist by sequence, if no files found in system, there will show a pop-up hint

	Figure
ScanMode	
Scanning Barcode and add to Workinglist	

• Set Repeat Times for the Same File

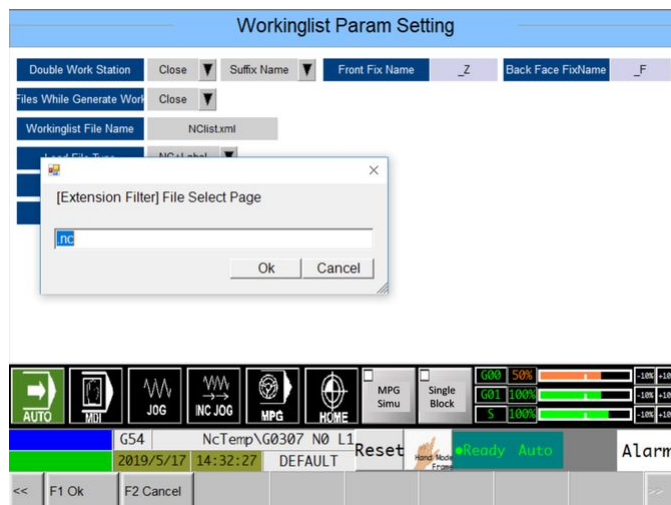
- The default of repeat time is 1, every time the process done, it will minus 1 time.
- If set it be 3, this file will be repeat executed till the number be 0.



4.5 WorkinglistSetting (MainFenu F8 → F7)

- There are several additional functions of workinglist according to customer's request and machine.
- User can set the parameters in this browser.

4.5.1 NCfile Extension Name Setting



- Set the NC files's extension name, if file name is xxxxx.nc, set it be ".nc", if file name is xxxx.anc, set it be ".anc"
- This Setting is "**MUST BE**", if user have wrong setting for the use, the workinglist would show nothing.
- If no extension name in NC files, just set it be empty (no content)

4.5.2 Double Work Station Machine Setting

- This Function is specifically used for machine with 2 stations which allows user place 2 panels on the plate of the machine.
- Sometimes, a panel would come with 2 NCfile cause there 2 faces need to be processed (Back side and Front side).
- The name of NC files need to be similar by means of "fix string" at the start of end, suck like "00001_F.nc", "00001_B.nc"
- Close the function. The station value on workinglist is always 1.

Open the function, the sequence of NCfiles will be auto-arranged for a better and friendly procedure. check the spec as below:

Setting Ways

- There are 3 ways provided to set the double sort function.
 1. Prefix name rule, the string fix at front of ncfiles' name (ex: F_00001.nc)
 2. Suffix name rule, the string fix at end of ncfiles' name (ex: 00001_F.nc)
 3. Normal, the station will show 1,2,1,2,repeatedly, but no sort the file sequence.
- the fix string can be defined on the screen.

Limitation	• MUST CHECK! the file name rule comes with some limitation • Only fix the string at the Start or End of NCfile ! • If eliminate the fix string of the NCfiles for the same panel, the 2 name need to be the same. i. Suffix Example: 0001_F.nc, 0001_Z.nc eliminate "_F" and "_Z", get the same result 0001.nc ii. Prefix Example: 5_0001.nc, 8_0001.nc eliminate "5_" and "8_", get the same result 0001.nc • At least 1 information need to be set in Back side string or Front side string. • Each panel "MUST" have front side program, the back side program is not necessary
Result	• After Setting the double sort setting parameters, the sequence will be arranged automatically while output workinglist. • The work station will show as 1,2,1,2...repeatedly • If the last panel comes with back side and front side program, the station value will show as 1,2,1,2,1,1 or 1,2,1,2,2 • Do the back side first and do the front side

• Example

	Test Example	Sort Result														
1	• Prefix name rule, • Front Side: Z_ • Back Side: F_ a. F_0001.nc, Z_0001.nc b. Z_0002.nc c. Z_0003.nc d. F_0004.nc, Z_0004.nc	<table><tr><th>Sort</th><th>Work Station</th></tr><tr><td>F_0001</td><td>1</td></tr><tr><td>Z_0002</td><td>2</td></tr><tr><td>Z_0001</td><td>1</td></tr><tr><td>Z_0003</td><td>2</td></tr><tr><td>F_0004</td><td>1</td></tr><tr><td>F_0004</td><td>1</td></tr></table>	Sort	Work Station	F_0001	1	Z_0002	2	Z_0001	1	Z_0003	2	F_0004	1	F_0004	1
Sort	Work Station															
F_0001	1															
Z_0002	2															
Z_0001	1															
Z_0003	2															
F_0004	1															
F_0004	1															

	Test Example	Sort Result																
2	- Set different strng comparing to example_1 - Suffix name Rule - Front Side: _5 - Back Side: _8 a. 0001_5.nc , 0001_8.nc b. 0002_5.nc , 0002_8.nc c. 0003_5.nc , d. 0004_5.nc ,	<table><tr><th>Sort</th><th>Work Station</th></tr><tr><td>0001_8</td><td>1</td></tr><tr><td>0002_8</td><td>2</td></tr><tr><td>0001_5</td><td>1</td></tr><tr><td>0002_5</td><td>2</td></tr><tr><td>0003_5</td><td>1</td></tr><tr><td>0004_5</td><td>2</td></tr></table>	Sort	Work Station	0001_8	1	0002_8	2	0001_5	1	0002_5	2	0003_5	1	0004_5	2		
Sort	Work Station																	
0001_8	1																	
0002_8	2																	
0001_5	1																	
0002_5	2																	
0003_5	1																	
0004_5	2																	
3	a. F_0001.nc , Z_0001.nc b. Z_0002.nc c. Z_0003.nc d. F_0004.nc , Z_0004.nc e. Z_0005.nc	<table><tr><th>Sort</th><th>Work Station</th></tr><tr><td>F_0001</td><td>1</td></tr><tr><td>Z_0002</td><td>2</td></tr><tr><td>Z_0001</td><td>1</td></tr><tr><td>Z_0003</td><td>2</td></tr><tr><td>F_0004</td><td>1</td></tr><tr><td>Z_0005</td><td>2</td></tr><tr><td>F_0004</td><td>1</td></tr></table>	Sort	Work Station	F_0001	1	Z_0002	2	Z_0001	1	Z_0003	2	F_0004	1	Z_0005	2	F_0004	1
Sort	Work Station																	
F_0001	1																	
Z_0002	2																	
Z_0001	1																	
Z_0003	2																	
F_0004	1																	
Z_0005	2																	
F_0004	1																	

	Test Example	Sort Result																		
4	• Case of only front side panel got the fix string a. Z_0001.nc b. Z_0002.nc, 0002.nc c. Z_0003.nc, 0003.nc d. Z_0004.nc, 0004.nc e. Z_0005.nc	<table><tr><th>Sort</th><th>Work Station</th></tr><tr><td>Z_0001</td><td>1</td></tr><tr><td>0002</td><td>2</td></tr><tr><td>0003</td><td>1</td></tr><tr><td>Z_0002</td><td>2</td></tr><tr><td>Z0003</td><td>1</td></tr><tr><td>0004</td><td>2</td></tr><tr><td>Z_0005</td><td>1</td></tr><tr><td>Z_0004</td><td>2</td></tr></table>	Sort	Work Station	Z_0001	1	0002	2	0003	1	Z_0002	2	Z0003	1	0004	2	Z_0005	1	Z_0004	2
Sort	Work Station																			
Z_0001	1																			
0002	2																			
0003	1																			
Z_0002	2																			
Z0003	1																			
0004	2																			
Z_0005	1																			
Z_0004	2																			
5	• Case of only back side panel got the fix string a. 0001.nc b. 0002.nc, 0002_F.nc c. 0003.nc, d. 0004.nc, 0004_F.nc	<table><tr><th>Sort</th><th>Work Station</th></tr><tr><td>0001</td><td>1</td></tr><tr><td>0002_F</td><td>2</td></tr><tr><td>0003</td><td>1</td></tr><tr><td>0002</td><td>2</td></tr><tr><td>0004_F</td><td>1</td></tr><tr><td>0004</td><td>1</td></tr></table>	Sort	Work Station	0001	1	0002_F	2	0003	1	0002	2	0004_F	1	0004	1				
Sort	Work Station																			
0001	1																			
0002_F	2																			
0003	1																			
0002	2																			
0004_F	1																			
0004	1																			

4.5.3 Send Files While Generate Workinglist

- Only When user use the dipole connection and manipulate on PC, this setting need to be open.
- Please DO NOT change this setting without technician's comment if you are the endUser of machine.

4.5.4 Panel Row Number

- The number of rows in Workinglist

4.5.5 Label Row Number

- Not the standard function for this machine, please connect Syntec Technician for more information.

4.5.6 Workinglist Q&A

Question	Answer
Why I choose the file/folder, and output list. But I see nothing on the workinglist?	• Step1: Check the extension name setting for ncfiles is corresponded. • Step2: Check if you open the double sort function, if you don't need and it is open, close it. • Step3: Check the load file type in workinglist.
I use the double sort function But the sort result is not what I want??	• Check the setting of double sort function in workinglist param setting. • Check if the ncfile's name rule is out of the specification of the function.

4.6 Program (MainFenu: F2)



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

4.6.1 Execute

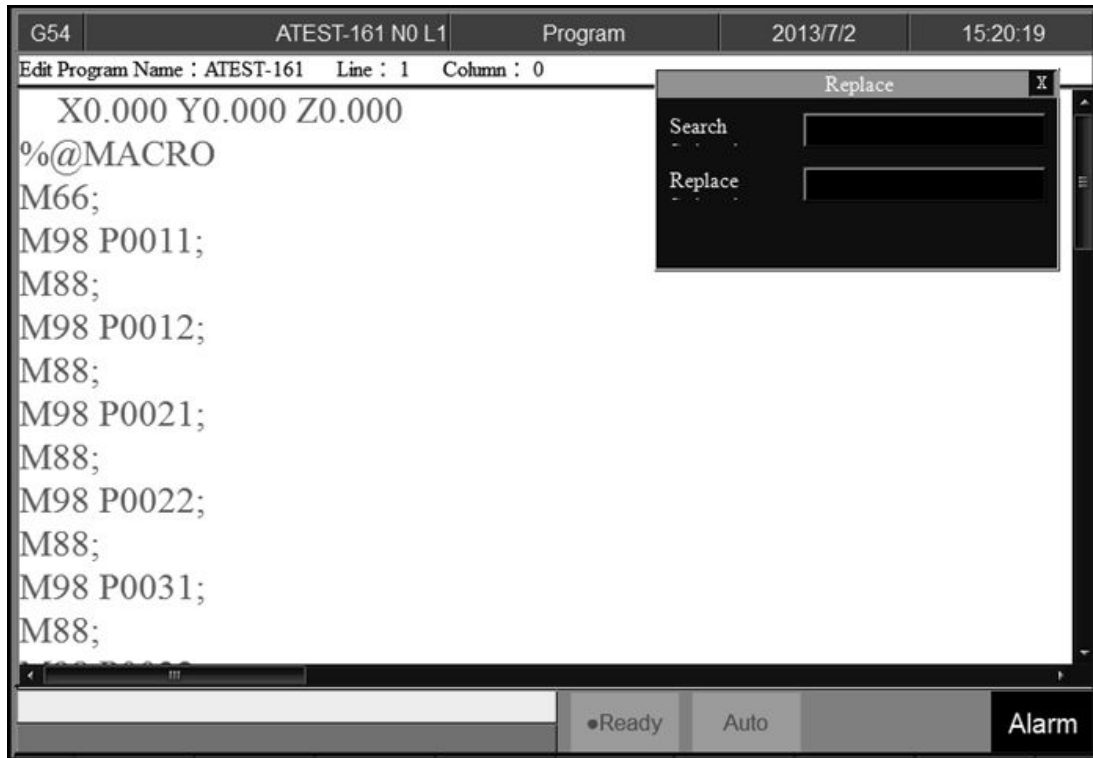
- Command
 - F2 Program→F1 Execute
- Function
 - Execute current program and also change the screen to 「Monitor」 page.
- Note
 - This function will be disabled during machining.

4.6.2 Delete Line

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

SYNTEC

4.6.3 Search/Replace



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

Find Next

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

Replace

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

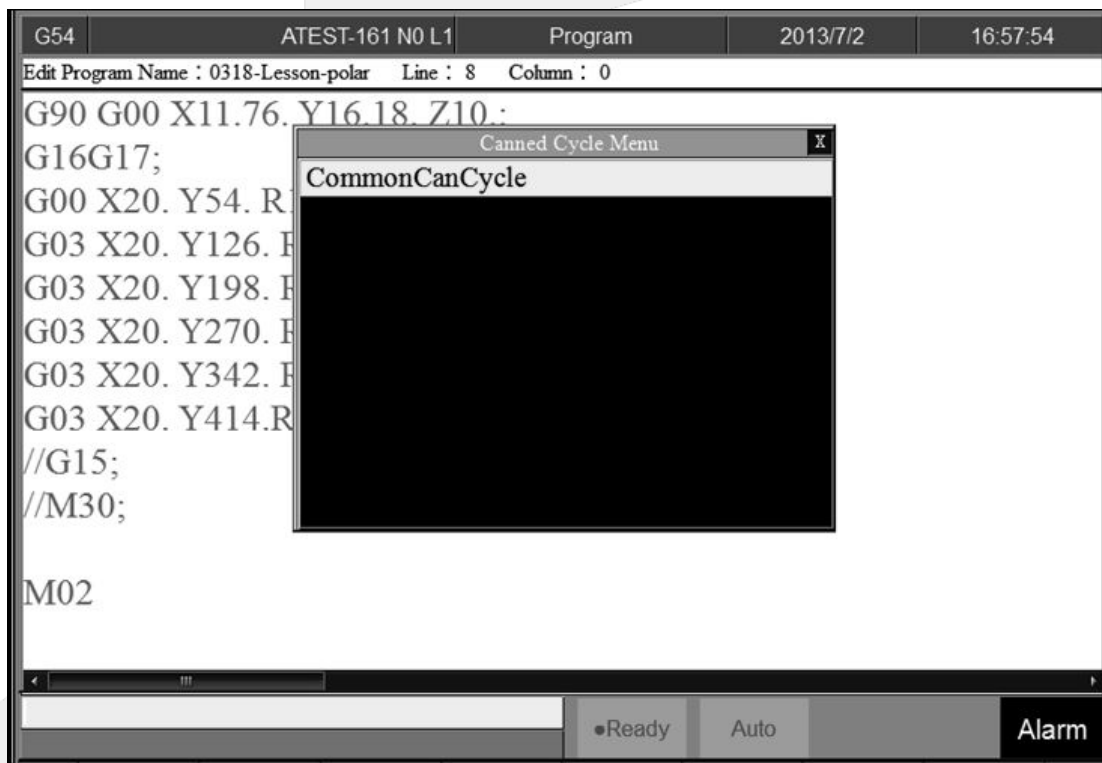
Replace All

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

Modify Setting

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

4.6.4 Can Cycle



- Command
 - 5 Buttons type: F2 Program→Next→F1 Can Cycle
 - 8 Buttons type: F2 Program→F4Can Cycle
- Function
 - Because SYNTEC system provides many kind of G code and different G code has each function. When editing the program, this function can help user to edit G code easily.

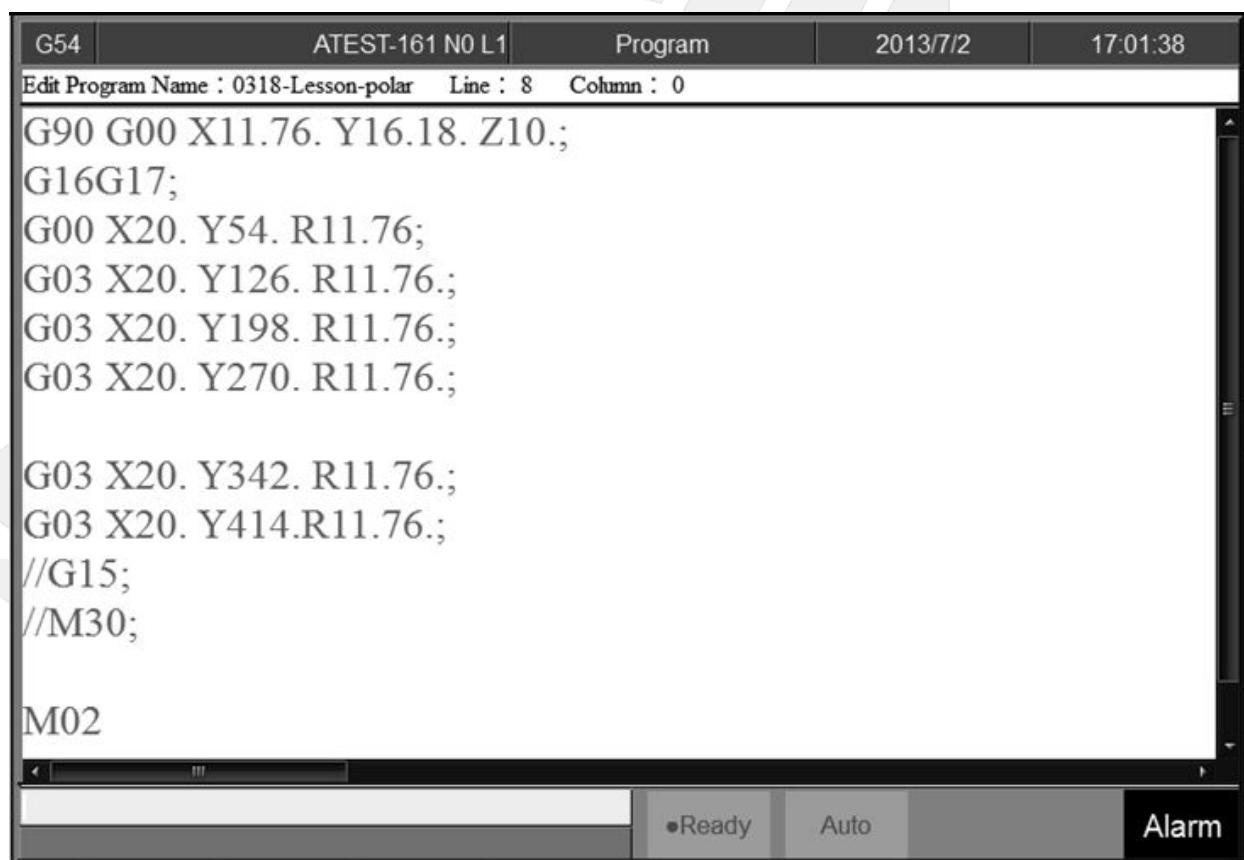
Insert Cycle

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

Edit Cycle

- Command
 - 5 buttons type: F2 Program→Next→F1 Can Cycle→F2 Edit Cycle
 - 8 buttons type: F2 Program→F4 Can Cycle→F2 Edit Cycle
- Function
 - Edit the current cycle.
- Operation Method
 - Move the cursor to the cycle and press 「Edit Cycle」, a modify page will show up. Modify the contents and press 「OK」, the contents of the current cycle will change.

4.6.5 Block Copy



- Path
 - 5 buttons type: F2 Program→Next→F2 Block Copy
 - 8 Buttons type: F2 program→F5 Block Copy
- Function
 - Select, cut, copy and paste more than one line of programs.

Start Line

- Command
 - 5 Buttons: F2 Program→Next→F2 Block Copy→F1 Start Line
 - 8 Buttons: F2 Program→F5Block Copy→F1 Start Line
- Function
 - Define the start line of block.

End Line

- Command
 - 5 Buttons: F2 Program→Next→F2 Block Copy→F2 End Line
 - 8 Buttons: F2 Program→F5Block Copy→F2 End Line
- Function
 - Define the end line of block.

Block Cut

- Command
 - 5 Buttons: F2 Program→Next→F2 Block Copy→F3 Block Cut
 - 8 Buttons: F2 Program→F5Block Copy→F3 Block Cut
- Function
 - Cut the block that had been selected.

Block Copy

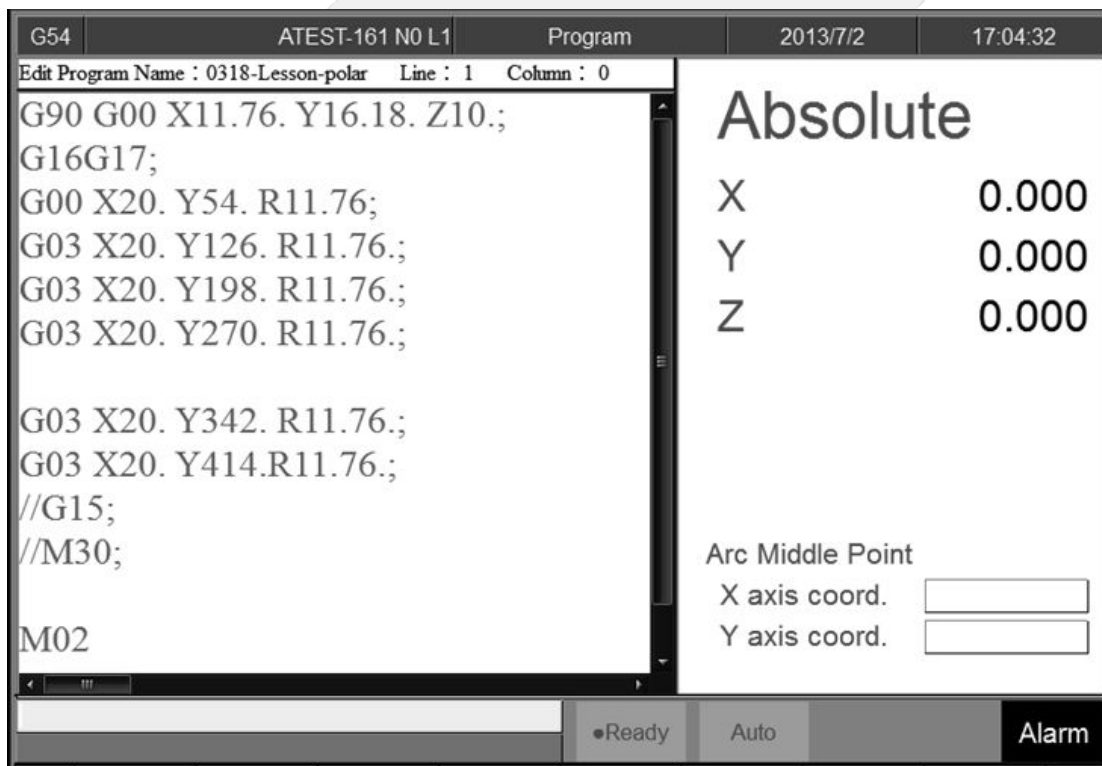
- Command
 - 5 Buttons: F2 Program→Next→F2 Block Copy→F4 Block Copy
 - 8 Buttons: F2 Program→F5Block Copy→F4 Block Copy
- Function
 - Copy the block that had been selected.

Block Paste

- Command
 - 5 Buttons: F2 Program→Next→F2 Block Copy→F5 Block Paste
 - 8 Buttons: F2 Program→F5Block Copy→ F5 Block Paste
- Function
 - Paste the block that had been 「Block Cut」 and 「Block Copy」 .
- Operation Method
 - Move the cursor to the desire line and press 「Start Line」 and function key 「End Line」 enable.
 - Press 【↑】 【↓】 【Page Up】 【Page Down】 to select desired area.
 - Press 「End Line」 ,block between 「Start Line」 and 「End Line」 will be selected.
 - i. →Function key 「End Line」 disable.
 - ii. →Function key 「Block Copy」 enable.

- iii. →Function key 「Block Cut」 enable.
- iv. →If 「Block Cut」 being use, the whole block that had been highlight will be cut off.
- v. →Function key 「Block Copy」 / 「Block Cut」 disable.
- vi. →Function key 「Block Paste」 enable.
- Move the cursor to the desire location and press 「Block Paste」 , the content that had been cut or copy will paste at the cursor location.
- If 「Block Copy」 is used, the block that had been selected will not disappear.
- Note
 - If 「Block Cut」 is use, and do not paste the content immediately, the cut program will disappear.
 - The contents of 「Block Cut」 can be pasted for only one time but the contents of 「Block Copy」 can be pasted for many times.

4.6.6 Teach



- Command
 - 5 Buttons type: F2 Program→Next→F3 Teach
 - 8 Buttons type: F2 Program→F6 Teach
- Function
 - Move the machine table with 『MPG』 / 『JOG』 / 『INJOG』 to the destination and use 『Teach』 function, input the current absolute coordinate value to NC program.
 - Omit the manual input problem.

Rapid Teach

- Command
 - 5 Buttons type: F2 Program→Next→F3 Teach→F1 Rapid Teach

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

Line Cut Teach

- Command
 - 5 Buttons type: F2 Program→Next→F3 Teach→F2 Line Cut Teach
 - 8 Buttons type: F2 Program→Next→F6 Teach→F2 Line Cut Teach
- Function
 - Add the current absolute coordinate as the value of 「G01 Linear Cutting」 function in current program.

Arc Cut Teach

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

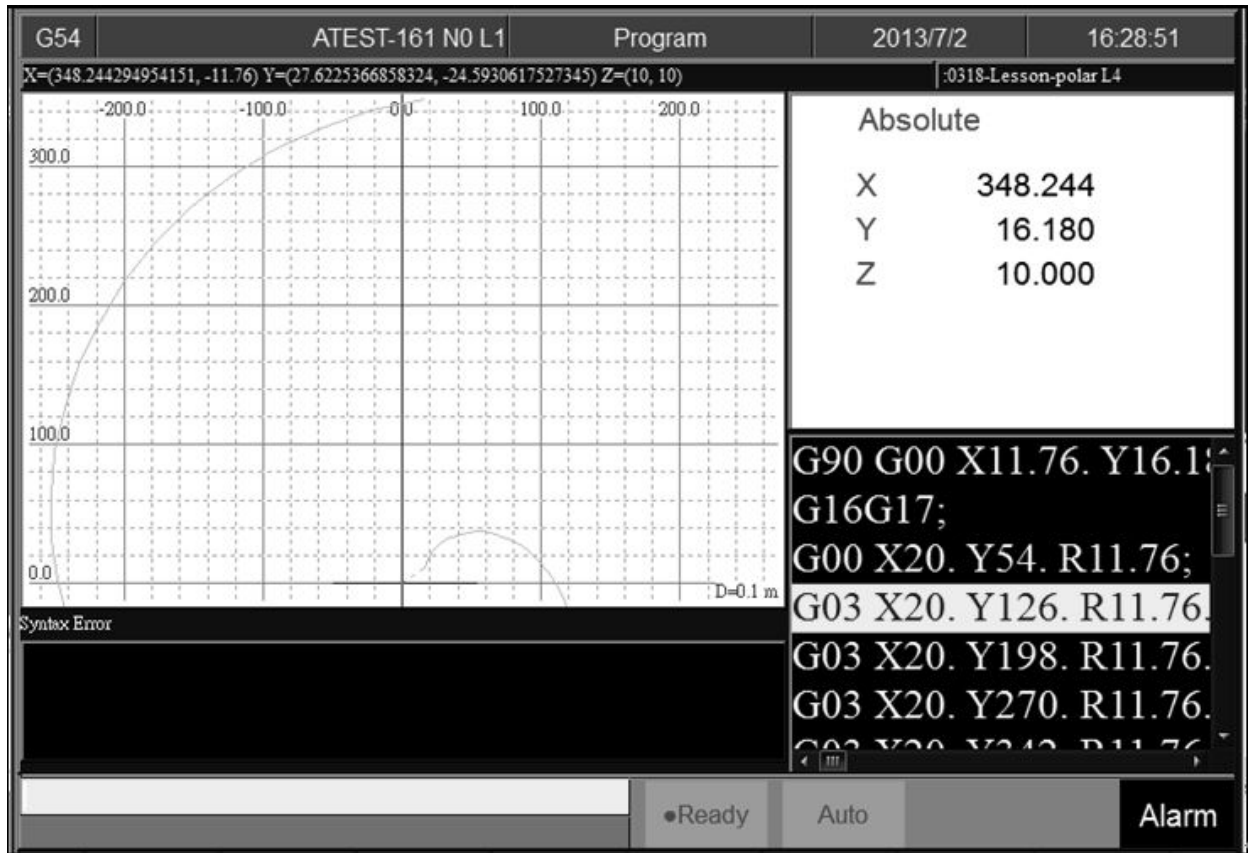
Cancel Arc Middle

- Command
 - 5 Buttons type: F2 Program→Next→F3 Teach→F4 Cancel Arc Middle
 - 8 Buttons type: F2 Program→F6 Teach→F4 Cancel Arc Middle
- Function
 - Clear the arc middle that had been set.
 - If the arc middle are not being set, this function will not enable.

Point Teach

- Command
 - 5 buttons type: F2 Program→Next→F3 Teach→F5 Point Teach
 - 8 buttons type: F2 Program→F6 Teach→F5 Point Teach
- Function
 - Move the worktable to the arc center and press 「Point Teach」, current absolute coordinate will be input into current cursor location.

4.6.7 Simulation



- Command
 - 5 buttons type: F2 Program→F5 Simulation
 - 8 buttons type: F2 Program→F7Simulation
- Function
 - Program simulation for the actual machining route.
 - Capability of debug.
 - Default display range will be the span of the full program.
 - Simulation setting can be modified by F5 「simulate Setting」 ..

Step

- Command
 - 5 Buttons type: F2 Program→F5 Simulation→F1 Step
 - 8 Buttons type: F2 Program→F7 Simulation→F1 Step
- Function
 - Simulate the program block by block.
 - Monitor the variation of the coordinate for single block.

Continue

- Command
 - 5 Buttons type: F2 Program→F5 Simulation→F2 Continue

- 8 Buttons type: F2 Program→F7 Simulation→F2 Continue
- Function
 - System will scan all of the programs and then do the simulation.

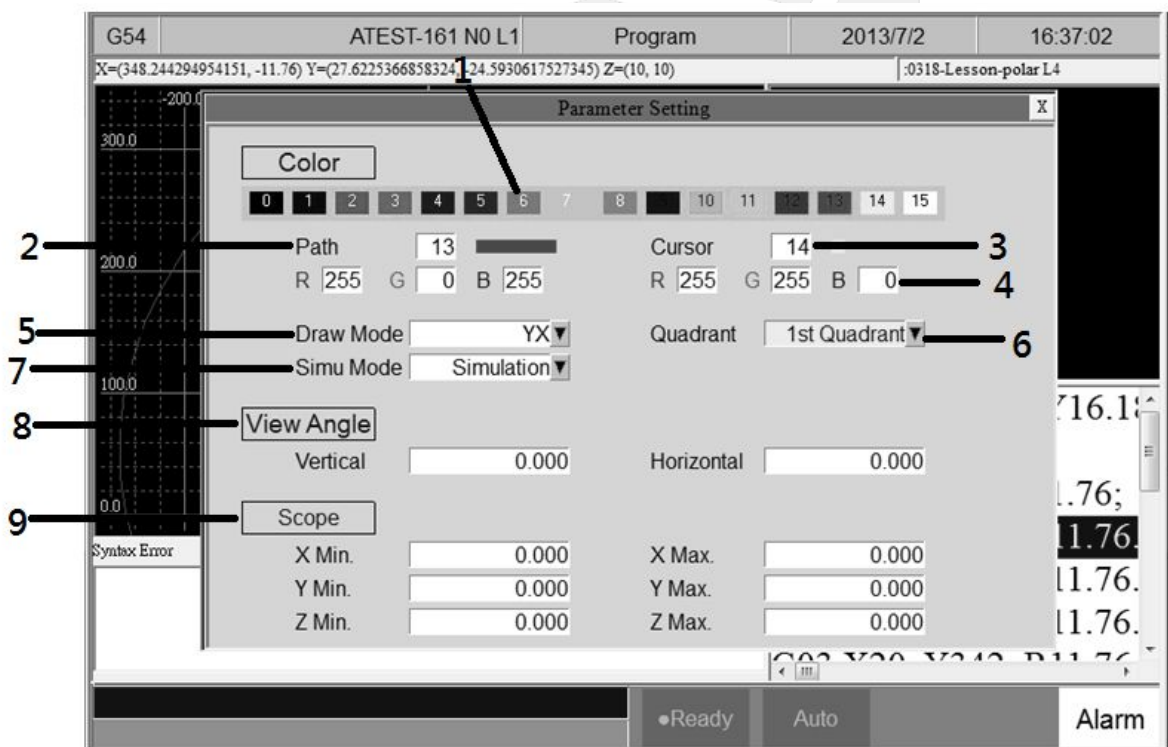
Zoom

- Command
 - 5 Buttons type: F2 Program→F5 Simulation→F3 Zoom
 - 8 Buttons type: F2 Program→F7 Simulation→F3 Zoom
- Function
 - Zoom in/out the simulation window.
- Operation Method
 - Press F3 「Zoom」, there will be a block show up. Use 【↑】 【↓】 【←】 【→】 can move the window up, down, left and right.
 - Use 【Page Up】 【Page Down】 to change the zooming area.
 - Press 【ENTER】 to check the result.

Graph Reset

- Command
 - 5 buttons type: F2 Program→F5 Simulation→F4 Graph Reset
 - 8 buttons type: F2 Program→F7 Simulation→F4 Graph Reset
- Function
 - Reset the simulation result to default.

Simu. Setting



- Command
 - 5 buttons type: F2 Program→F5 Simulation→F5 Simu. Setting
 - 8 buttons type: F2 Program→F7 Simulation→F5 Simu. Setting
- Function
 - Setting the relative simulation item.
- Simulation parameter
 - a. Color
 - Provide 16 different colors (Setting 0~15).
 - b. Path Color
 - Simulation path color.
 - Provide 16 different colors (Setting 0~15).
 - c. Cursor Color
 - Color of cursor point
 - Provide 16 different colors (Setting 0~15).
 - d. RGB Value
 - Except the 16 default color, user can define the color by them self.
 - e. Draw Mode
 - User can define the profile simulate plane.
 - Plane can define are as below.

• XYZ	• YX
• XY	• ZY
• YZ	• XZ
• ZX	

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

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

4.6.8 File Manager

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

- Command
 - 5 Buttons type: F2 Program→F4 File Manager
 - 8 Buttons type: F2 Program→F8 File Manager
- Function
 - This function key can manage all of the NC files within the data storage device. The device can be setting with Pr3213.
- Operation Method
 - Use 【↑】 【↓】 on the key pad to move the cursor to anywhere on the screen for editing purpose.
 - With 【Page Up】 【Page Down】 to switch the cursor between pages.
 - Press 【ENTER】 on the key pad, to assign the current cursor file as the execute file, screen will show up with the program content and can enable to edit the program.

New File

- Command
 - 5 Buttons type: F2 Program→F4 File Manager→F1 New File
 - 8 Buttons type: F2 Program→F8 File Manager→F1 New File
- Function
 - Open a new file, that file will be the current edit file.
- Operation Method
 - Press 「New File」 function key, a dialog box will appear, enter the new file name and press **【ENTER】** .
- Note
 - Default file name has no file extension. If user want to create a new file with file extension such as **.NC, just enter the extension (.NC)** as well.
 - The length of file name cannot be longer than 32 characters (include file extension)

Copy File

- Command
 - 5 Buttons type: F2 Program→F4 File Manager→F2 Copy File
 - 8 Buttons type: F2 Program→F8 File Manager→F2 Copy File
- Function
 - Copy the file that remarked by cursor.
- Operation Method
 - Use **【↑】 【↓】** to move the cursor to the file that want to copy.
 - Press 「Copy File」 function key.
 - A dialog box will appear, enter the new file name.
- Note
 - Default file name has no file extension. If user want to create a new file with file extension such as **.NC, just enter the extension (.NC)** as well.
 - The length of file name cannot be longer than 32 characters (include file extension)

Delete File

- Command
 - 5 Buttons type:F2 Program→F4 File Manager→F3 Delete File
 - 8 Buttons type:F2 Program→F8 File Manager→F3 Delete File
- Function
 - Delete file that remarked by cursor.
- Operation Method
 - Press F3 「Delete File」 , check box will show up in front of the NC file within the 「File Manager」 monitor page. Use **【↑】 【↓】** to select the delete file.
- Sub-function Key
 - Select: Select file, can select more than one file and also can cancel the selection of one file.
 - Select All: Select all file.
 - Cancel Select: Deselect all files.
 - Delete File: Delete all of the selected files.
 - Delete All: Delete all file within data storage device.
- Note
 - Current Programming and machining file cannot be deleted.

File Import

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

File Export

- Command
 - 5 Buttons type: F2 Program→F4 File Manager→F4 File Transfer→F2 File Export
 - 8 Buttons type: F2 Program→F8 File Manager→F4 File Transfer→F2 File Export
- Function
 - Export file within controller to outer device.
- Function Page Explanation
 - The upper block shows the outer devices selection with the following choice.
 - USBDisk
 - DiskA
 - Network
 - USBDisk2
 - Left column shows data structure of the outer device.
 - Right column shows data structure of the inner storage of the controller.
- Sub-function Explanation
 - Copy: Copy the remarked file from the outer device to the controller.
 - Select: Select or deselect each file. (Not available for folder)
 - Select All: Select all files.
 - Cancel Select: Deselect all files.

- Device Change: Change outer device selection.
- Operation Method
 - Press 「File Export」, a dialog box will appear.
 - Default outer device is USBDisk.
 - If you want to change the destination device, press F5 「Device Change」, switch the cursor to the desire device and press 【Enter】, then the below left column data structure will change, according to the selected device.
 - Use 【↑】 【↓】 to select file.
 - Move the cursor to the export file and press F2 「Select」 or 【Space】 to remark file.
 - After remarked all of the export file, press F1 「Copy」 then all of the remarked file will export from controller to outer device.

Execute

- Command
 - 5 Buttons type: F2 Program→F4 File Manager→F5 Execute
 - 8 Buttons type: F2 Program→F8 File Manager→F5 Execute
- Function
 - Execute current program. and also change the screen to the 「monitor」 page.
- Operation Method
 - Use 【↑】 【↓】 to select file, and then press 「Execute」 the selected file will be the executive file.
- Note
 - This function is invalid when machining.

Choose DNC File

- Command
 - 5 Buttons type: F2 Program→F4 File Manager→Next→F6 Execute
 - 8 Buttons type: F2 Program→F8 File Manager→F6 Execute
- Function
 - Execute program stored in external devices, such as USB/Network.
- Operation Method
 - Use 【↑】 【↓】 to select file, and then press 「Confirm」 the selected file to be executed.
 - Title of selected file will be showed : DNC + file name
- Note
 - Valid from - 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 a file reading fail, the control will retry 100 times before pop up a error message
 - A DNC program must use M30 as the end of program
 - USB DNC function does not support MACRO function
 - USB DNC function supports O888800, O999900, and Pr3851=0 functions (after 2.2.5)
 - When M98, G65, or G66 is used, the subprogram must be stored in controller

4.7 Offset/Setting (MainFenu: F3)

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

- 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.
- When it is not able to modify, alarm windows will

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.→F1 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→F1 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.→F4 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 5.000
 - 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 5.000
 - 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→F1 Workpiece Coord.→F5 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 avoid 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.

4.7.2 Tool Set

4.7.3 Single-axis tool table (default):

G54		N0 L1		Offset/ Setting		2022/7/13		14:40:27		Admin	
Input Mode(Absolute (I)Increment (Z)Measure						Machine					
Absolute T 0						X 0.000					
						Y 0.000					
						Z 0.000					
						C 0.000					
Radius(D) Length(H)						Absolute					
Geometry		Wear		Geometry		Wear		X 0.000			
								Y 0.000			
1	0.000	0.000	0.000	0.000	0.000	0.000	0.000	Z 0.000			
2	0.000	0.000	0.000	0.000	0.000	0.000	0.000	C 0.000			
3	0.000	0.000	0.000	0.000	0.000	0.000	0.000				
4	0.000	0.000	0.000	0.000	0.000	0.000	0.000				
5	0.000	0.000	0.000	0.000	0.000	0.000	0.000				
6	0.000	0.000	0.000	0.000	0.000	0.000	0.000				
7	0.000	0.000	0.000	0.000	0.000	0.000	0.000				
8	0.000	0.000	0.000	0.000	0.000	0.000	0.000				
						Relative					
						X 0.000					
						Y 0.000					
						Z 0.000					
						C 0.000					
						Alarm					
Clear Z Coord.		Set Tool Mach. Coord.		Set Tool Rel. Coord.				Tool No.		Tool Life Manag.	

- 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
 - Radius Geometry: G41/G42 Cutter radius Dn compensation (please input radius, not diameter)
 - Radius 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. Input 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.
 - Input with axis name is not supported. ex: Z10., Z1=20.
 - 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
 - F3 Offset/Setting → F2 Tool Set → F1 Clear Z Coord.
- Function
 - Clear the current Z-axis relative coordinate value.

Tool No.

- Path
 - F3 Offset/Setting → F2 Tool Set → F7 Tool No.
- Function
 - Please refer to chapter 3.4.4 for details

Tool Life Manag.

- Path
 - 10+8 key: (Right column) Offset/Set → F2 (Tool Set) → F8 (Tool Life Manag).
 - 8 key: F3 Offset/Setting → F2 Tool Set → F8 Tool Life Manag.
 - 5 key: F3 Offset/Setting → F2 Tool Set → F5 Tool Life Manag.
- Function
 - Please refer to chapter 3.4.5 for details

SYNTEC

4.7.4 Multi Z-axis tool table:

G54	N0 L1	Offset/ Setting	2021/3/5	14:09:55	Admin
Input Mode (A)bsolute (I)ncrement					
Absolute T 0					
ZLength Z1Length					
1	0.000	0.000			
2	0.000	0.000			
3	0.000	0.000			
4	0.000	0.000			
5	0.000	0.000			
6	0.000	0.000			
7	0.000	0.000			
8	0.000	0.000			
			Machine		
			X 0.000		
			Y 0.000		
			Z 0.000		
			Z1 0.000		
			C 0.000		
			Absolute		
			X 0.000		
			Y 0.000		
			Z 0.000		
			Z1 0.000		
			C 0.000		
			Alarm		
Tool Wear Set	Tool Length Set	Set Tool Radius			
			Tool No. Tool Life Manag.		

- Path
 - 5key, 8key: F3 Offset/Setting → F2 Tool Set
 - 10+8 key: G3 Offset/Setting → F2 Tool Set
- Valid Version
 - From 10.118.11
- Function
 - Enable condition: Tool length compensation mode using H-code multi-axis (Please refer to Pr3816 *Tool length compensation mode)
 - Multiple Z-axes can be displayed and set at the same time.
The displayed axes must meet the following three conditions at the same time:
 - Must be Z-axis
 - Must be a linear axis
 - The axis must be visible (not hidden)
 - Provides 96 sets of tool setups.
- Operation method
 - a. Move the cursor with arrow keys 【↑】 【↓】 【←】 【→】
 - b. Use 【PageUp】 【PageDown】 to switch the page.
 - c. Input 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.
 - Input with axis name is not supported. ex: Z10.、 Z1=20.
 - 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
 - Press F2 Set Tool Length → 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)
 - Press F2 Set Tool Length → 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
 - F3 Offset/Setting → F2 Tool Set → F2 Set Tool Length → F1 Clear Z Coord.
- Function
 - Clear the current Z-axis relative coordinate value.

Tool No.

- Path
 - F3 Offset/Setting → F2 Tool Set → F7 Tool No.
- Function
 - Please refer to chapter 3.4.4 for details.

Tool Life Manag.

- Path
 - 10+8 key: (Right column) Offset/Set → F2 (Tool Set) → F8 (Tool Life Manag).
 - 8 key: F3 Offset/Setting → F2 Tool Set → F8 Tool Life Manag.
 - 5 key: F3 Offset/Setting → F2 Tool Set → F5 Tool Life Manag.
- Function
 - Please refer to chapter 3.4.5 for details

4.7.5 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

- 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.

G54	TEST N0 L1	Monitor	2013/7/2	20:05:46
Absolute X -20.000 Z 100.000 Y 100.000		Dist. To Go X 0.000 Z 0.000 Y 0.000		G Code G1 G18 G90 G95 G71 G40 G49 Run Time 0 : 0 : 0 Accum 0 : 0 Run Time G00 Over 100 % G01 Over 100 % Spd Over 100 %
Feedrate 0.0mm/min Spindle 1000RPM		T 0000 Start Block No. 1		TotalAcumPar 0 Part Count 0
G00 X100.000 Y100.000 Z100.000 G0X100.Y100.Z100.C100. G00 X0.000 Z100.000; G04X5.S1000 G00 X100.000				
Ready Not Select Alarm				

- 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.

- 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

Part Count	
Accu. Prt.	0
Req. Parts	0
Parameter Set	
Feed Rate	1000.000 mm/min
Spindle	1000 RPM
G120.1 P0	Line NO. 0
	Col. NO. 0

- Screen 2: G code, Time, Override

G Code G1	Run Time	0 : 0 : 0
G17 G90 G94	Accu. Time	0 : 0 : 0
G71 G40 G49	G00 Over	50 %
	G01 Over	100 %
	Spd Over	100 %
Accu. Prt.	0	T 0 D 0 H 0
Prt. Count	0	Start Block 1

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

	Speed	Override	Loading
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 6 axes)

	Speed	Override	Loading
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

SYNTEC

G54		TEST NO L1		Monitor		2013/7/2		20:05:46	
Absolute		Dist. To Go		G Code		Run Time		0 : 0	
X	-20.000	X	0.000	G1		Accum		0 : 0	
Z	100.000	Z	0.000	G18 G90 G95		Run Time		0 : 0	
Y	100.000	Y	0.000	G71 G40 G49		G00 Over		100 %	
						G01 Over		100 %	
						Spd Over		100 %	
Feedrate		0.0mm/min		T 0000		TotalAcumPar		0	
Spindle		1000RPM		Start Block No.		1		Part Count	
G00 X100.000 Y100.000 Z100.000									
G0X100.Y100.Z100.C100.									
G00 X0.000 Z100.000;									
G04X5.S1000									
G00 X100.000									
Ready Not Select Alarm									

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

- This block will display current machining program
- Yellow bar indicate to the current running block.

- Function
 - It is overlap with 「Process Setting」 .
 - Press 「Parameter Set」 can switch display information.
- Description
 - a. G Code
 - It will show the G code under machining
 - b. Run Time Accum
 - Processing time for this workpiece now
 - c. Run Time

- Total machining process time
- d. Percentage ratio
 - G00 percentage
 - G01 percentage
 - Spindle speed percentage
- e. Total Acum Par
 - Total work pieces that had been finished.
 - System won't do any initialized action automatically.
 - If you want to do the initialization by manual, press 「Parameter Set」 switch to 「Part count」, set the 「Total Acum Par」 as 0.Part Count
- f. Part Count
 - Count no. will begin from zero when the program is running.
 - Total work pieces number machined by CNC
- g. Start Block
 - We can set the start block of machining process.
 - n: Set the start line number as n. (Ex. 20)
 - L+n: Set the start line number as n. (Ex.L20)
 - N+n: Search N+n located line number and then assign that line as the start line (Ex. N3).
 - T+n: Search T+n located line number and assign it as the start line (Ex. T01).
 - If line number is out of max line number, then it will assign to the last line.
 - Please refer to 3.4.3.4 break point initialization about start block go back.
- h. Tool Data
 - T
 - 4 numbers
 - The first two code are the tool no..
 - The last two code are the tool compensate no.

Display Area of Machining Setting

- Description
 - This area is overlap with 「Machining Information」, press F4 「Parameter Set」 to change the displays.
- Explanation of Display:
 - a. Interrupt Line No.
 - Display the last interrupted serial number (N)
 - b. Interrupt Colum No.
 - Display the last interrupted line number (L)
 - c. Spindle speed
 - Speed of spindle.
 - It is allow to setting when system is busy. Moreover, it will be enabled immediately
- Feed rate
 - Set the speed of the feed rate..
 - It is allow to setting when system is busy, but the value will be updated after completely executing processing block.
- Total AcumPar
 - Total work pieces number machined by CNCSystem cannot automatically reset this value to zero
- Part count
 - Setting current work pieces no.
 - Count no. will begin from zero when the running program is change.
 - When CNC executes M code defined by parameter 3804, part count would be added 1 and run time will be reset to 0. When required part number is reached, system will change to halt status.

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

Simulation Area

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

4.8.2 Open File to 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.

4.8.3 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.

4.8.4 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.

4.8.5 Parameter 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"

4.8.6 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.

4.8.7 Work Record

- Command:
 - 5 Buttons type: F4 monitor-> Next->F2 work record
 - 8 Buttons type: F4 monitor-> F7 work record
- Description
 - Check current machining record and export to external storage device.

4.8.8 Clear Acum Cycle Time

- Command:
 - 5 Buttons type: F4 monitor-> Next->F3 Clear Acum Cycle time
 - 8 Buttons type: F4 monitor ->F8 Clear Acum Cycle time
- Function
 - Clear the accumulative time

4.8.9 Graph Adjust

- Command:
 - 5 Buttons type: F4 monitor-> Next->F4 Graph adjust
 - 8 Buttons type: F4 monitor-> Next->F1 Graph adjust
- Description
 - Zoom in/out simulation graph, this function will enable under 「Simulation Switch」 is open.
- Operation
 - Please refer to 1.4.7 「Simulation」 .

4.9 Maintain (MainFenu: F5)

- Path
 - 5 key, 8 key: F5 Maintain
 - 10+8 key: G5 Maintain
- Explanation

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

4.9.1 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 → F1 Pending Alarm
- Explanation
 - Display current system alarm.
 - Up to 20 alarms of CNC and driver can be displayed.

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

Alarm Analysis

Available versions: 10.118.42S、10.118.48D、10.118.50 and above.

CNC Monitor v10.118.48					
G54		123 N0 L1		Alarm	2021/9/13 17:54:10 DEFAULT
No.	Module	ID	Cnt.	Issue Time	Content
1	Motion	81	32	21/08/17 19:13:47	CAxis Poor contact of communication wire
2	Motion	80	32	21/08/17 19:13:47	ZAxis Poor contact of communication wire
3	Operation	5	8	21/08/17 19:13:47	YAxis Poor contact of communication wire
4	MLC	87	3	21/08/17 19:13:47	XAxis Poor contact of communication wire
5	Operation	21	1	21/08/17 18:20:36	CAxis Poor contact of communication wire
				21/08/17 18:20:36	ZAxis Poor contact of communication wire
				21/08/17 18:20:36	YAxis Poor contact of communication wire
				21/08/17 18:20:36	XAxis Poor contact of communication wire
				21/08/13 18:33:57	CAxis Poor contact of communication wire
				21/08/13 18:33:57	ZAxis Poor contact of communication wire
				21/08/13 18:33:57	YAxis Poor contact of communication wire
				21/08/13 18:33:57	XAxis Poor contact of communication wire
				21/08/13 18:32:55	CAxis Poor contact of communication wire
				21/08/13 18:32:55	ZAxis Poor contact of communication wire
				21/08/13 18:32:55	YAxis Poor contact of communication wire
				21/08/13 18:32:55	XAxis Poor contact of communication wire
				21/08/13 18:22:58	CAxis Poor contact of communication wire
				21/08/13 18:22:58	ZAxis Poor contact of communication wire
				21/08/13 18:22:58	YAxis Poor contact of communication wire
				21/08/13 18:22:58	XAxis Poor contact of communication wire
				21/08/13 18:21:20	CAxis Poor contact of communication wire
				21/08/13 18:21:20	ZAxis Poor contact of communication wire
				21/08/13 18:21:19	YAxis Poor contact of communication wire
				21/08/13 18:21:19	XAxis Poor contact of communication wire
				21/08/13 18:19:45	CAxis Poor contact of communication wire

Ready
Auto
Alarm

Pending Alarm
History Alarm
Save Alarm
Analyze All
Analyze CNC

- Descriptions

Alarm Analysis page has two parts:

- a. Left- Alarm category list

- Display **No.**, **Module**, **ID**, **Cnt.** informations.
- Display categories of alarms within 30-Days.
- Cnt. means how many times the alarm is triggered within 30-Days.
- Display categories in descending order of Cnt..

- b. Right-Alarm detail list

- Display history of the alarm selected in the left list.
 - History records is not limited in 30-Days.
 - History records before 30-Days will be displayed in grey text color.
- Display **IssueTime**, **Content** informations

- c. Operations

- Use up/down key to move focus.
- Use PageUp/Down key to switch pages.
- Change [Alarm category list] focus to refresh right-side detail records.
- Use F9/F12 to view help of the alarm.
- Use right/left key to switch focus between [Alarm category list] and [Alarm detail list]

Analyze All

- Path

- 5 key, 8 key: F5 Maintain → F1 Alarm → F4 Analyze All
- 10+8 key: G5 Maintain → F1 Alarm → F4 Analyze All
- Explanation
 - Analyze alarms of all types.

Analyze CNC

- Path
 - 5 key, 8 key: F5 Maintain → F1 Alarm → F5 Analyze CNC
 - 10+8 key: G5 Maintain → F1 Alarm → F5 Analyze CNC
- Explanation
 - Analyze alarms of CNC type.
 - *The **CNC alarms** means all alarms excluding the **MLC alarms**.

4.9.2 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

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

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
 - 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
 - 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 NO L1	Network Setting	2013/7/2	20:45:58
-----	------------	-----------------	----------	----------

Kernel Server Setting

Start server while boot

TimeOut (milisec)

	●Ready	Not Select	Alarm
--	--------	------------	-------

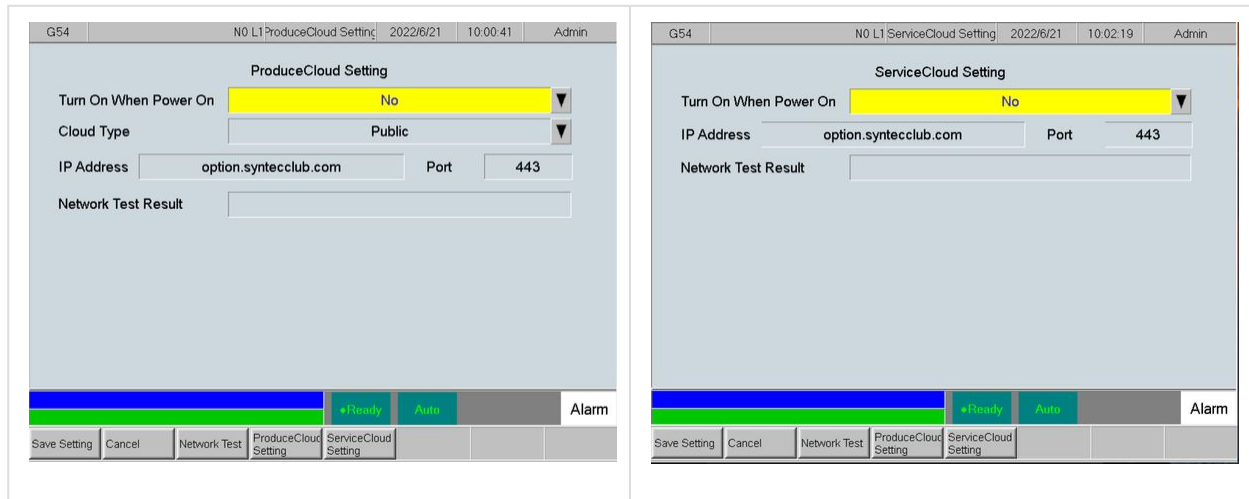
- Path
 - F5 Maintain → F2 Set Kernel Server → F5 Set Kernel Server
- Explanation
 - Set the related function of kernel server.
- Parameters
 - Start server while boot
 - Start the server after booting or not
 - Timeout(ms)
 - Set the acceptable timeout for connection failure

Start Server

- Path

- F5 Maintain → F2 Set Kernel Server → F5 Set Kernel Server → F1 Start Server
- Explanation
 - Start the kernel server immediately.

CloudAgent Setting



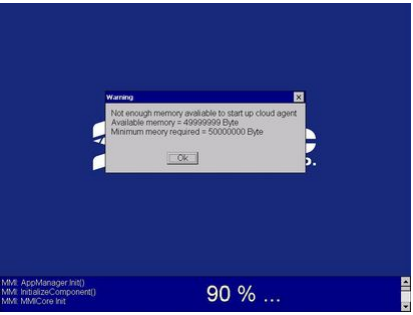
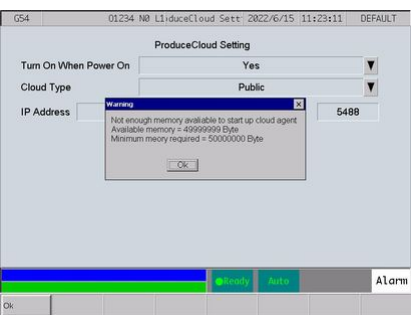
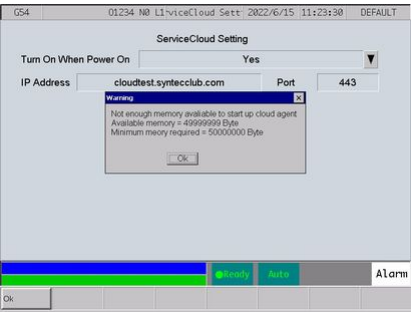
- Path
 - 5 key: F5 Maintain → F2 Set Kernel Server → F3 CloudAgent Setting
 - 8 key: F5 Maintain → F2 Set Kernel Server → F3 CloudAgent Setting
 - 10+8 key: G5 Maintain → F2 Set Kernel Server → F3 CloudAgent Setting
- Explanation
 - Set up the parameters of ProduceCloud and ServiceCloud.
 - Require enough memory to start up CloudAgent. (REF : [Appendix1](#))
 - Network Test function (REF : [Appendix2](#))
 - All setting changes take effect after rebooting.
 - From 10.118.44H, 10.118.46B, 10.118.48M, 10.118.52G, 10.118.56A, 10.118.57 (included).
- Description of parameters
 - Turn On When Power On : Yes/No
 - Default Setting is "No". If choose "Yes", it will start up when power on.
 - Cloud Type : Public Cloud/Private Cloud
 - Only Support "ProduceCloud Setting".
 - Choose public cloud or private cloud according to server.
 - IP Address:
 - Input IP address of server. The default setting is option.syntecclub.com.
 - Port:
 - The default setting is 443.
- Description of fenu button
 - F1: Save Setting
 - Save setting and return to "Set Kernel Server". The changes of produce cloud and service cloud need to be saved separately.
 - F2: Cancel
 - Cancel and return to "Set Kernel Server".
 - F3: Network Test

- Doing a network test according to the IP address, and display the test result on the screen
- F4: ProduceCloud Setting
 - Switch screen to "ProduceCloud Setting".
- F5: ServiceCloud Setting
 - Switch screen to "ServiceCloud Setting".
- Appendix1 :

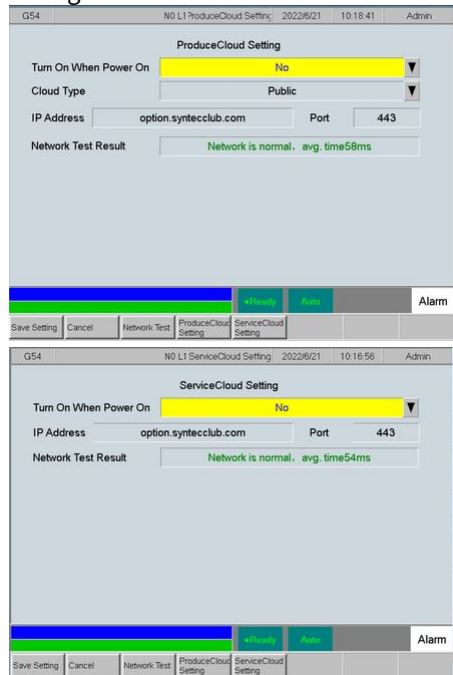
Version	Minimum available memory	Supplemental instruction
From 10.118.44H, 10.118.46B, 10.118.48M, 10.118.52G, 10.118.56A, 10.118.57	30MB	Require at least 30MB of memory space to start up ProduceCloud or ServiceCloud.



SYNTEC

Version	Minimum available memory	Supplemental instruction
From 10.118.44J, 10.118.46J, 10.118.48U, 10.118.52O, 10.118.56I, 10.118.60C	50MB	1.Require at least 50MB of memory space to start up ProduceCloud or ServiceCloud.  2. Setting "Turn On When Power On" Choosing "Yes" will do memory check, if less than 50MB show the warning message and set the value to "No".  

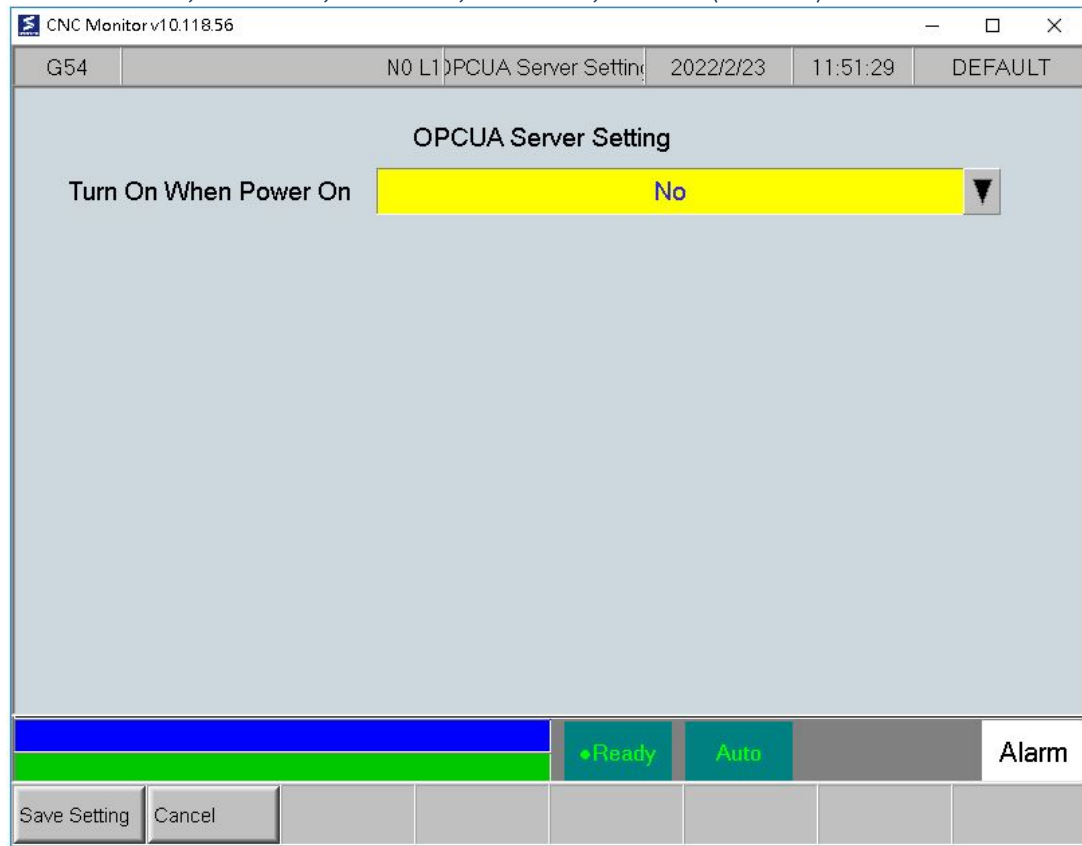
- Appendix2 :

Version	Supplemental instruction
From 10.118.44J, 10.118.46J, 10.118.48U, 10.118.52O, 10.118.56I, 10.118.60C	a. Press the network test, it will ping the IP address 4 times. - Switching the setting of "Turn On When Power On", it will start to do network test. - During the network test, the button will be disabled. b. If all ping tests are successful, it will display the network is normal and the average time.  c. If there are 1~3 failures, it will display the network unstable and the number of failures. d. If all ping tests are failed, it will display the network error.

OPCUA Server

- Path
 - 5 key: F5 Maintain → F2 Set Kernel Server → Next → F8 OPCUA Server
 - 8 key: F5 Maintain → F2 Set Kernel Server → F8 OPCUA Server
 - 10+8 key: G5 Maintain → F2 Set Kernel Server → F8 OPCUA Server
- Explanation
 - Set the connection parameters of OPCUA server.
 - All setting changes take effect after reboot.

- Need to enable option 51.
- Require at least 20MB of memory space to start up OPCUA server.
- From 10.118.46C, 10.118.48N, 10.118.52H, 10.118.56B, 10.118.57(included).



- Description of parameters
 - Turn On When Power On : Yes/No
 - Default Setting is "No". If choose "Yes", it will start up when power on.
- Description of fenu button
 - F1: Save Setting
 - Save setting and return to "Set Kernel Server".
 - F2: Cancel
 - Cancel and return to "Set Kernel Server".

4.9.3 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 NO 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 Intrap. Times	852420	29.MPG Abs. Pos.	0	54.Intrap. Timeout Times	0
2.PLC Scan Times	852419	12.DA Voltage	2000	55. Fine Intrap. Timeout Times	0
3.I/O Scan Times	852419	44.Spindle Cmd.	1000	68.Axis Card Sync. Fail Times	0
4.Motion Intrap. 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
 - 5 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.

SYNTEC

Axes Data

G54	TEST N0 L1	Diagnostic	2020/1/22	13:47:59	DEFAULT
Controller Axes Data		X	Y	Z	
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.

SYNTEC

4.9.4 PLC Parameter

G54		TEST NO L1	Offset/ Setting					2020/1/22				13:55:21				DEFAULT			
Index	Item	Value																	
		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

Set MMIconfig

G54	N0 L1	Parameter	2021/2/24	15:09:25	Admin
Item		Value	1 / 10		
Program Font Size		20			
[Extension Filter] File Select Page		F;*.PIM;*.TAP;*.PRT;*.MPF;*.P			
[Extension Filter] File Manager Page		F;*.PIM;*.TAP;*.PRT;*.MPF;*.P			
[Program Editor] Auto Save		TRUE			
[Program Editor] Keyword Type		5			
[Fenubar] Font Size		11			
[Fenubar] Alignment		System Default			
[Fenubar] Button3D		System Default			
[Fenubar] Button 3D Level		System Default			
[Fenubar] No Function No.		System Default			
Value : 20 Hint InputRange: (7 ~ 100) Use default value if the input format is not correct. Use Lower/Upper bound value if the input value is out of range.					
		Ready	Auto	Alarm	
Modify		Set Default	Set Default All	My Favor F6	My Favor F7
				My Favor F8	

- 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

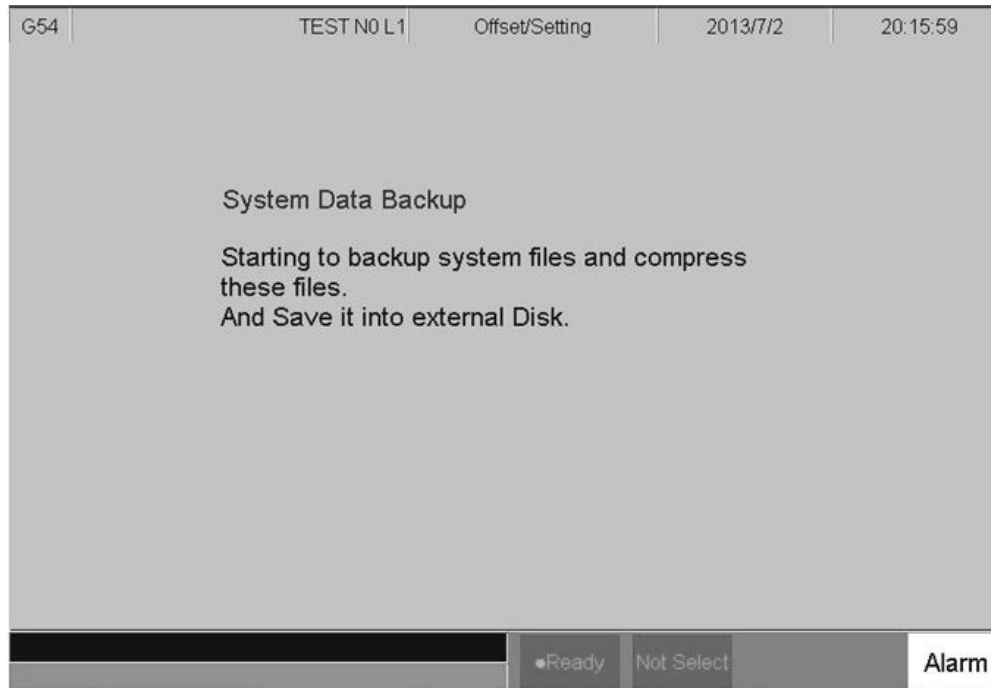
My Favor



- 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

SYNTEC

4.9.6 Backup System

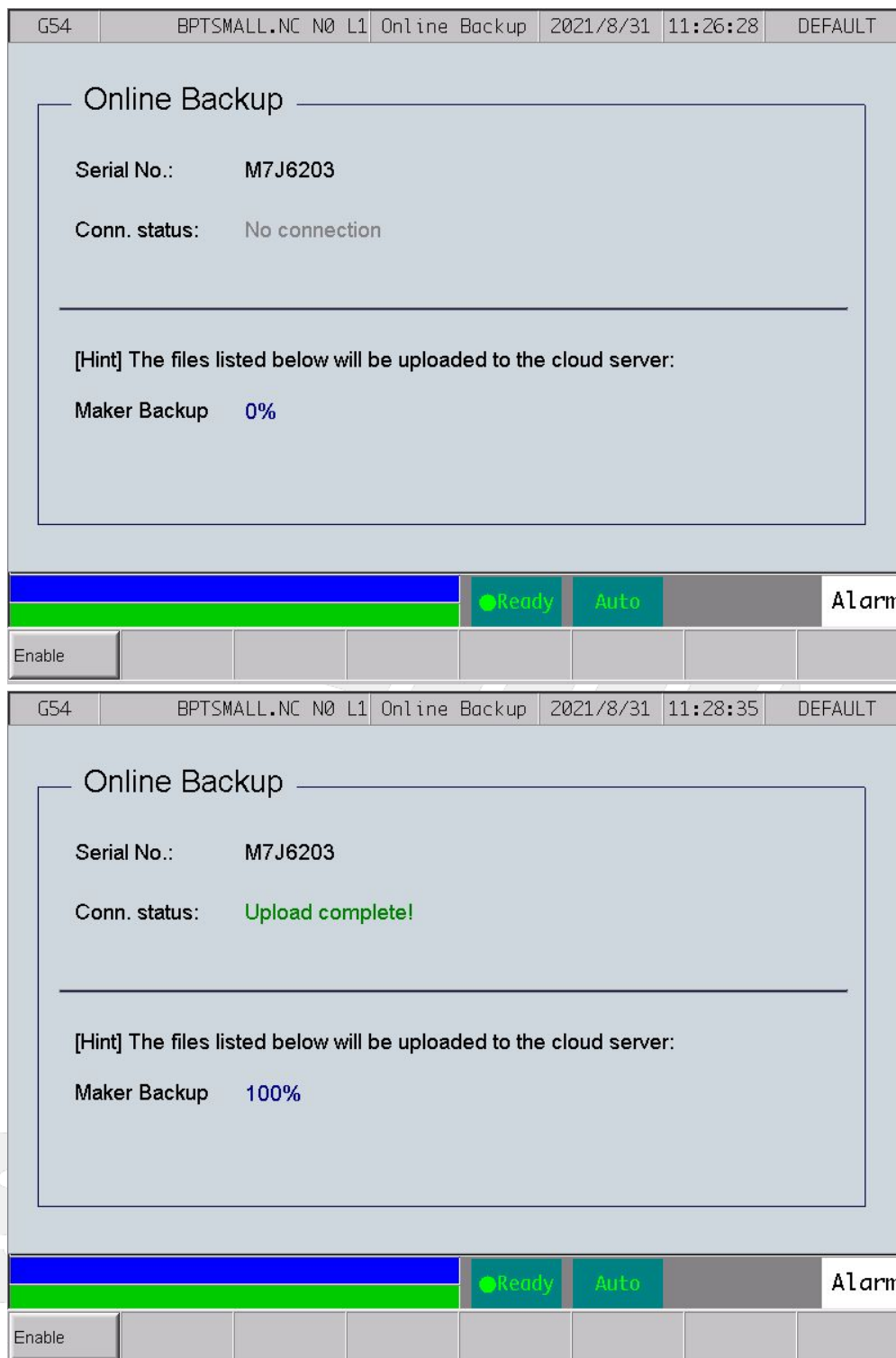


- Path
 - 5 key: F5 Maintain → Next → F1 Backup Sys.
 - 8 key: F5 Maintain → F6 Backup Sys.
 - 10+8 key: F5 Maintain → F6 Backup Sys.
- Explanation
 - Backup system data as a compressed file and saved into the assigned external device.

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
 - Uploading the system backup data to the cloud and using it for maintenance.

SYNTEC



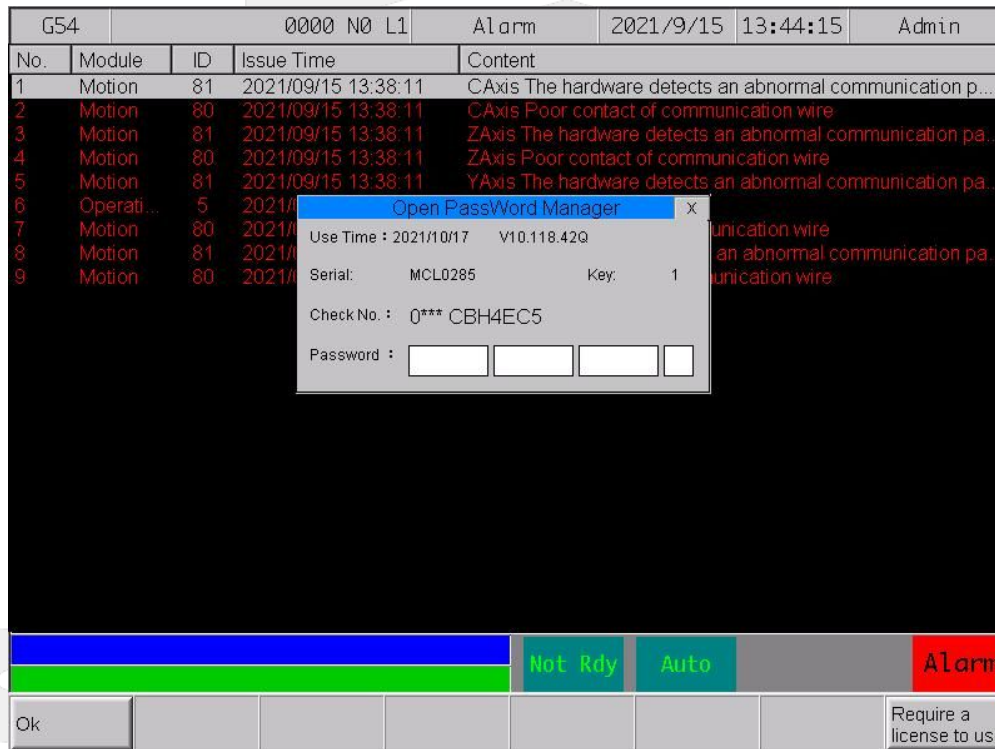
4.9.7 System Permissions

- 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

Remote Authorization

- Path
 - 5 keys: F5 Maintain → Next → F3 Use Time Setting → F5 Require a license to use
 - 8 key: F5 Maintain → F8 Use Time Setting → F8 Require a license to use
 - 10+8 key: G5 Maintain → F10 Use Time Setting → F10 Require a license to use
- Explanation
 - This fenu button will only be displayed when CNC has an authorization term and connects to cloud service successfully.
 - Press this fenu button to apply to the machinery for remote authorization. If remote authorization is successfully, the system use time will be extended.
 - From 10.118.42Q, 10.118.48B, 10.118.49.



4.9.8 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.

4.9.9 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.

4.9.10 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.

4.9.11 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.

4.9.12 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

4.9.13 About

- Path
 - 5 key: F5 Maintain → Next → Next → Next → F5 About
 - 8 key: F5 Maintain → Next → F8 About
 - 10+8 key: G8 System → F10 About
- Explanation
 - Provide system information of the controller.
 - Provide information of axial driver and motor.
 - Provide information of the spindle driver and motor.
 - System permissions-related settings.
 - Machinery information.

About

- Path
 - 5 key: F5 Maintain → Next → Next → Next → F5 About → F1 About
 - 8 key: F5 Maintain → Next → F8 About → F1 About
 - 10+8 key: G8 System → F10 About
- Explanation
 - Display system information



Normal Axis Information

Normal Axis

- Path
 - 5 key: F5 Maintain → Next → Next → Next → F5 About → F2 Normal Axis Information → F1 Normal Axis
 - 8 key: F5 Maintain → Next → F8 About → F2 Normal Axis Information → F1 Normal Axis
- Explanation
 - Display information of normal axis.

SYNTEC

G54	TEST N0 L1	NormalAxis	2019/5/6	15:37:00	Admin
[1/2] X					
Drive Model	N/A				
Drive Version	2.13.000				
Motor Model	N/A				
Enc. Model	N/A				
Enc. Version	0.00.000				
Enc. Resolution	10000				
2nd Enc. Model	N/A				
2nd Enc. Version	0.00.000				
2nd Enc. Resolution	262144				
Add-On Card Model	P06-MD4-ADP-AD-DA-1.4				
Add-On Card Version	1.00.000				
		Not Rdy	Auto	Alarm	
G54	TEST N0 L1	NormalAxis	2019/5/6	15:37:33	Admin
[2/2] X					
Drive SN	N/A				
Motor SN	N/A				
Enc. SN	N/A				
2nd Enc. SN	N/A				
Add-On Card SN	P5J49059				
		Not Rdy	Auto	Alarm	

- Displays invalid content as N/A.

SPLCA

- Path
 - 5 key: F5 Maintain → Next → Next → Next → F5 About → F2 Normal Axis Information → F2 SPLCA
 - 8 key: F5 Maintain → Next → F8 About → F2 Normal Axis Information → F2 SPLCA

- Explanation
 - Display information of SPLCA axis.
 - Displays invalid content as N/A.

G54	TEST N0 L1	SPLCA	2021/3/5	11:28:12	DEFAULT
[1/2] SPLC_1					
Drive Model	N/A				
Drive Version	2.13.000				
Motor Model	N/A				
Enc. Model	N/A				
Enc. Version	0.00.000				
Enc. Resolution	10000				
2nd Enc. Model	N/A				
2nd Enc. Version	0.00.000				
2nd Enc. Resolution	262144				
Add-On Card Model	P06-MD4-ADP-AD-1.4				
Add-On Card Version	1.00.000				
		Not Rdy	Home	Connect To 192.168.1.1	Alarm
NormalAxis	SPLCA	ROT	ATC		

ROT

- Path
 - 5 key: F5 Maintain → Next → Next → Next → F5 About → F2 Normal Axis Information → F3 ROT
 - 8 key: F5 Maintain → Next → F8 About → F2 Normal Axis Information → F3 ROT
- Explanation
 - Display information of ROT axis.
 - Displays invalid content as N/A.

G54	TEST N0 L1	ROT	2021/3/5	11:28:35	DEFAULT
[1/1] ROT_1					
Drive Model	N/A				
Drive Version	2.13.000				
Drive SN	N/A				
Motor Model	N/A				
Motor SN	N/A				
Enc. Version	0.00.000				
Enc. Resolution	10000				
Enc. SN	N/A				
		Not Rdy	Home	Connect To 192.168.1.1	Alarm
NormalAxis	SPLCA	ROT	ATC		

ATC (Support from 10.118.22)

- Path
 - 5 key: F5 Maintain → Next → Next → Next → F5 About → F2 Normal Axis Information → F4 ATC

- 8 key: F5 Maintain → Next → F8 About → F2 Normal Axis Information → F4 ATC
- Explanation
 - Display information of ATC axis.
 - Displays invalid content as N/A.

G54	TEST N0 L1	ATC	2021/3/5	11:28:54	DEFAULT
[1/1] ATC_1					
Drive Model	N/A				
Drive Version	2.10.004				
Drive SN	N/A				
Motor Model	N/A				
Motor SN	N/A				
Enc. Version	0.00.000				
Enc. Resolution	4194304				
Enc. Sensor Type	N/A				
Enc. SN	N/A				
		Not Rdy	Home	Connect To 192.168.1.1	Alarm
NormalAxis	SPLCA	ROT	ATC		

Spindle Information

- Path
 - 5 key: F5 Maintain → Next → Next → Next → F5 About → F3 Spindle Information
 - 8 key: F5 Maintain → Next → F8 About → F3 Spindle Information
- Explanation
 - Display information of spindle.

G54	TEST N0 L1	Spindle Info.	2019/5/6	15:37:16	Admin
[1/2] S1					
Drive Model	N/A				
Drive Version	2.13.000				
Motor Model	N/A				
Enc. Model	N/A				
Enc. Version	0.00.000				
Enc. Resolution	4194304				
2nd Enc. Model	N/A				
2nd Enc. Version	0.00.000				
2nd Enc. Resolution	10000				
Add-On Card Model	N/A				
Add-On Card Version	0.00.000				
		Not Rdy	Auto		Alarm

- Displays invalid content as N/A.

Fitting Information

Driver

- Path
 - 5 key: F5 Maintain → Next → Next → Next → F5 About → F4 Fitting Information → F1 Driver
 - 8 key: F5 Maintain → Next → F8 About → F4 Fitting Information → F1 Driver
- Explanation
 - Display information of driver fittings.

G54	TEST N0 L1	Fitting-Driver	2019/5/6	15:24:59	Admin
[1/1]					
Add-On Card Model	P06-MD4-ADP-AD-DA-1.4	P01-MD4-T8P16-1.3			
Add-On Card Version	1.00.000	1.00.000			
Add-On Card SN	P5J49059	P5H44783			
Host Drive Model	N/A	N/A			
Host Drive Station ID	1004	1005			
			Not Rdy	Auto	Alarm

- Displays invalid content as N/A.

SRI

- Path
 - 5 key: F5 Maintain → Next → Next → Next → F5 About → F4 Fitting Information → F2 SRI
 - 8 key: F5 Maintain → Next → F8 About → F4 Fitting Information → F2 SRI
- Explanation
 - Display information of SRI fittings.

- Displays invalid content as N/A.

Machine Builder Information

- Fenu and System Introduction – 79

G54	BPTSMALL.NC N0 L1	hine Builder In	2021/8/31	11:29:26	DEFAULT
-----	-------------------	-----------------	-----------	----------	---------

Builder Info.

Machine Model:

Serial No:

Built Date:

Wiring SN:

Builder Phone:

Builder Code: 5***

PLC Ver: 0.00

Machine Builder Info.	● Ready	Auto	Alarm
-----------------------	---------	------	-------

Machine Builder Info.	Product Reg.	Delivery Backup					
-----------------------	--------------	-----------------	--	--	--	--	--

- Editing authority management of machine builder information
 - All users are permitted to edit machine builder information after logged in, the cursor will appear when entering this page.
 - If the authority of editing needs to be limited, please switch to “Authority Management” and set “[About] Maker Info.” to “**Display Only**”. If the user without permission enters this screen, only the program information will be displayed.

SYNTEC

G54	N0 L1	Alarm	2021/3/5	16:59:03	Admin
SAMPLE1	Account: SAMPLE1	Password: 520	Page 4 / 5		
SAMPLE2	37	[Parameter] 5 Axis Param Setting	Open		
	38	[Parameter] Electric Param Setting	Open		
	39	[Parameter] HSHP Param Setting	Open		
	40	[Parameter] Driver Tuning	Open		
	41	[System Admin]	Open		
	42	[PLC Ladder] Edit Mode	Open		
	43	[Parameter] Adjust Tunning	Open		
	44	[Parameter] Serial PLC Axis Param.	Open		
	45	[Parameter] ROT Param.	Open		
	46	[About] Maker Info.	Open		
	47	[Parameter] I/O Mapping Table	Open		
	48	[Parameter] ATC Param.	Open		
		Ready	Auto	Alarm	
Close		Display Only	Open		

Product Registration(Warranty)

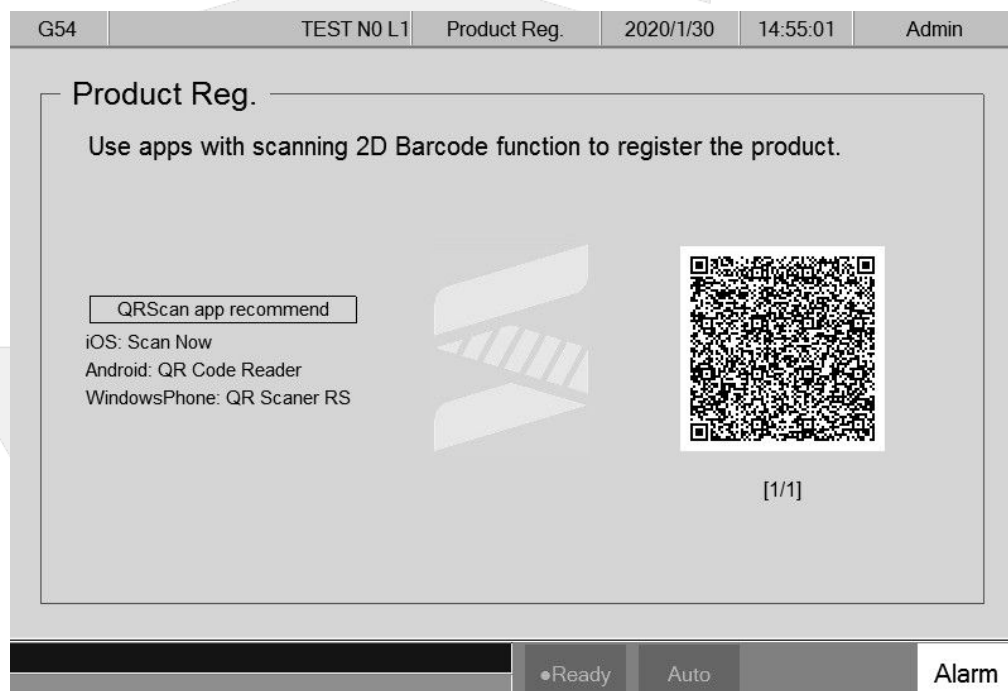
- Path
 - 5 key: F5 Maintain → Next → Next → Next → F5 About → F5 Machine Builder Information → F2 Product Registration
 - 8 key: F5 Maintain → Next → F8 About → F5 Machine Builder Information → F2 Product Registration
 - 110+8 key: G8 System → F10 About → F5 Machine Builder Information → F2 Product Registration
- Explanation
 - To provide 2D bar code and register the products online after scanning.
- Operation Method

SYNTEC

- Enter the page and wait for the controller to create 2D bar code.



- Use APP to scan 2D bar code and register the products.
 - If there are multiple 2D barcodes, the barcode can be switched by function keys "Next" and "Previous"; and all barcodes must be scanned and registered before leaving the page by the "Finish" button.



Delivery Backup

- Path
 - 5 key: F5 Maintain → Next → Next → Next → F5 About → F5 Machine Builder Information → F3 Delivery Backup
 - 8 key: F5 Maintain → Next → F8 About → F5 Machine Builder Information → F3 Delivery Backup
 - 110+8 key: G8 System → F10 About → F5 Machine Builder Information → F3 Delivery Backup
- Explanation
 - Providing cloud backup service when delivering the machine.
 - If the backup is not uploaded, the time of backup would display N/A.
 - From 10.118.32N, 10.118.40Q, 10.118.41Q, 10.118.48B, 10.118.49(included)



SYNTEC

G54	BPTSMALL.NC N0 L1	Delivery Backup	2021/8/31	11:29:45	DEFAULT
-----	-------------------	-----------------	-----------	----------	---------

Delivery Backup

Serial No.: M7J6203

Backup Time: N/A

Conn. status: No connection

[Hint1] The files listed below will be uploaded to the cloud server.
[Hint2] The upper limit is 10 records.

Maker Backup 0%

Ready
Auto
Alarm

Send

G54	BPTSMALL.NC N0 L1	Delivery Backup	2021/8/31	11:31:50	DEFAULT
-----	-------------------	-----------------	-----------	----------	---------

Delivery Backup

Serial No.: M7J6203

Backup Time: 2021-08-31 11:31:38

Conn. status: Upload complete!

[Hint1] The files listed below will be uploaded to the cloud server.
[Hint2] The upper limit is 10 records.

Maker Backup 100%

Ready
Auto
Alarm

Send

Online Service

- Path

- 5 key: F5 Maintain → Next → Next → Next → F5 About → F6 Online Service
- 8 key: F5 Maintain → Next → F8 About → F6 Online Service
- 10+8 key → G8 System → F10 About → F6 Online Service
- Explanation
 - Provide a 2D bar code for scanning, which can upload the controller-related information to the website of customer service.
 - The valid version starts from 10.116.36O, 10.116.45, 10.117.45 (included)
 - The information contained in the bar code
 - For Wechat user
 - URL of customer service of SYNTEC
 - Controller information
 - CNC Information: bar code version, controller information, CPU board type, image file version, software version
 - Machine Builder Information: What you can see in "Machine Builder Information" Screen
 - Axis Information:
 - Yaskawa (YAS): drive model, motor model
 - SYNTEC (SYN); driver serial number, motor serial number, driver firmware version, the first and the second encoder serial number and firmware version
 - History Alarm Information: Up to 30 History Alarms
 - For common user
 - URL of customer service of Syntec
 - Controller information
 - CNC Information: bar code version, controller information, CPU board type, image file version, software version
 - Machine Builder Information: What you can see in "Machine Builder Information" Screen
- Operation Method
 - For Wechat user
 - Select Wechat User

SYNTEC

G54	TEST N0 L1	Coord.	2020/1/30	15:20:51	Admin
-----	------------	--------	-----------	----------	-------

Online Service

SYNTEC TECHNOLOGY CO.,LTD.

Please select user type to call customer service

☒ Wechat User

☐ Common User

Ready

Auto

Alarm

View changes

- Add SYNTEC Wechat official account (can be skipped if you already have our official account)

G54	TEST N0 L1	Coord.	2020/1/30	15:21:29	Admin
-----	------------	--------	-----------	----------	-------

Online Service

SYNTEC TECHNOLOGY CO.,LTD.

Add Syntec WeChat (Official) Accounts



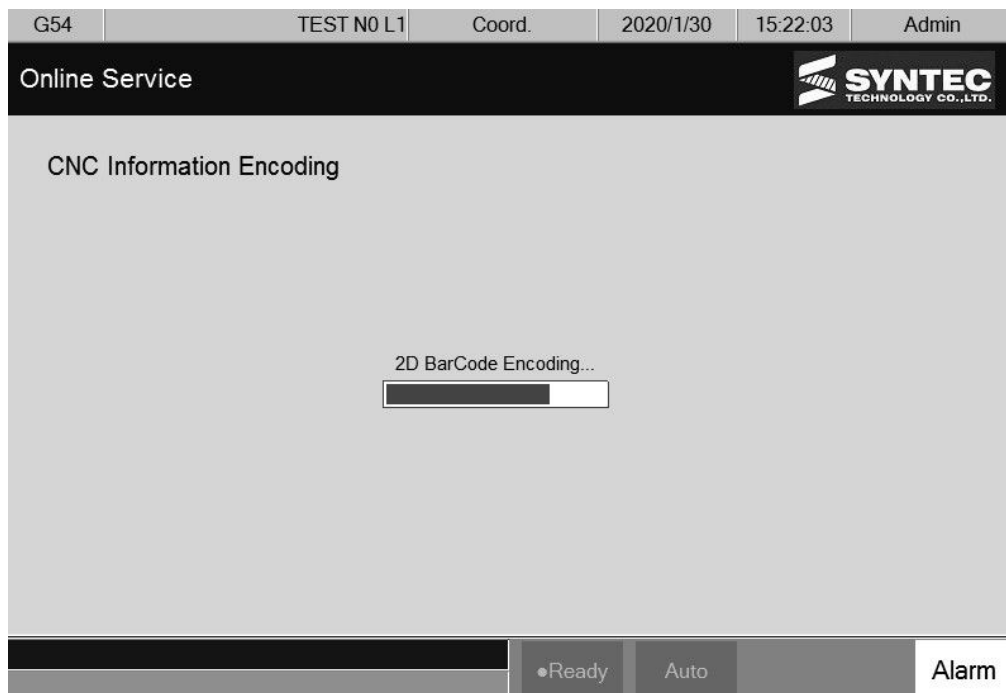
If you have already added SYNTEC Whechat contact, please skip this step.

Ready

Auto

Alarm

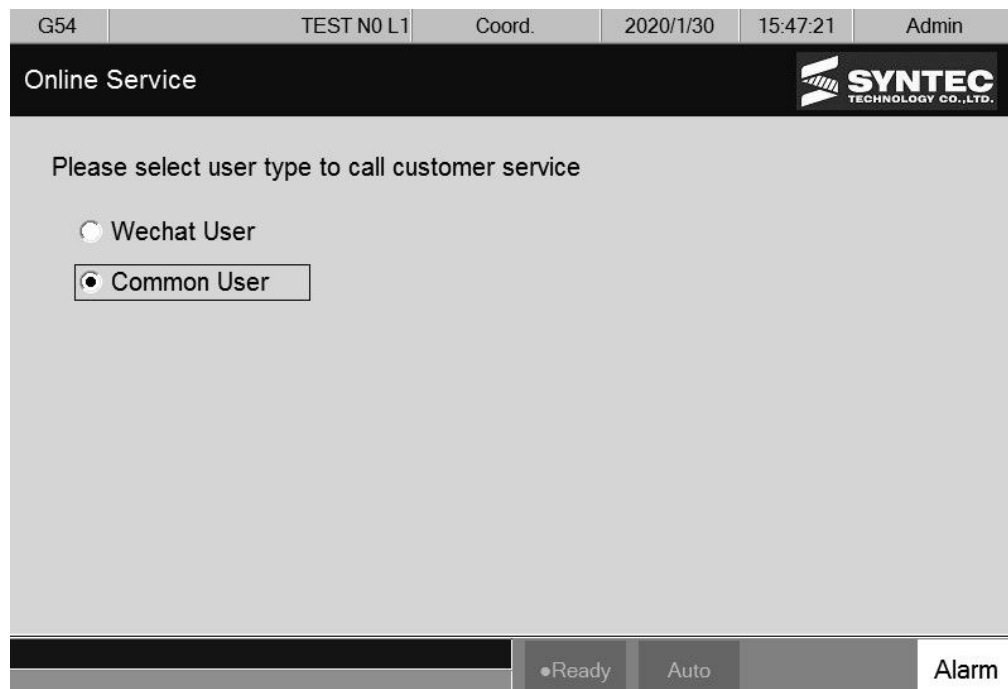
- Wait for 2D bar code



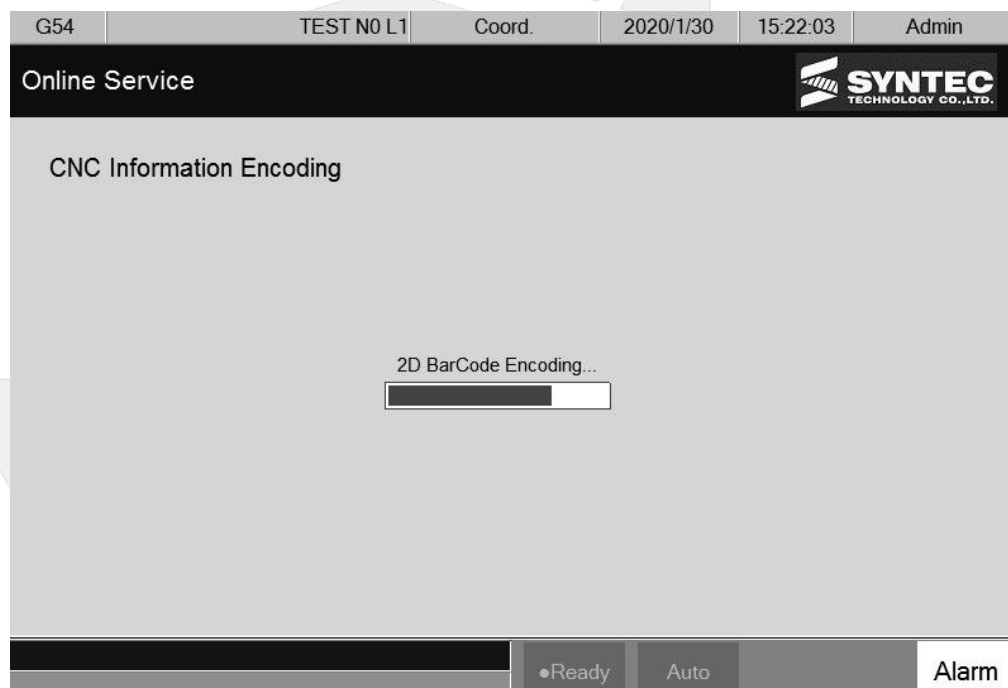
- Scan 2D bar code
 1. Please select "客户专区" → "客户服务申请" via Wechat APP.
 2. Press OK and the scan box of Wechat will appear, scan 2D bar code and the controller information will be uploaded to the customer service request page.
 3. Fill in the required information
 4. Press "Save" to send the information and press "Finish" to leave the screen on the controller.



- For Common user
 - Select Common User

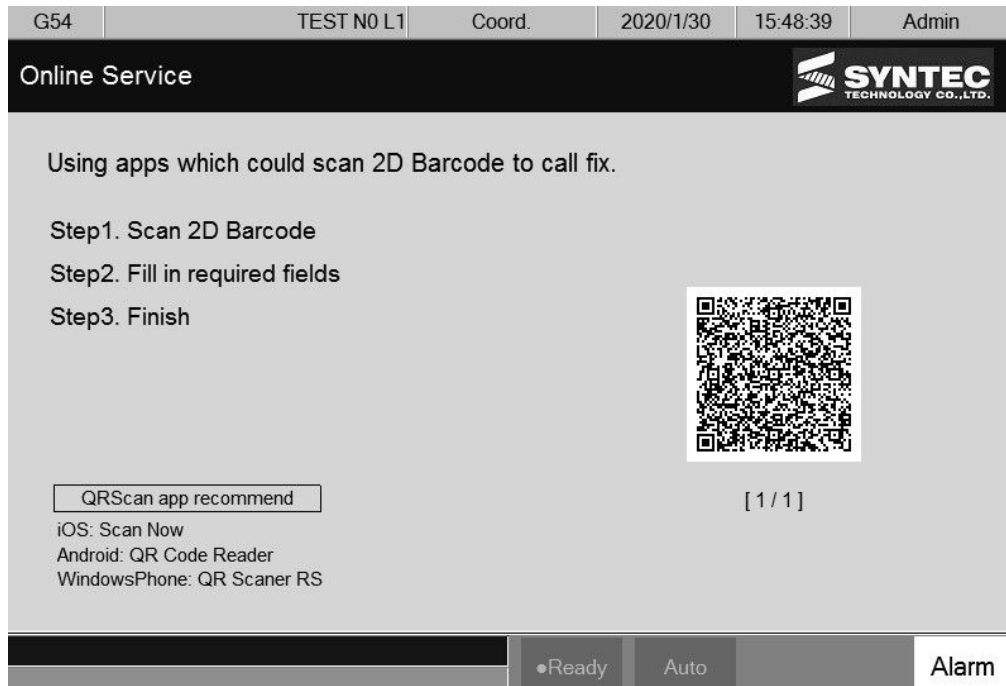


- Wait for 2D bar code



- Scan 2D bar code

1. Prepare an app with 2D bar code scanning function. (The apps showed on the controller screen are recommended.)
2. Scan 2D bar code and the controller information will be uploaded to the customer service request page.
3. Fill in the required information.
4. Press “Save” to send the information and press “Finish” to leave the screen on the controller.



- Note
 - When the controller information is too much to display in one bar code, two or more bar codes will be created. When scanning, the system will also remind the users to scan the remaining bar codes.
 - If two or more bar codes are created, the code number will be shown on the screen like [1/2] and [2/2], also “Next” and “Previous” buttons can be used to switch.
 - The hotkeys on the operation panel will be disabled when applying the online service action.

System Permissions

Serial No. Setting

- Path
 - 5 key: F5 Maintain → Next → Next → Next → F5 About → F4 System Permissions → F1 Ser. No. Setting
 - 8 key: F5 Maintain → Next → F8 About → F7 System Permissions → F1 Ser. No. Setting
 - 10+8 key: G8 System → F10 About → F4 System Permissions → F1 Ser. No, Setting
- Explanation
 - Can be used to set the system serial number
 - Please ask the original manufacturer for the setup password

Option Setting

- Path
 - 5 key: F5 Maintain → Next → Next → Next → F5 About → F4 System Permissions → F2 Option Setting
 - 8 key: F5 Maintain → Next → F8 About → F7 System Permissions → F2 Option Setting
 - 10+8 key: G8 System → F10 About → F4 System Permissions → F2 Option Setting
 - Explanation
 - Can be used to set the system function options
 - The function will be enabled after entering the password, there are two ways to access the password
 - i. Please ask the original manufacturer for the setup password
 - ii. Press "Online Decrypt" and scan the bar code with phones or tablets to access the password online (supported after 10.118.22N)
- "Online Decrypt" Button location:
1. 5 key: F5
 2. 8 key: F8
 3. 10+8 key: F10

Option Information Restore

- Path
 - 5 key: F5 Maintain → Next → Next → Next → F5 About → F7 System Permissions → F3 Op. Info Restore
 - 8 key: F5 Maintain → Next → F8 About → F7 System Permissions → F3 Op. Info Restore
 - 10+8 key: G8 System → F10 About → F4 System Permissions → F3 Op. Info Restore
- Explanation
 - Can be used to restore the system authorities
 - Please ask the original manufacturer for the setup password

The 2Dbarcode of Operation Manual

- Path
 - 5 key: F5 Maintain → Next → Next → Next → F5 About → F9 Help
 - 8 key: F5 Maintain → Next → F8 About → F9 Help
 - 10+8 key: G8 System → F10 About → F12 Help
- Explanation
 - Display the 2Dbarcode of the operation manual
 - From 10.118.28Z, 10.118.29Z, 10.118.40F, 10.118.44 (included)

SYNTEC



SynFactory Device Binding

- Path
 - 5 key: F5 Maintain → Next → Next → Next → F5 About → Next → F3 SynFactory Binding
 - 8 key: F5 Maintain → Next → F8 About → F8 SynFactory Binding
 - 10+8 key: G8 System → F10 About → F8 SynFactory Binding
- Explanation
 - Display the 2Dbarcode of the operation manual
 - After finishing the device binding, please reboot CNC to activate cloud service

SYNTEC

- From 10.118.320, 10.118.40Q, 10.118.41Q, 10.118.48B, 10.118.49(included)



4.9.14 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 → F1 Pending Alarm
- Explanation
 - Display current system alarm.
 - Up to 20 alarms of CNC and driver can be displayed.

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

Alarm Analysis

Available versions: 10.118.42S、10.118.48D、10.118.50 and above.

CNC Monitor v10.118.48						
G54		123 N0 L1		Alarm	2021/9/13	17:54:10
					17:54:10	DEFAULT
No.	Module	ID	Cnt.	Issue Time	Content	
1	Motion	81	32	21/08/17 19:13:47	CAxis Poor contact of communication wire	
2	Motion	80	32	21/08/17 19:13:47	ZAxis Poor contact of communication wire	
3	Operation	5	8	21/08/17 19:13:47	YAxis Poor contact of communication wire	
4	MLC	87	3	21/08/17 19:13:47	XAxis Poor contact of communication wire	
5	Operation	21	1	21/08/17 18:20:36	CAxis Poor contact of communication wire	
				21/08/17 18:20:36	ZAxis Poor contact of communication wire	
				21/08/17 18:20:36	YAxis Poor contact of communication wire	
				21/08/17 18:20:36	XAxis Poor contact of communication wire	
				21/08/13 18:33:57	CAxis Poor contact of communication wire	
				21/08/13 18:33:57	ZAxis Poor contact of communication wire	
				21/08/13 18:33:57	YAxis Poor contact of communication wire	
				21/08/13 18:33:57	XAxis Poor contact of communication wire	
				21/08/13 18:32:55	CAxis Poor contact of communication wire	
				21/08/13 18:32:55	ZAxis Poor contact of communication wire	
				21/08/13 18:32:55	YAxis Poor contact of communication wire	
				21/08/13 18:32:55	XAxis Poor contact of communication wire	
				21/08/13 18:22:58	CAxis Poor contact of communication wire	
				21/08/13 18:22:58	ZAxis Poor contact of communication wire	
				21/08/13 18:22:58	YAxis Poor contact of communication wire	
				21/08/13 18:22:58	XAxis Poor contact of communication wire	
				21/08/13 18:21:20	CAxis Poor contact of communication wire	
				21/08/13 18:21:20	ZAxis Poor contact of communication wire	
				21/08/13 18:21:19	YAxis Poor contact of communication wire	
				21/08/13 18:21:19	XAxis Poor contact of communication wire	
				21/08/13 18:19:45	CAxis Poor contact of communication wire	

Ready
Auto
Alarm

Pending Alarm
History Alarm
Save Alarm
Analyze All
Analyze CNC

- Descriptions

Alarm Analysis page has two parts:

- a. Left- Alarm category list
 - Display **No.**, **Module**, **ID**, **Cnt.** informations.
 - Display categories of alarms within 30-Days.
 - Cnt. means how many times the alarm is triggered within 30-Days.
 - Display categories in descending order of Cnt..
- b. Right-Alarm detail list
 - Display history of the alarm selected in the left list.
 - History records is not limited in 30-Days.
 - History records before 30-Days will be displayed in grey text color.
 - Display **IssueTime**, **Content** informations
- c. Operations
 - Use up/down key to move focus.
 - Use PageUp/Down key to switch pages.
 - Change [Alarm category list] focus to refresh right-side detail records.
 - Use F9/F12 to view help of the alarm.
 - Use right/left key to switch focus between [Alarm category list] and [Alarm detail list]

Analyze All

- Path
 - 5 key, 8 key: F5 Maintain → F1 Alarm → F4 Analyze All
 - 10+8 key: G5 Maintain → F1 Alarm → F4 Analyze All
- Explanation
 - Analyze alarms of all types.

Analyze CNC

- Path
 - 5 key, 8 key: F5 Maintain → F1 Alarm → F5 Analyze CNC
 - 10+8 key: G5 Maintain → F1 Alarm → F5 Analyze CNC
- Explanation
 - Analyze alarms of CNC type.
 - *The **CNC alarms** means all alarms excluding the **MLC alarms**.

SYNTEC

4.9.15 Network Setting

G54	TEST NO 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

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.

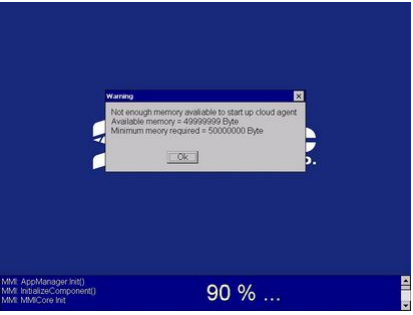
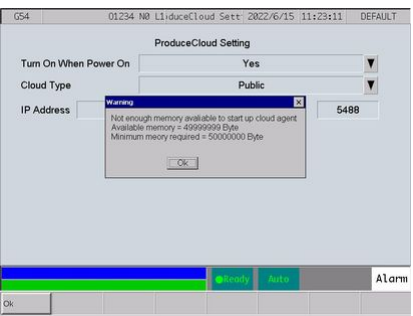
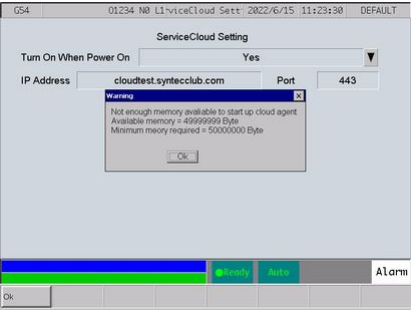
SYNTEC

CloudAgent Setting

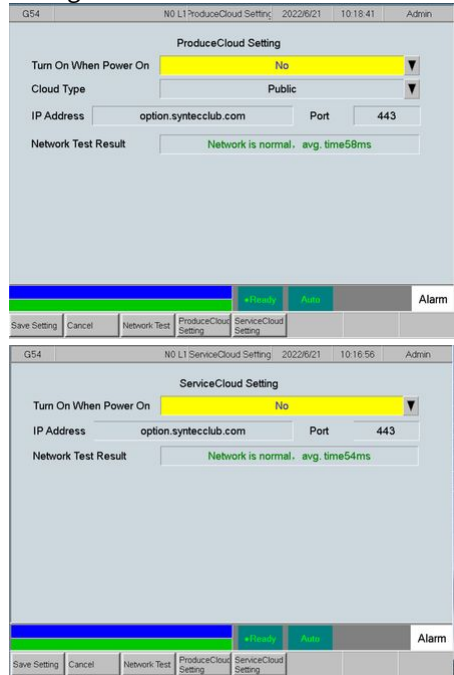
The image displays two side-by-side screenshots of the CloudAgent Setting interface. The left screenshot is titled 'ProduceCloud Setting' and the right is titled 'ServiceCloud Setting'. Both screens show a status bar at the top with 'G54', 'N0 L1', the current setting, date/time, and user 'Admin'. The main area contains settings for 'Turn On When Power On' (set to 'No'), 'Cloud Type' (set to 'Public'), 'IP Address' (set to 'option.syntecclub.com'), and 'Port' (set to '443'). A 'Network Test Result' field is present but empty. The bottom status bar includes buttons for 'Save Setting', 'Cancel', 'Network Test', 'ProduceCloud Setting', 'ServiceCloud Setting', and 'Alarm'.

- Path
 - 5 key: F5 Maintain → F2 Set Kernel Server → F3 CloudAgent Setting
 - 8 key: F5 Maintain → F2 Set Kernel Server → F3 CloudAgent Setting
 - 10+8 key: G5 Maintain → F2 Set Kernel Server → F3 CloudAgent Setting
- Explanation
 - Set up the parameters of ProduceCloud and ServiceCloud.
 - Require enough memory to start up CloudAgent. (REF : [Appendix1](#))
 - Network Test function (REF : [Appendix2](#))
 - All setting changes take effect after rebooting.
 - From 10.118.44H, 10.118.46B, 10.118.48M, 10.118.52G, 10.118.56A, 10.118.57 (included).
- Description of parameters
 - Turn On When Power On : Yes/No
 - Default Setting is "No". If choose "Yes", it will start up when power on.
 - Cloud Type : Public Cloud/Private Cloud
 - Only Support "ProduceCloud Setting".
 - Choose public cloud or private cloud according to server.
 - IP Address:
 - Input IP address of server. The default setting is option.syntecclub.com.
 - Port:
 - The default setting is 443.
- Description of fenu button
 - F1: Save Setting
 - Save setting and return to "Set Kernel Server". The changes of produce cloud and service cloud need to be saved separately.
 - F2: Cancel
 - Cancel and return to "Set Kernel Server".
 - F3 : Network Test
 - Doing a network test according to the IP address, and display the test result on the screen
 - F4: ProduceCloud Setting
 - Switch screen to "ProduceCloud Setting".
 - F5: ServiceCloud Setting
 - Switch screen to "ServiceCloud Setting".

• Appendix1 :

Version	Minimum available memory	Supplemental instruction
From 10.118.44H, 10.118.46B, 10.118.48M, 10.118.52G, 10.118.56A, 10.118.57	30MB	Require at least 30MB of memory space to start up ProduceCloud or ServiceCloud.
From 10.118.44J, 10.118.46J, 10.118.48U, 10.118.52O, 10.118.56I, 10.118.60C	50MB	1.Require at least 50MB of memory space to start up ProduceCloud or ServiceCloud.  2. Setting "Turn On When Power On" Choosing "Yes" will do memory check, if less than 50MB show the warning message and set the value to "No".  

• Appendix2 :

Version	Supplemental instruction
From 10.118.44J, 10.118.46J, 10.118.48U, 10.118.52O, 10.118.56I, 10.118.60C	a. Press the network test, it will ping the IP address 4 times. - Switching the setting of "Turn On When Power On", it will start to do network test. - During the network test, the button will be disabled. b. If all ping tests are successful, it will display the network is normal and the average time.  c. If there are 1~3 failures, it will display the network unstable and the number of failures. d. If all ping tests are failed, it will display the network error.

OPCUA Server

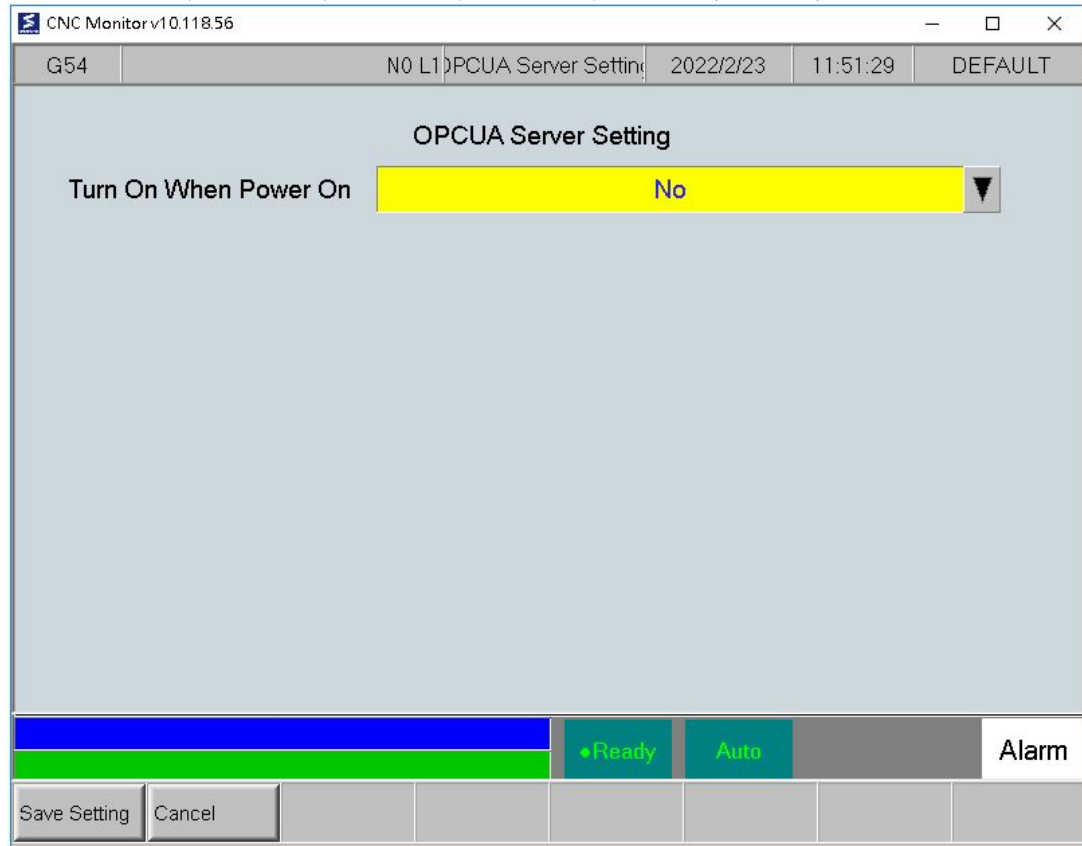
• Path

- 5 key: F5 Maintain → F2 Set Kernel Server → Next → F8 OPCUA Server
- 8 key: F5 Maintain → F2 Set Kernel Server → F8 OPCUA Server
- 10+8 key: G5 Maintain → F2 Set Kernel Server → F8 OPCUA Server

• Explanation

- Set the connection parameters of OPCUA server.

- All setting changes take effect after reboot.
- Need to enable option 51.
- Require at least 20MB of memory space to start up OPCUA server.
- From 10.118.46C, 10.118.48N, 10.118.52H, 10.118.56B, 10.118.57(included).



- Description of parameters
 - Turn On When Power On : Yes/No
 - Default Setting is "No". If choose "Yes", it will start up when power on.
- Description of fenu button
 - F1: Save Setting
 - Save setting and return to "Set Kernel Server".
 - F2: Cancel
 - Cancel and return to "Set Kernel Server".

4.9.16 Fast Diagnostic

- 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 NO 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 Intrap. Times	852420	29.MPG Abs. Pos.	0	54.Intrap. Timeout Times	0
2.PLC Scan Times	852419	12.DA Voltage	2000	55. Fine Intrap. Timeout Times	0
3.I/O Scan Times	852419	44.Spindle Cmd.	1000	68.Axis Card Sync. Fail Times	0
4.Motion Intrap. 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
 - 5 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.

SYNTEC

Axes Data

G54	TEST N0 L1	Diagnostic	2020/1/22	13:47:59	DEFAULT
Controller Axes Data		X	Y	Z	
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.

SYNTEC

4.9.17 PLC Param Setting

G54	TEST NO 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
 - Move the cursor with arrow keys **【↑】** **【↓】** **【←】** **【→】**
 - Use **【PageUp】** **【PageDown】** to switch pages.
 - Users can only input 0 or 1.
 - Users can add comments to each Bit.
 - Corresponding file name for the comments is ParamExt_RBit_(L).xml
 - (L) = COM/CHT/CHS/language

Set MMIconfig

G54	N0 L1	Parameter	2021/2/24	15:09:25	Admin
Item		Value	1 / 10		
Program Font Size		20			
[Extension Filter] File Select Page		F;*.PIM;*.TAP;*.PRT;*.MPF;*.P			
[Extension Filter] File Manager Page		F;*.PIM;*.TAP;*.PRT;*.MPF;*.P			
[Program Editor] Auto Save		TRUE			
[Program Editor] Keyword Type		5			
[Fenubar] Font Size		11			
[Fenubar] Alignment		System Default			
[Fenubar] Button3D		System Default			
[Fenubar] Button 3D Level		System Default			
[Fenubar] No Function No.		System Default			
Value : 20 Hint InputRange: (7 ~ 100) Use default value if the input format is not correct. Use Lower/Upper bound value if the input value is out of range.					
		●Ready	Auto	Alarm	
Modify		Set Default	Set Default All	My Favor F6	My Favor F7
				My Favor F8	

- 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

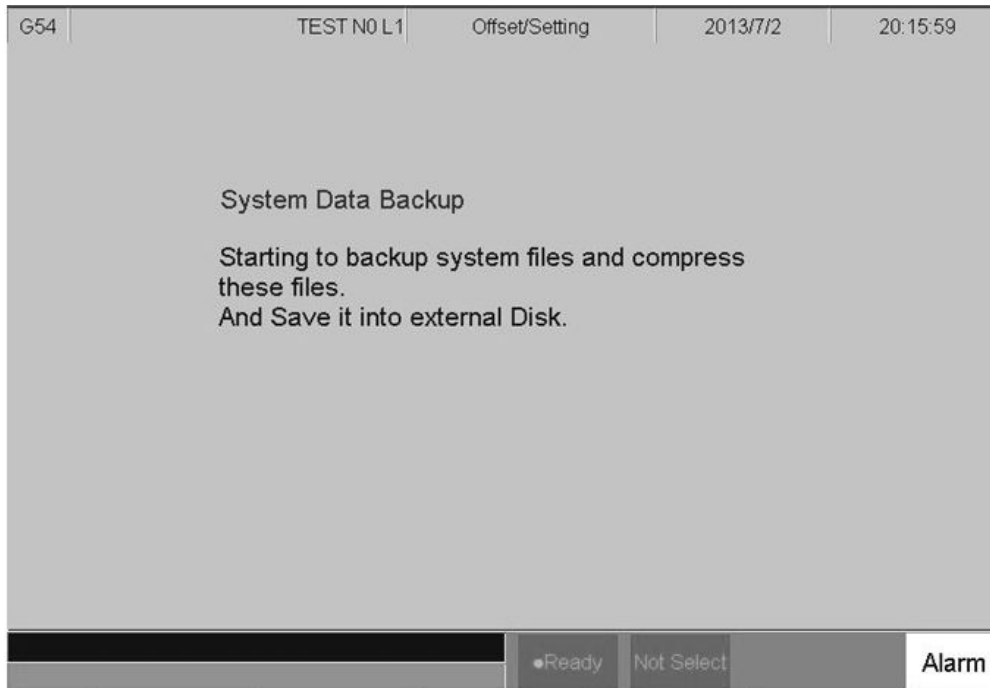
My Favor



- 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

SYNTEC

4.9.19 Backup System

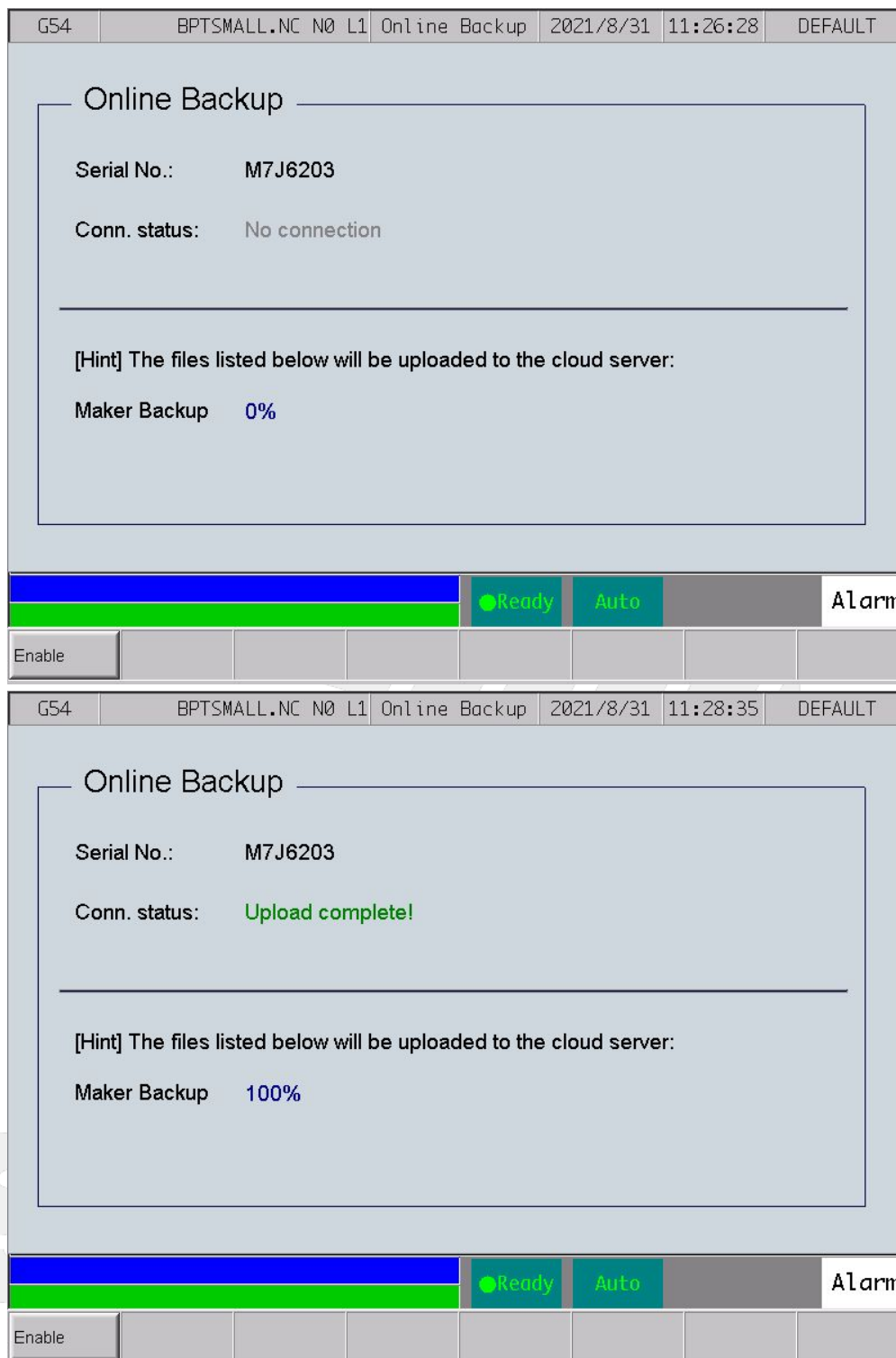


- Path
 - 5 key: F5 Maintain → Next → F1 Backup Sys.
 - 8 key: F5 Maintain → F6 Backup Sys.
 - 10+8 key: F5 Maintain → F6 Backup Sys.
- Explanation
 - Backup system data as a compressed file and saved into the assigned external device.

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
 - Uploading the system backup data to the cloud and using it for maintenance.

SYNTEC



4.9.20 About

- Command:

G54	TEST N0 L1										Diagnostic					2014/12/25					19:23:59				
I Bits																									
	0	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19					
0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0					
20	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0					
40	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0					
60	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0					
80	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0					
100	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0					
120	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0					
140	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0					
160	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0					
180	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0					
200	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0					
220	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0					
240	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0					
										●Ready					Auto										Alarm

Key Selections:

F1: PLC Status



SYNTEC

- Function: Display the component states of PLC and state of staircase chart.

G54	TEST N0 L1										Diagnostic				2014/12/25				19:26:22			
I Bits																						
	0	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19		
0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0		
20	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0		
40	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0		
60	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0		
80	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0		
100	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0		
120	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0		
140	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0		
160	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0		
180	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0		
200	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0		
220	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0		
240	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0		
										●Ready				Auto				Alarm				

F2: Display System Data

SYNTEC

- Function: Display current system data (refer to application manual for each corresponding meaning of the serial number.)

G54	TEST NO L1				Diagnostic		2014/12/25		19:34:39	
System Data										
No	Data	No	Data	No	Data	No	Data	No	Data	
0	98517	16	3333	32	0	48	0	64	0	
1	6074671	17	3333	33	0	49	0	65	0	
2	2024890	18	2222	34	0	50	0	66	0	
3	10124450	19	2222	35	0	51	0	67	0	
4	1966	20	0	36	0	52	0	68	0	
5	5898	21	0	37	0	53	0	69	0	
6	78	22	430	38	32	54	0	70	0	
7	13160448	23	119	39	40	55	0	71	0	
8	0	24	0	40	0	56	0	72	0	
9	0	25	0	41	0	57	0	73	0	
10	0	26	0	42	0	58	0	74	0	
11	0	27	0	43	0	59	0	75	0	
12	0	28	0	44	0	60	0	76	0	
13	0	29	0	45	0	61	0	77	12357632	
14	0	30	0	46	0	62	0	78	12357632	
15	0	31	0	47	0	63	0	79	10.116.0X	
					●Ready	Auto				
									Alarm	

F3: Display Global

G54	TEST NO L1		Diagnostic		2014/12/25		19:35:41	
Global Data								
No	Data	No	Data	No	Data	No	Data	
1		17		33				
2		18		34				
3		19		35				
4		20		36				
5		21		37				
6		22		38				
7		23		39				
8		24		40				
9		25		41				
10		26		42				
11		27		43				
12		28		44				
13		29		45				
14		30		46				
15		31		47				
16		32		48				
				●Ready	Auto	Alarm		

4.12 Sys. Admin (MainFenu: F8)

F1: Install Software

F2: Backup System

F3: Restore System

F4: Export/Import

F5: Install Image

F6: System Info.

F7: Update Boot Picture



SYNTEC

5 How to Operate 60 Series Controller

5.1 System Status

5.1.1 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.

5.1.2 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.

5.1.3 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.

5.1.4 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.

5.1.5 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.

5.2 Machine Preparation

5.2.1 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

5.2.2 Machining Process

AUTO

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

Single Block

- Function:
 - Execute each single block in program
- Condition:
 - System is on "Ready" status
 - Single block mode is selected

- Operation:
 - Press 「CYCLE START」 button
 - System will execute process the current single block in program
 - System status will be switched from "Ready" to "busy" and backs to "Ready" when machining is finished

Home

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

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

5.3 Workpiece Preparation

5.3.1 Workpiece Coord set

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

- Conditions
 - N/A
- Operation Procedure

SYNTEC

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

- b. Move cursor to the required coordinate with $\uparrow \downarrow \leftarrow \rightarrow$ and page $\uparrow$, page $\downarrow$
- 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.

5.3.2 Tool Prepare

Tool Set

- Purpose
 - We can set the length and the diameter geometry and wear
 - Operation condition
 - Can be used at Manual or Auto mode
 - Wear value increment upper limit is 1.0, warning will appear if exceed 1.0.
- Operation method
 - Use direction key $\uparrow$ $\downarrow$ $\leftarrow$ $\rightarrow$ move the cursor.
 - Use $\uparrow$ $\downarrow$ switch the page.
 - Key in method: Absolute, Increment, Measure method.
 - Absolute: for radius and length compensate.
 - Increment: for radius wear and length wear.
 - Radius compensate + radius wear = real G41/G42 compensate.

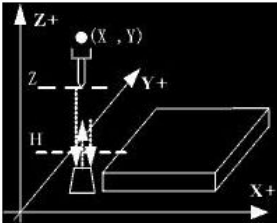
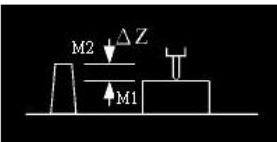
- Length compensate +length wear = real G43/G44 compensate.

Manual Measurement of Tool Length

- Purpose
 - Describe how to use manual measurement to set tool length and work piece coordinate in order to set the machining tool.
- Function
 - We usually set the tool length and offset in the External Shift if we only use one tool in this machine.
 - If we have more than one tool, we need to set every tool length we can use relative length or absolute length.
- Operation condition
 - Manual mode.
- Manual Operation method
 - Relative method.
 - Use manual function moving the reference tool to the Z-coordinate code 0 position of the workpiece (Ex. surface), set the value at G54.
 - Clean all the relative coordinate value.
 - Install the measured tool, move the tool to the same position as before (reference surface).
 - The tool length is the different between measured tool and reference tool.
 - Key in this number to the corresponding tool No.
 - If tool break during machining and replace a new tool, we need to change to a new tool. If the Z-cord 0 position already disappear, we can use the standard tool to touch anywhere of the machine and then use new tool to touch the same position. The relative cord. is tool length.
 - Absolute method
 - Use manual function moving the measuring tool to a reference point (Ex. Work piece table).
 - Measure every tool repeatedly.
 - Move the machining tool to the reference position, clean all the relative coordinate.
 - Move the tool to the Z-coord code zero position, set it to G54.
 - If tool break during machining so we need change a new tool, we must touch the reference position first, and then the Z coordinate value is tool length.

Tool Tip Measure

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

G54	N0 L1	Offset/Setting	2013/8/22	18:12:45
Auto Tool Function		WorkPiece No. P	0	Machine
AutoTool 1		Feedrate F	0.000	X 0.000
1:One Tool One Workpiece,2:On		Use Reference	0	Y 0.000
3:Many Tools Many Workpie		Ref Coord. X	0.000	Z 0.000
		Ref Coord. Y	0.000	A 4.158
		Start Coord. Z	0.000	
		Min. Z Mach. H	0.000	Relative
		Select if use Ref Point		X 0.000
		1:Set All measure parameter		Y 0.000
		2:If not use Ref, Take tool tip to upper of measurement		Z 0.000
		3:Press F1, Measure Start		A 4.158
Delta Z Set		Delta Z Set	0.000	Aux. Coord
		Do tool tip measure before do Delt		X 0.000
		1:Take tool tip to top of good		Y 0.000
		2:Press F3, Delta Z Set		Z 0.000
(1~3)		●Ready	Auto	Alarm

Auto Tool Function

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

SYNTEC

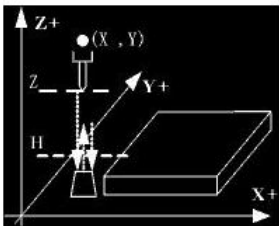
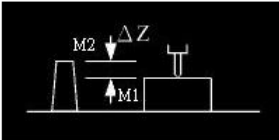
G54	N0 L1	Offset/Setting	2013/8/22	18:12:45	
Auto Tool Function		WorkPiece No. P	0	Machine	
AutoTool 1		Feedrate F	0.000	X	0.000
1:One Tool One Workpiece,2:On		Use Reference	0	Y	0.000
3:Many Tools Many Workpie		Ref Coord. X	0.000	Z	0.000
		Ref Coord. Y	0.000	A	4.158
		Start Coord. Z	0.000		
		Min. Z Mach. H	0.000	Relative	
		Select if use Ref Point		X	0.000
		1:Set All measure parameter		Y	0.000
		2:If not use Ref, Take tool tip to upper of measurement		Z	0.000
		3:Press F1, Measure Start		A	4.158
Delta Z Set		Delta Z Set	0.000	Aux. Coord	
		Do tool tip measure before do Delt		X	0.000
		1:Take tool tip to top of good		Y	0.000
		2:Press F3, Delta Z Set		Z	0.000

- Table X- Coordinate table

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

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

- Move the tool tip (Manual) to the surface of workpiece, press Z Delta Set, the distance between alignment tool and workpiece surface will be calculate and set into this workpiece Z coordinate.
- One Tool One Workpiece is finished
- One Tool Many Workpieces

G54	N0 L1	Offset/Setting	2013/8/22	18:13:59
Auto Tool Function AutoTool 2 1:One Tool One Workpiece,2:On 3:Many Tools Many Workpie			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	
				
Delta Z Set 				
Feedrate F 0.000 Use Reference 0 Ref Coord. X 0.000 Ref Coord. Y 0.000 Start Coord. Z 0.000 Min. Z Mach. H 0.000 Select if use Ref Point 1:Set All measure parameter 2:If not use Ref, Take tool tip to upper of measurement 3:Press F1, Measure Start WorkPoiece No. P 0 Do tool tip measure before do Delt 1:Take tool tip to top of good 2:Press F3, Delta Z Set				
(1~3)			Ready Auto Alarm	

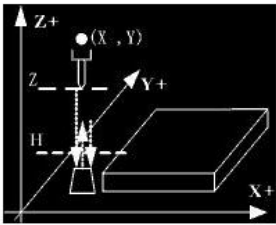
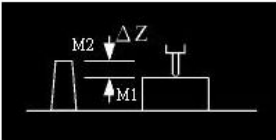
- Set the Auto Tool number as 2.(left-upper corner)
- Set the feed rate F for the first time alignment and pull back every time.
- Please skip to next step setting XY Ref. Coord. Teach if tool alignment equipment already has fixed mechanical coordinate. If current position is the same with tool alignment equipment, please set Use Reference as 0.
- While setting the XY Ref. Coord. Teach, you will need code "520", enter it and choose yes. These values will be entered in to Ref coord.X and Ref coord.Y.
- Move the cursor to Start Coord. Z, setting the start point for the auto alignment. Press F10 and then press Z Mach. Coord. Teach, you can insert the current Z coord. value into Start Coord. Z.
- Move the cursor to the Min. Z Mach. H, setting the minimum height for the alignment. System will pop out warning message and stop if the tool is lower than it.
- Please switch to auto mode after setting these and then press Start.
- After finishing alignment, this tool height value will be saved into Aux. Coord. and External Shift.
- Move the tool tip (Manual) to the surface of workpiece, press Z Delta Set, the distance between alignment tool and workpiece surface will be calculate and set into this workpiece Z coordinate.
- Move the tool tip (Manual) to the surface of next workpiece, press Z Delta Set, the distance between alignment tool and workpiece surface will be calculate and set into this workpiece Z coordinate.

Workpiece coord No. P	Workpiece coord	Workpiece coord No. P	Workpiece coord
--------------------------	-----------------	--------------------------	-----------------

1	G54	2	G55
3	G56	4	G57
5	G58		

Z Delta set table

- If you have another workpieces need to set do the previous step again.
- Many Tool Many Workpieces

G54	N0 L1	Offset/Setting	2013/8/22	18:15:00
Auto Tool Function AutoTool 3 1:One Tool One Workpiece,2:On 3:Many Tools Many Workpie		Tool No. T 0 Feedrate F 0.000 Use Reference 0 Ref Coord. X 0.000 Ref Coord. Y 0.000 Start Coord. Z 0.000 Min. Z Mach. H 0.000 Select if use Ref Point 1:Set All measure parameter 2:If not use Ref, Take tool tip to upper of measurement 3:Press F1, Measure Start	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	
 Delta Z Set 		WorkPoiece No. P 0 Do tool tip measure before do Delt 1:Take tool tip to top of good 2:Press F3, Delta Z Set		
(1~3)		●Ready Auto Alarm		

- Set the Auto Tool number as 3.(left-upper corner)
- Set the Tool No. T for the tool you want to align.
- Set the feed rate F for the first time alignment and pull back every time.
- Please skip to next step setting XY Ref. Coord. Teach if tool alignment equipment already has fixed mechanical coordinate. If current position is the same with tool alignment equipment, please set Use Reference as 0.
- While setting the XY Ref. Coord. Teach, you will need code "520", enter it and choose yes. These values will be entered in to Ref coord.X and Ref coord.Y.
- Move the cursor to Start Coord. Z, setting the start point for the auto alignment. Press F10 and then press Z Mach. Coord. Teach, you can insert the current Z coord. values into Start Coord. Z.
- Move the cursor to the Min. Z Mach. H, setting the minimum height for the alignment. System will pop out warning message and stop if the tool is lower than it.
- Please switch to auto mode after setting these and then press Start.
- After finishing alignment, this tool height value will be saved into Aux. Coord. and External Shift.
- Move the tool tip (Manual) to the surface of workpiece, press Z Delta Set, the distance between alignment tool and workpiece surface will be calculate and set into this workpiece Z coordinate.

- Move the tool tip (Manual) to the surface of next workpiece, press Z Delta Set, the distance between alignment tool and workpiece surface will be calculate and set into this workpiece Z coordinate.
- If you have another workpieces and tools need to set do the previous step again.
- Finish alignment.

Tool No. Setting

This Chapter will describe how to set the tool No.

- Purpose:
 - We need to confirm the relation between Tool No. and real tool in order to change tool correctly.
- Condition:
 - Both Auto mode or Manual mode are OK..
- Operation Method
 - Tool Set→Tool No.
 - You will see the table about Tool No. and Tool MG.
 - MG5—T = 7 means we install tool No.7 at Tool case No.5

G54	N0 L1		Offset/Setting		2013/8/22	18:16:09
MG And Tool No. T Table						
	MG	T	MG	T	MG	T
	1	0	11	0	21	0
	2	0	12	0	22	0
	3	0	13	0	23	0
	4	0	14	0	24	0
	5	0	15	0	25	0
	6	0	16	0	26	0
	7	0	17	0	27	0
	8	0	18	0	28	0
	9	0	19	0	29	0
	10	0	20	0	30	0
Turrent No		0				
Spindle No		0				
			●Ready	Auto	Alarm	

- This table will renew during change a new tool.
- Time to Modify
 - First initialized when it is made from factory.
 - Need to confirm the MG No. and T No. Manually when it is disordered.

Tool Manager Function

- Purpose:

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

customer need to use this function, please contact to machine maker.

G54	TEST N0 L1			Offset/Setting		2013/7/2	20:18:05
Tool Manager Function							
No	Turret	group	Information	Cur. Life	Max. Life	Announce	Status
01	0	0	U N C -	0	0	0	No Managed
02	0	0	U N C -	0	0	0	No Managed
03	0	0	U N C -	0	0	0	No Managed
04	0	0	U N C -	0	0	0	No Managed
05	0	0	U N C -	0	0	0	No Managed
06	0	0	U N C -	0	0	0	No Managed
07	0	0	U N C -	0	0	0	No Managed
08	0	0	U N C -	0	0	0	No Managed
09	0	0	U N C -	0	0	0	No Managed
10	0	0	U N C -	0	0	0	No Managed
11	0	0	U N C -	0	0	0	No Managed
12	0	0	U N C -	0	0	0	No Managed
				●Ready	Not Select		
(0~96) Turret tool No.				Alarm			

- Condition
 - Both auto and manual can be used.
- Operation
 - Pr.3228 is the on/off control of 「Tool management」
- Description
 - Turret
 - Current tool case no. that tool located.
 - Group
 - Same kind of tool within in one group, if the first tool of that group is on lock state or 「Tool life」 is end, whenever user use T code to change the tool, system will skip the first tool and use the second one, when the second one is lock or 「Tool life」 is end, will use the third one, and so on.
 - Tool information (Status)
 - L—Lock / U—Unlock
 - If the status of tool is lock, that tool can't be use and when T code is use to change the tool, system will skip that tool.
 - B—Large diameter Tool/ N—Normal Diameter Tool
 - Adjacent side of large diameter tool set is empty(for display)
 - T—working time T / C—Number of working times
 - Decide the current life time, the maximum life time, life time prediction, unit of timing and number of time.
 - R—effective value / - —non effective value.
 - Current tool are using tool management or not.
 - Current Life time
 - Current Tool Using Condition
 - Maximum Life Time

- Maximum lifetime of tool.
- Lifetime prediction
 - When lifetime of tool is greater than lifetime prediction, alarm will be show up.
- Current Status of Tool
 - (0)Without management: Set values are not effective.
 - (1) Without use: Lifetime of tool is zero.
 - (2) Usable: $0 < \text{Tool Life Time} < \text{lifetime prediction}$
 - (3) End prediction: $\text{Lifetime prediction} < \text{Tool Lifetime} < \text{Maximum Lifetime}$
 - (4) End of Life: $\text{Maximum Lifetime} < \text{Tool Lifetime}$
 - (5)ware of tool

5.4 Program Preparation and Execute Machining

5.4.1 Select Machining Program

- Condition:
 - Except single block mode
- Specify current programming file as machining program
 - Operation:
 - i. Switch to edit page
 - ii. Press F1 "Execute", 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 "Execute", 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

5.4.2 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 error of content, they will be displayed on the screen with corresponding line number
 - c. F1(Step): 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. F6(Simu. Setting): Set simulation parameter

5.4.3 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

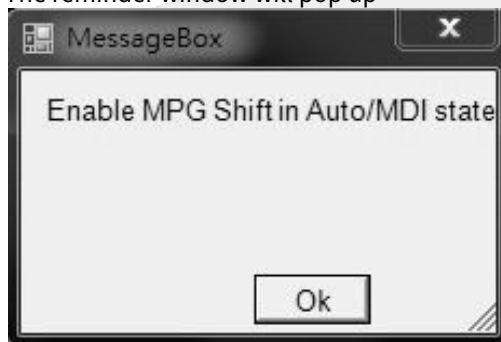
- Condition:
 - Only for single block(MDI) and auto mode
- Operation:
 - a. Press single block button on operation panel
 - b. "Cycle Start"
 - c. After executing a single block, machine decelerates to 0, state of system is switched to B-stop
 - d. Press "CYCLE START" again
 - e. After completing next 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.

5.4.4 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 $0 < \text{part count} < \text{required part count}$, at the time of changing machining files
 - At the moment of modifying the required part count which must be smaller than the part count.

5.4.5 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.

SYNTEC

5.4.6 Network Setting

G54	TEST NO 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.

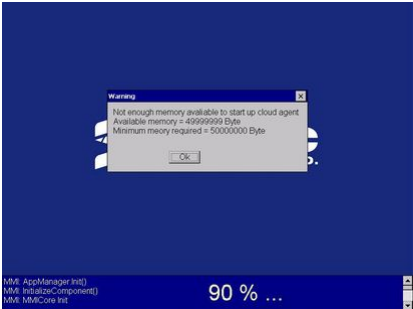
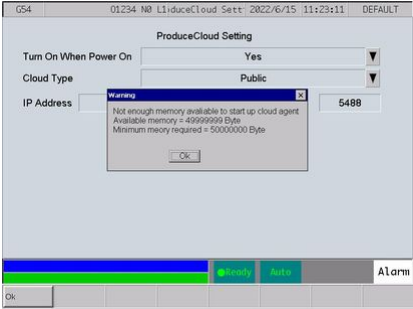
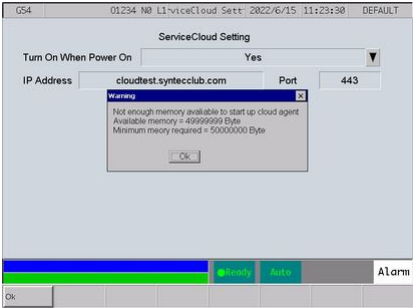
SYNTEC

CloudAgent Setting

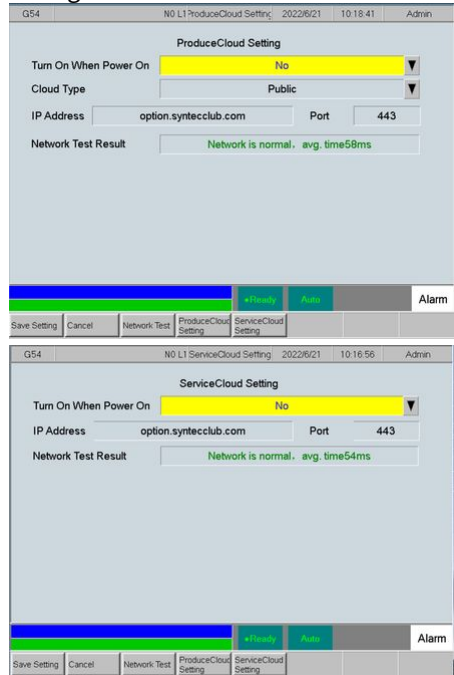
The image displays two side-by-side screenshots of the CloudAgent Setting interface. The left screenshot is titled 'ProduceCloud Setting' and the right is titled 'ServiceCloud Setting'. Both screens show a status bar at the top with 'G54', 'N0 L1', the screen title, date '2022/6/21', time '10:00:41' (left) or '10:02:19' (right), and 'Admin'. The main area contains settings for 'Turn On When Power On' (set to 'No'), 'Cloud Type' (set to 'Public'), 'IP Address' (set to 'option.syntecclub.com'), and 'Port' (set to '443'). A 'Network Test Result' field is present but empty. The bottom status bar includes buttons for 'Save Setting', 'Cancel', 'Network Test', 'ProduceCloud Setting', 'ServiceCloud Setting', and 'Alarm'.

- Path
 - 5 key: F5 Maintain → F2 Set Kernel Server → F3 CloudAgent Setting
 - 8 key: F5 Maintain → F2 Set Kernel Server → F3 CloudAgent Setting
 - 10+8 key: G5 Maintain → F2 Set Kernel Server → F3 CloudAgent Setting
- Explanation
 - Set up the parameters of ProduceCloud and ServiceCloud.
 - Require enough memory to start up CloudAgent. (REF : [Appendix1](#))
 - Network Test function (REF : [Appendix2](#))
 - All setting changes take effect after rebooting.
 - From 10.118.44H, 10.118.46B, 10.118.48M, 10.118.52G, 10.118.56A, 10.118.57 (included).
- Description of parameters
 - Turn On When Power On : Yes/No
 - Default Setting is "No". If choose "Yes", it will start up when power on.
 - Cloud Type : Public Cloud/Private Cloud
 - Only Support "ProduceCloud Setting".
 - Choose public cloud or private cloud according to server.
 - IP Address:
 - Input IP address of server. The default setting is option.syntecclub.com.
 - Port:
 - The default setting is 443.
- Description of fenu button
 - F1: Save Setting
 - Save setting and return to "Set Kernel Server". The changes of produce cloud and service cloud need to be saved separately.
 - F2: Cancel
 - Cancel and return to "Set Kernel Server".
 - F3 : Network Test
 - Doing a network test according to the IP address, and display the test result on the screen
 - F4: ProduceCloud Setting
 - Switch screen to "ProduceCloud Setting".
 - F5: ServiceCloud Setting
 - Switch screen to "ServiceCloud Setting".

• Appendix1 :

Version	Minimum available memory	Supplemental instruction
From 10.118.44H, 10.118.46B, 10.118.48M, 10.118.52G, 10.118.56A, 10.118.57	30MB	Require at least 30MB of memory space to start up ProduceCloud or ServiceCloud.
From 10.118.44J, 10.118.46J, 10.118.48U, 10.118.52O, 10.118.56I, 10.118.60C	50MB	1.Require at least 50MB of memory space to start up ProduceCloud or ServiceCloud.  2. Setting "Turn On When Power On" Choosing "Yes" will do memory check, if less than 50MB show the warning message and set the value to "No".  

• Appendix2 :

Version	Supplemental instruction
From 10.118.44J, 10.118.46J, 10.118.48U, 10.118.52O, 10.118.56I, 10.118.60C	a. Press the network test, it will ping the IP address 4 times. - Switching the setting of "Turn On When Power On", it will start to do network test. - During the network test, the button will be disabled. b. If all ping tests are successful, it will display the network is normal and the average time.  c. If there are 1~3 failures, it will display the network unstable and the number of failures. d. If all ping tests are failed, it will display the network error.

OPCUA Server

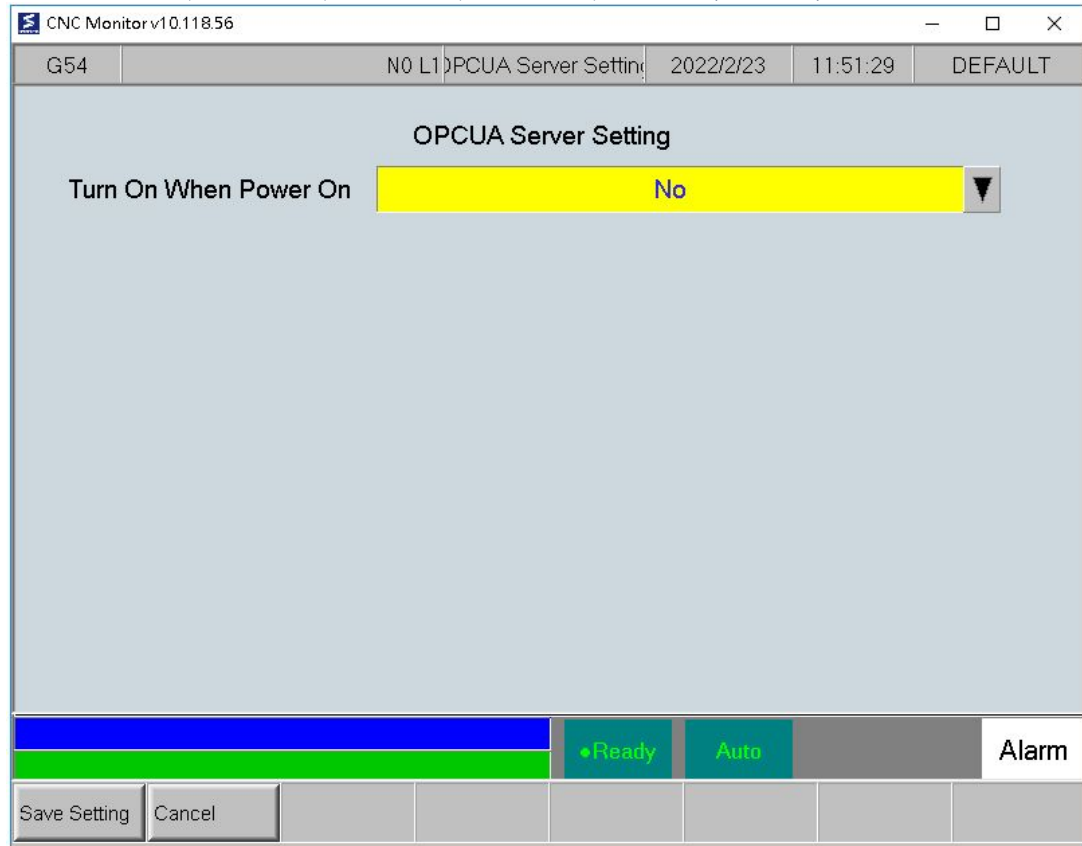
• Path

- 5 key: F5 Maintain → F2 Set Kernel Server → Next → F8 OPCUA Server
- 8 key: F5 Maintain → F2 Set Kernel Server → F8 OPCUA Server
- 10+8 key: G5 Maintain → F2 Set Kernel Server → F8 OPCUA Server

• Explanation

- Set the connection parameters of OPCUA server.

- All setting changes take effect after reboot.
- Need to enable option 51.
- Require at least 20MB of memory space to start up OPCUA server.
- From 10.118.46C, 10.118.48N, 10.118.52H, 10.118.56B, 10.118.57(included).



- Description of parameters
 - Turn On When Power On : Yes/No
 - Default Setting is "No". If choose "Yes", it will start up when power on.
- Description of fenu button
 - F1: Save Setting
 - Save setting and return to "Set Kernel Server".
 - F2: Cancel
 - Cancel and return to "Set Kernel Server".

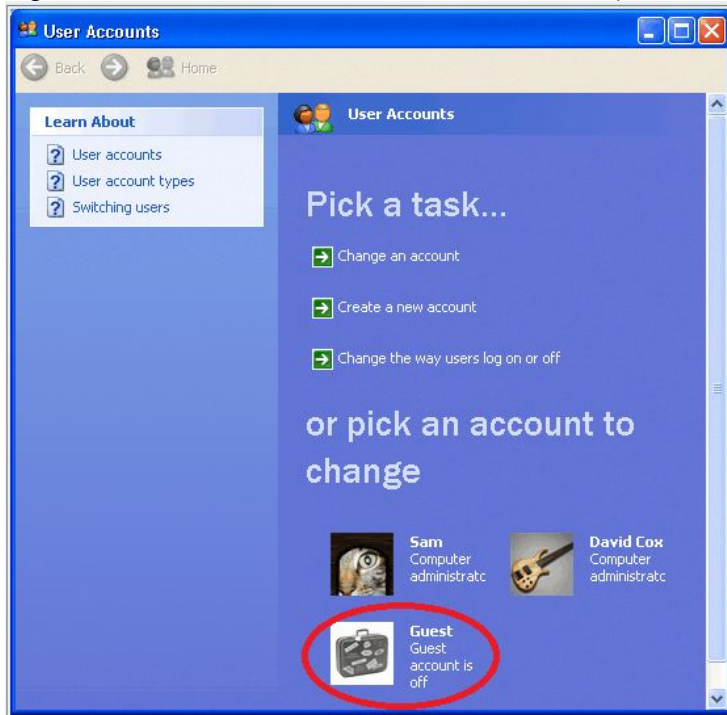
SYNTEC

5.4.7 PC Setting

XP OS

1. Guest account setting

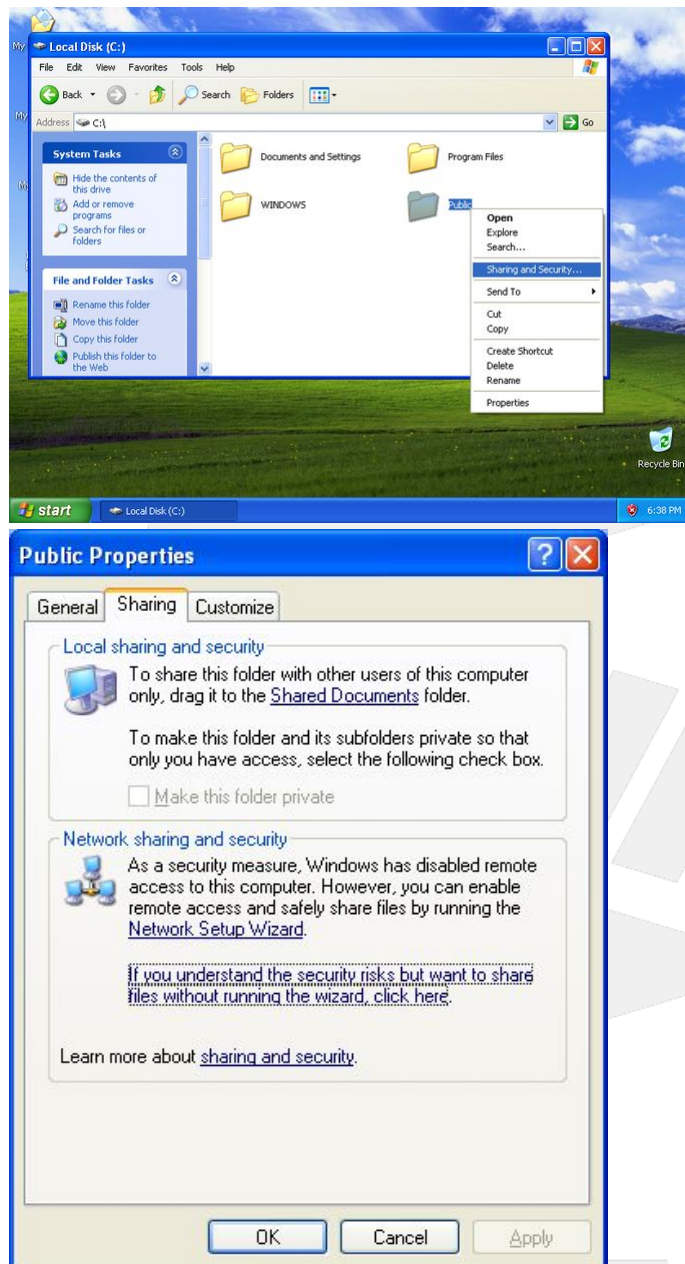
Log in as an Administrator and select "start"-> "control panel"-> "user account"-> Guest



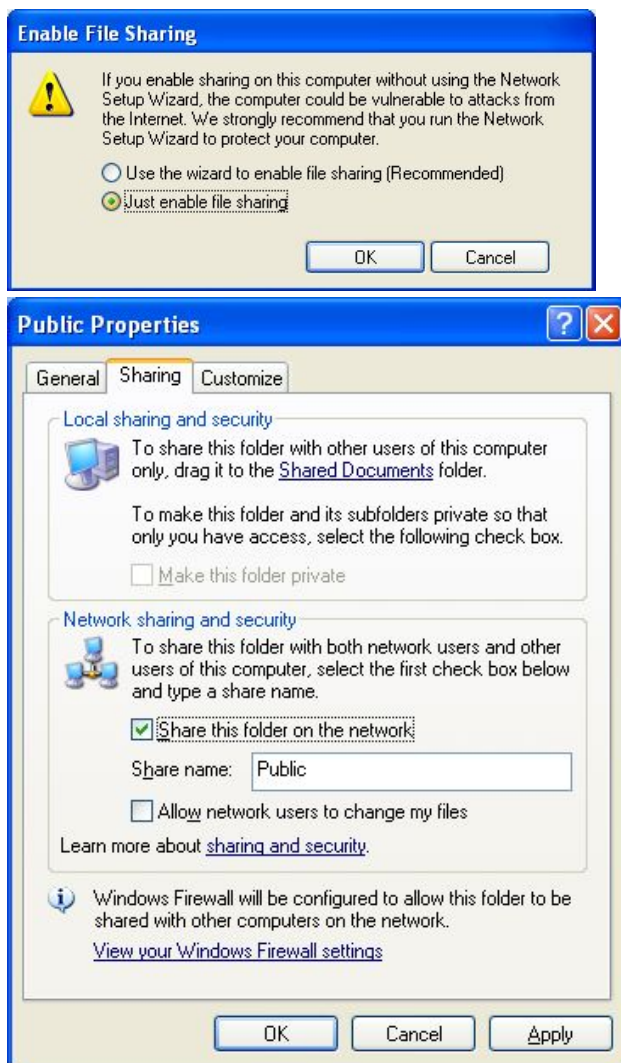
2. Sharing resource setting

- Right click the folder you want to share and select "Sharing and security"
- Click on "If you understand security risks but want to share files without running the wizard, click here"

SYNTEC



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

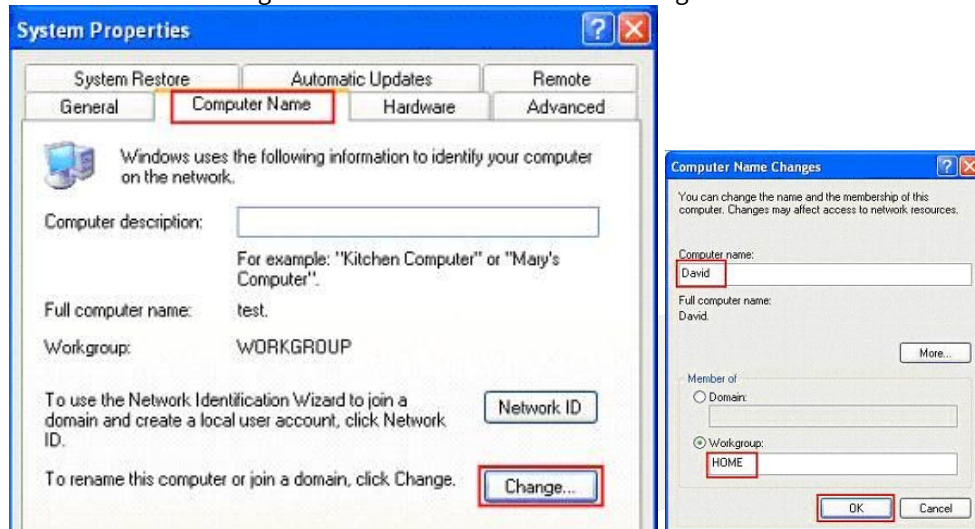


4. Setting PC name and workgroup

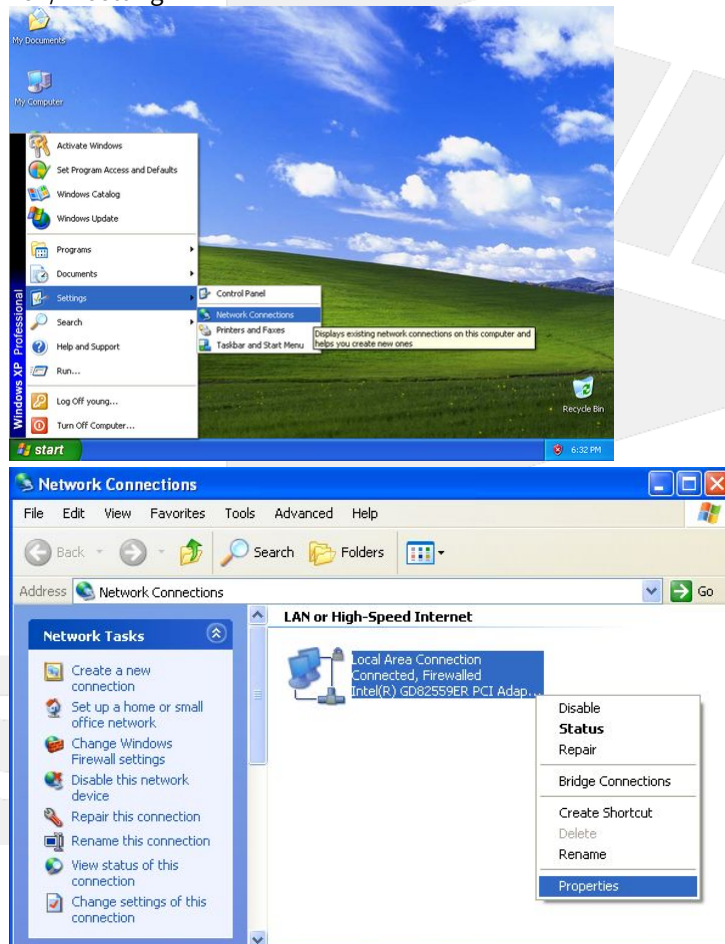
"Start" -> "control panel" -> "System" -> "change" to set "Computer Name" and "Workgroup", and

SYNTEC

remember these setting contents to use later on when setting controller.

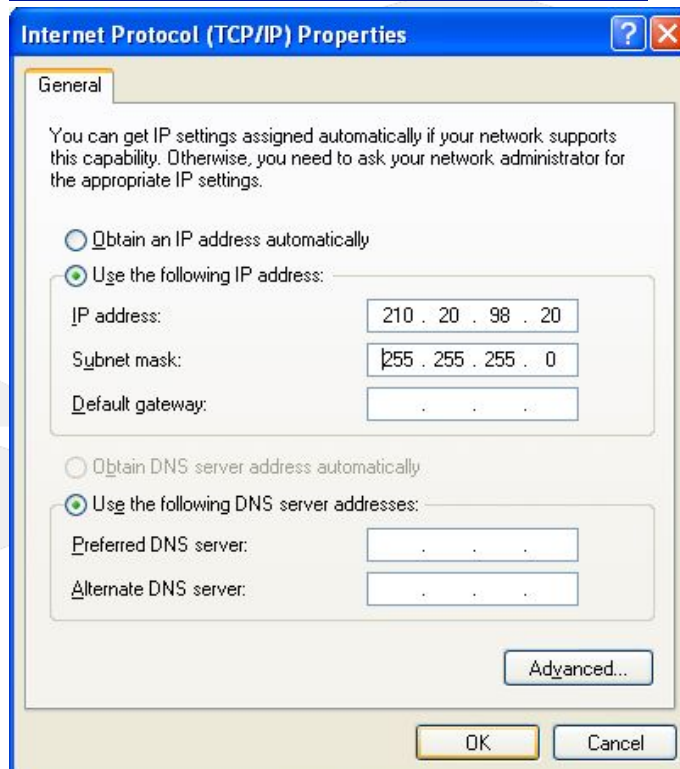
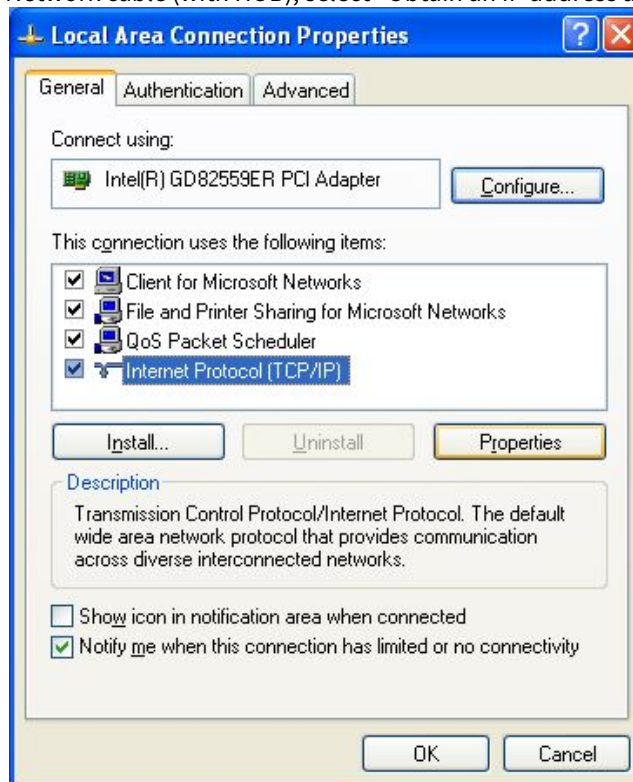


5. TCP/IP setting



"Start" => "Setting" => "Network connections" and right click on "Properties", and select "Internet Protocol [TCP/IP]"

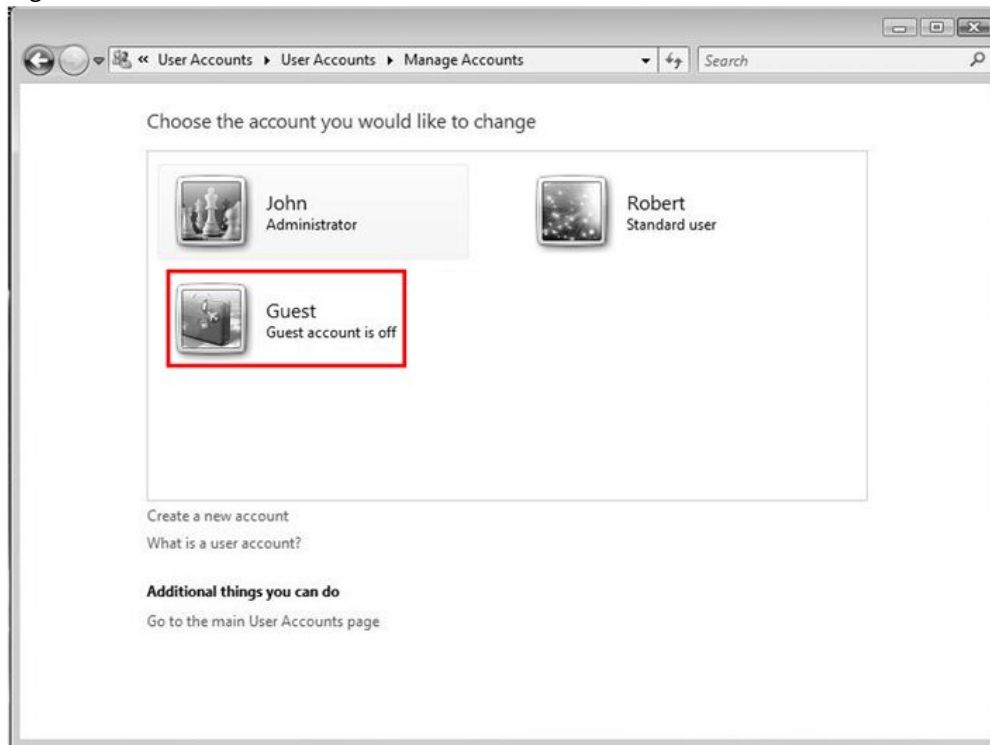
- Jumper cable (without HUB), select "use the following IP address" and enter IP address (the forth number is different from controller setting) and Subnet mask (same with controller setting)
- Network cable (with HUB), select "Obtain an IP address automatically"



VISTA OS

1. Guest account setting

Log in as Administrator and select "Start" "Control Panel" "User Account" Guest



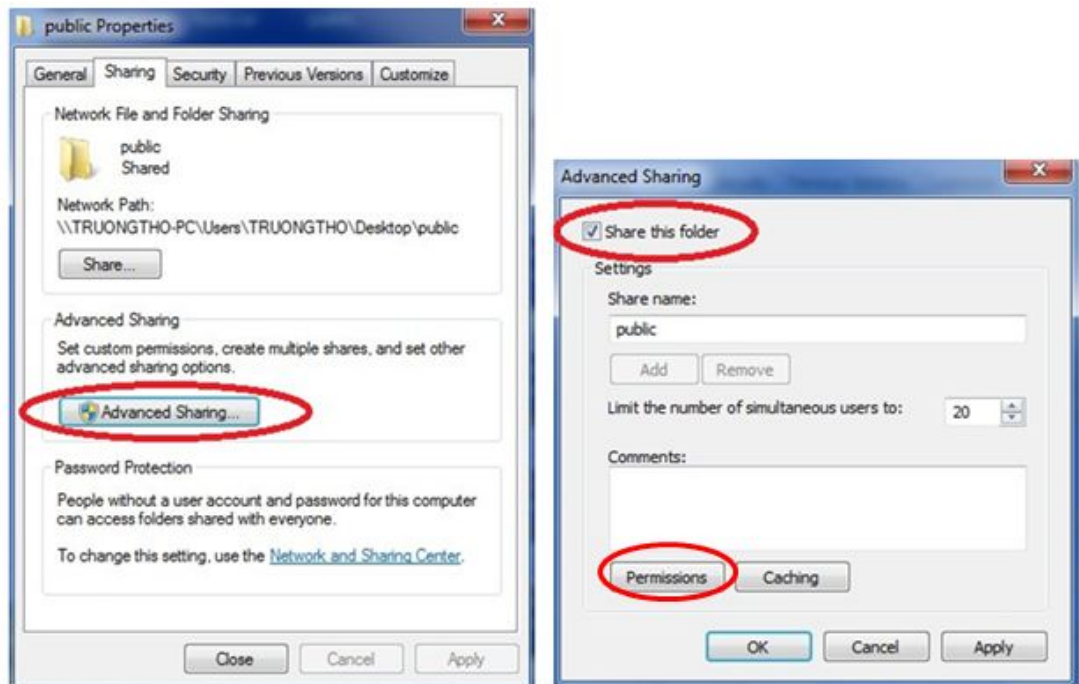
2. Sharing Resource Setting

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

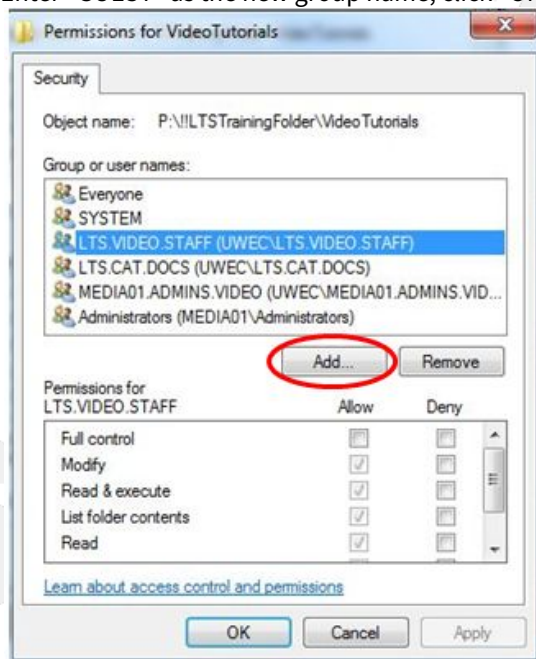
- a. Click on "advanced sharing"

SYNTEC

- b. Click on "share this folder"



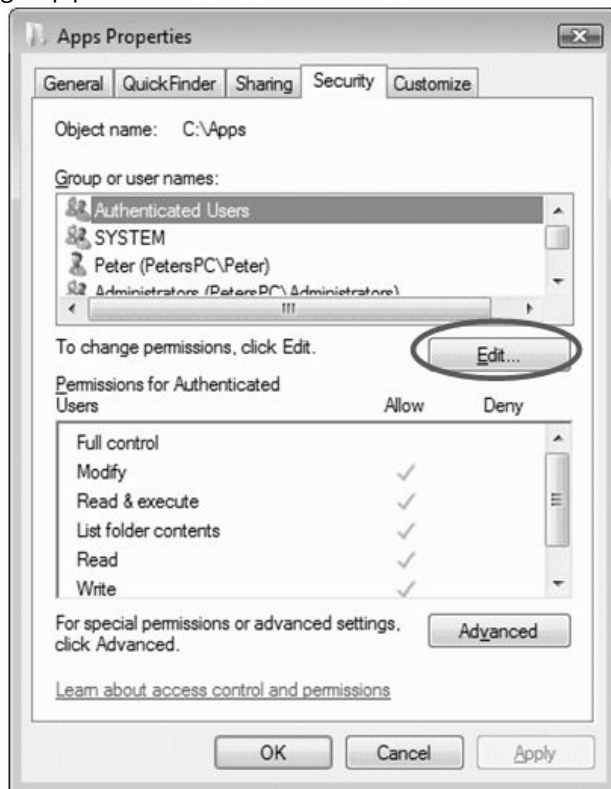
- c. Click on "permission"
 d. Click on "add"
 e. Enter "GUEST" as the new group name, click "OK" to complete setting



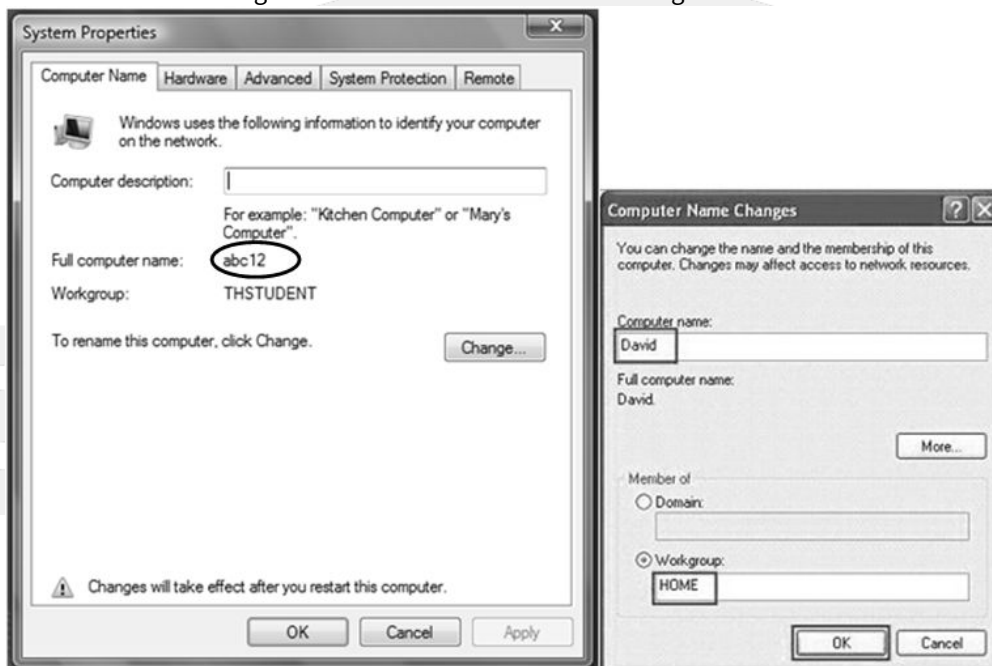
3. Security setting

Right click on folder to share-> properties-> security-> Edit-> add "Guest" as a new group, then open

group permissions to maximum.

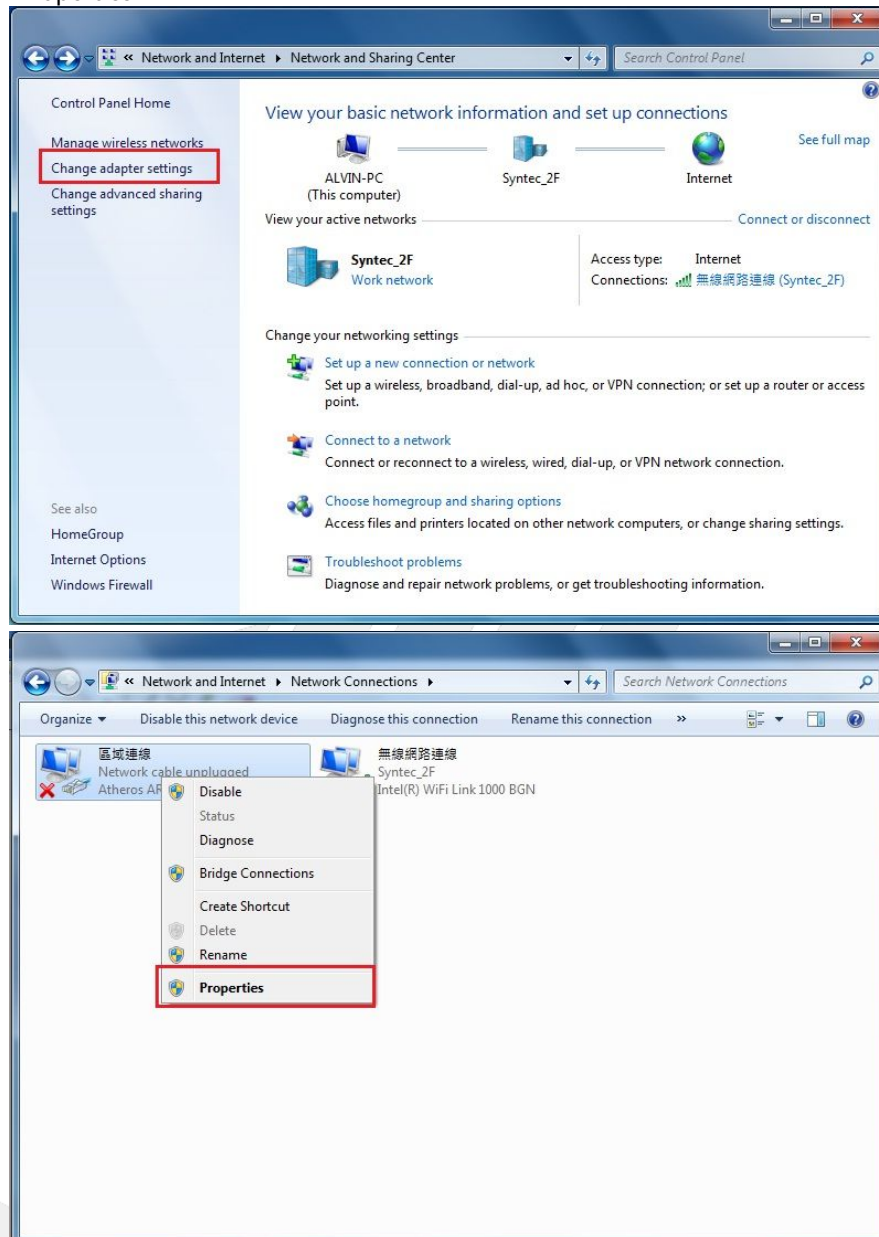


4. Setting PC name and workgroup
 "Start" -> "control panel" -> "System" -> "change" to set "Computer Name" and "Workgroup", and remember these setting contents to use later on when setting controller.



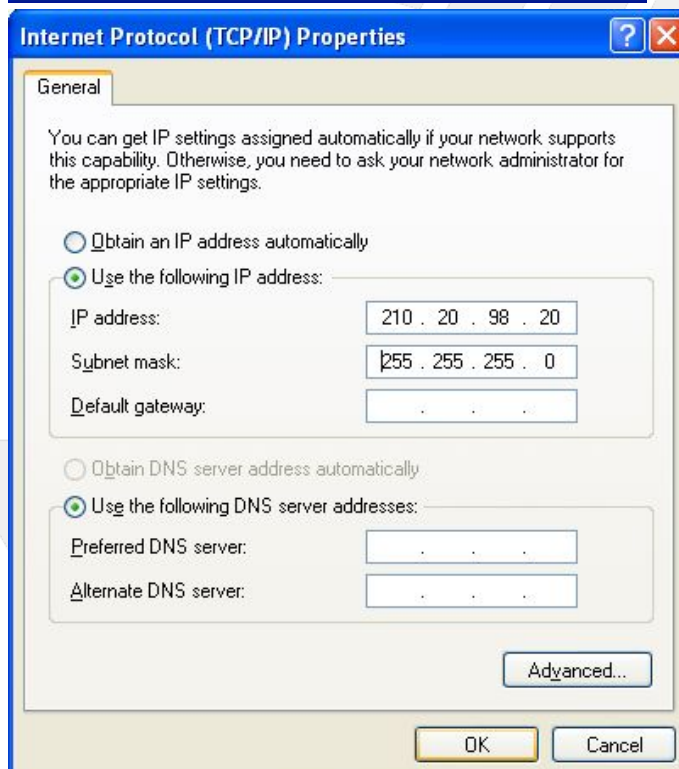
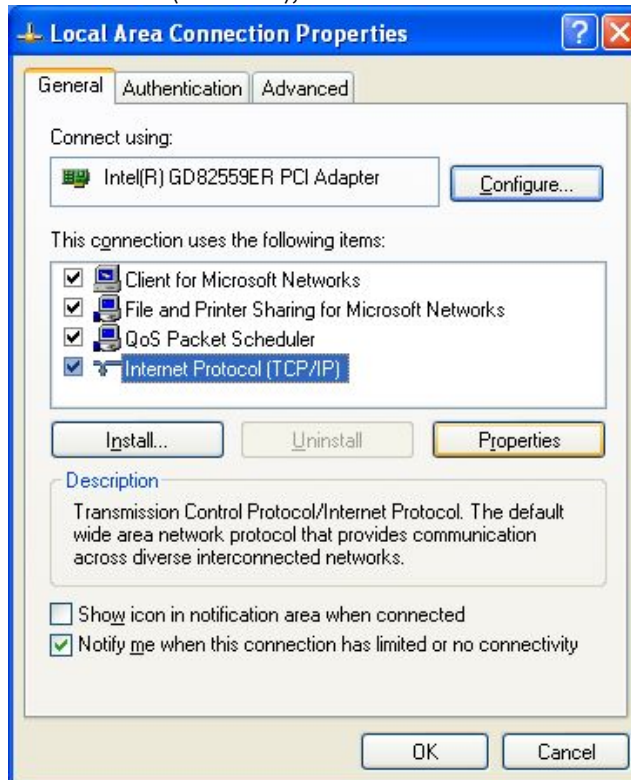
5. TCP/IP Setting

- a. "Start" -> "control panel" -> "Network and Internet"->"Network and Sharing Center" -> "Properties"



- b. Select "internet protocol(TCP/IP)" as shown below:
- Jumper cable (without HUB), select "use the following IP address" and enter IP address (the forth number is different from controller setting) and Subnet mask (same with controller setting)

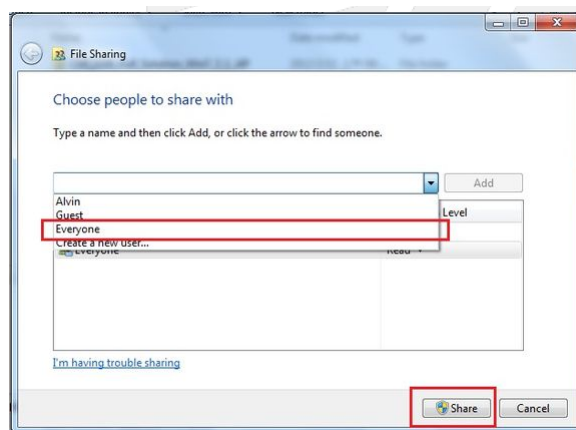
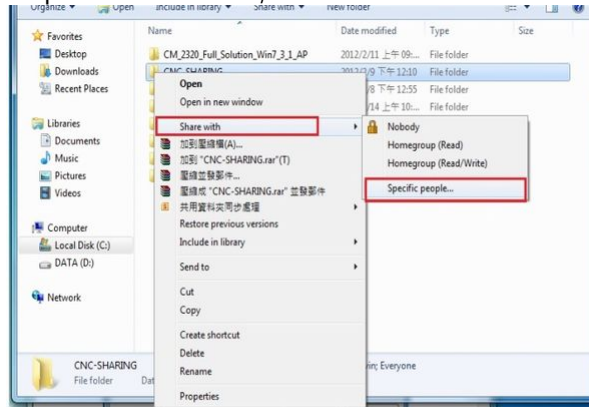
- Network cable (with HUB), select "Obtain an IP address automatically"



Win 7 OS

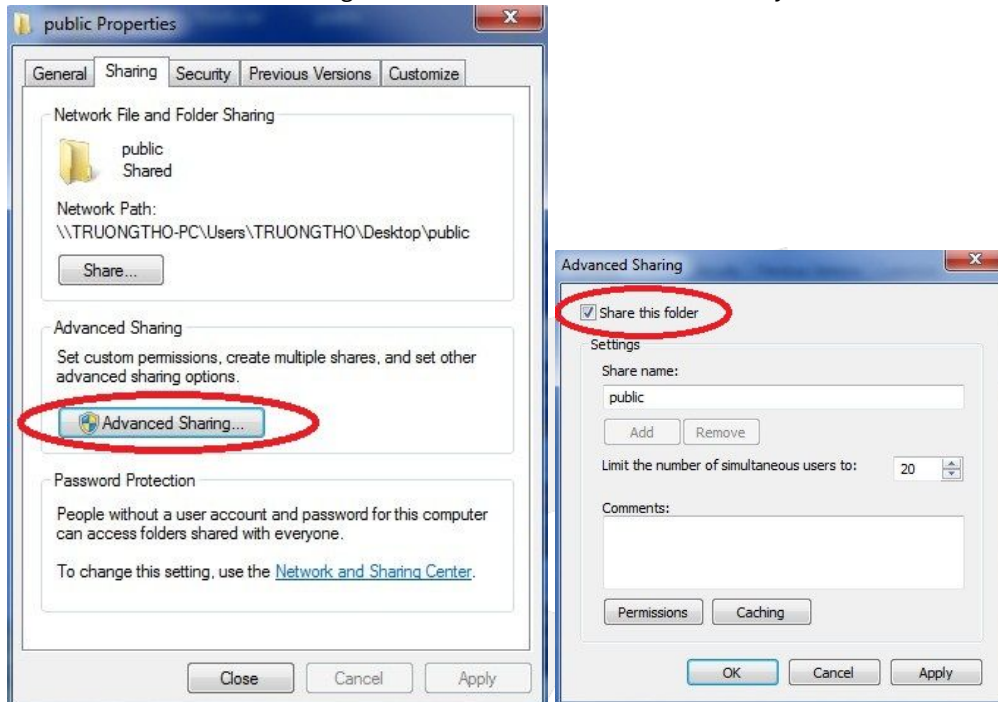
1. Sharing resource setting

- Right-click on folder wants to share, select "share with" and "specific people"
- Share this folder to everyone, and then click "Share" as follows.
- Set permission as write/read



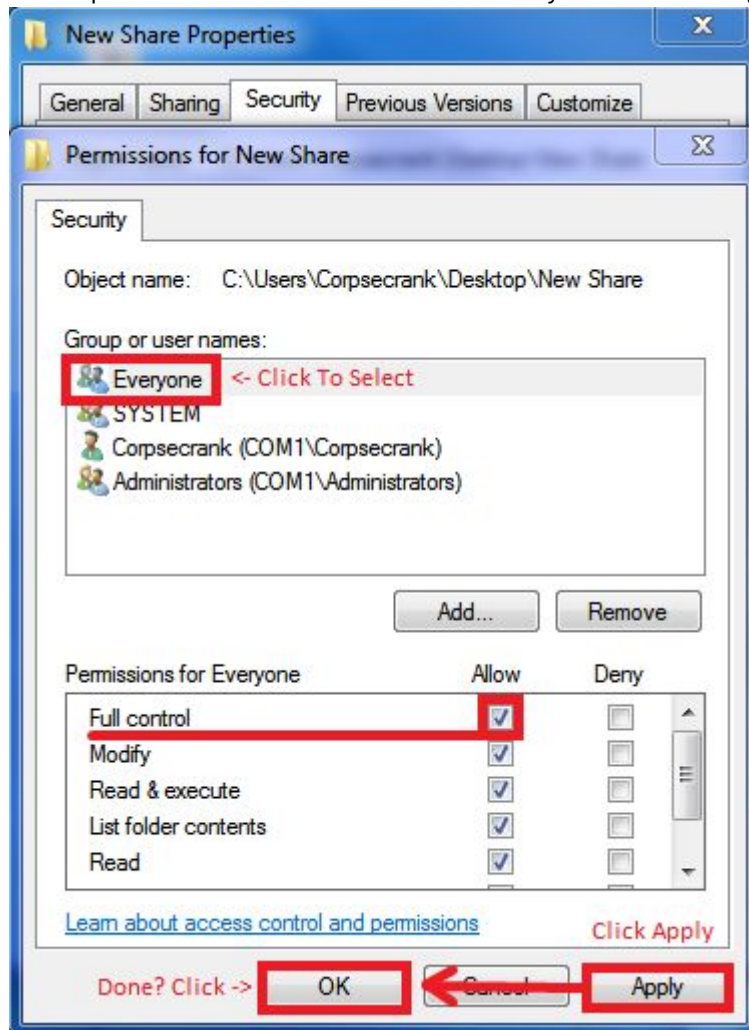
SYNTEC

- Left-click on "advanced sharing" and select "share this folder" to everyone.



SYNTEC

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

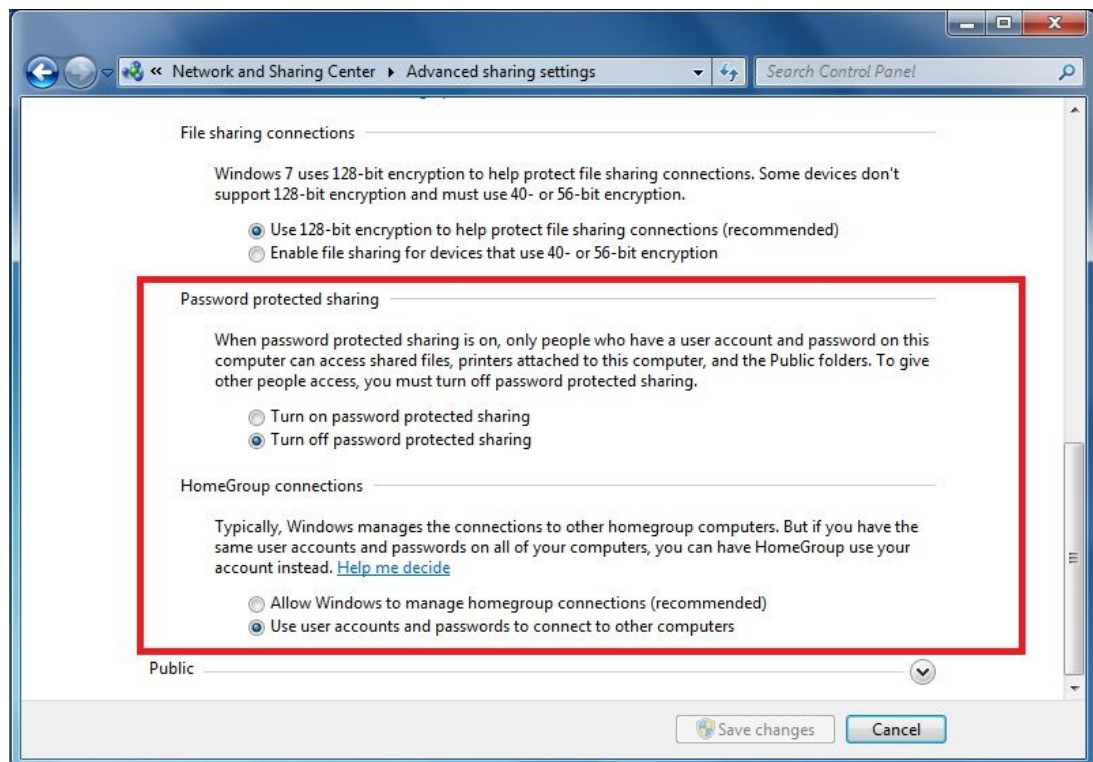


- Open "Network and sharing center", select "turn off password protected sharing" and "Open sharing....."

SYNTEC



SYNTEC

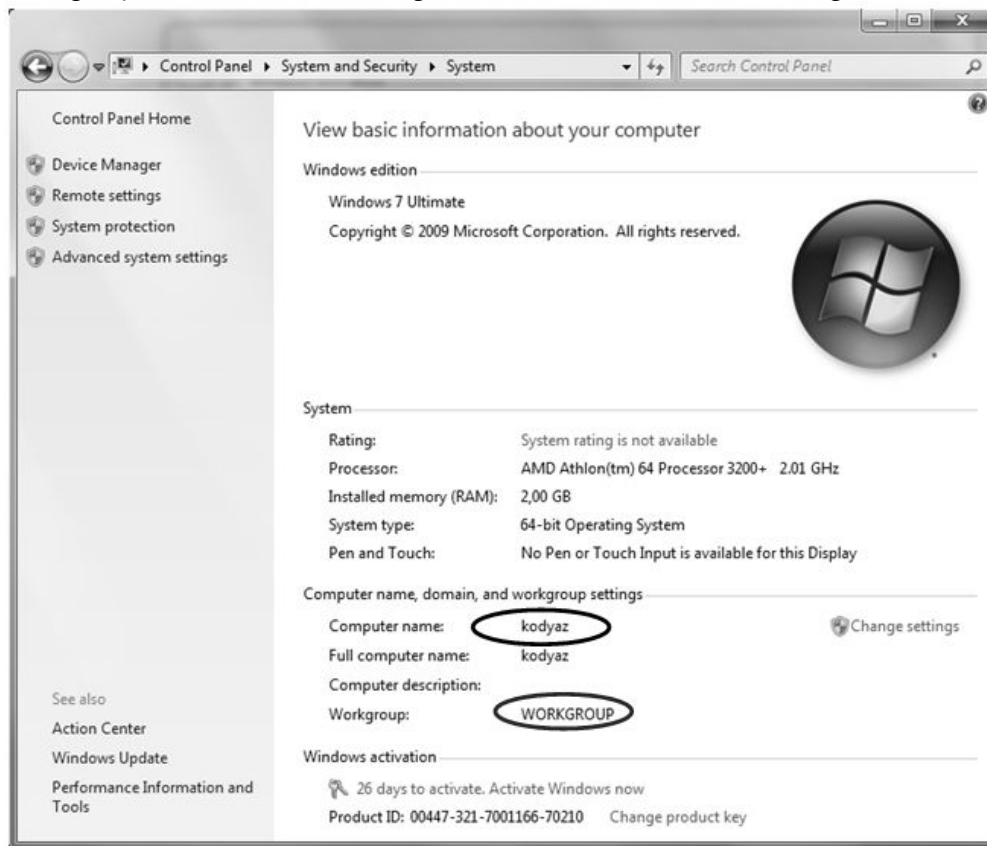


2. Setting PC name and workgroup

"Start"-> "control panel"->"system and security"-> "System"-> "change " to set "Computer Name" and

SYNTEC

"Workgroup", remember these setting contents to use later on when setting controller.

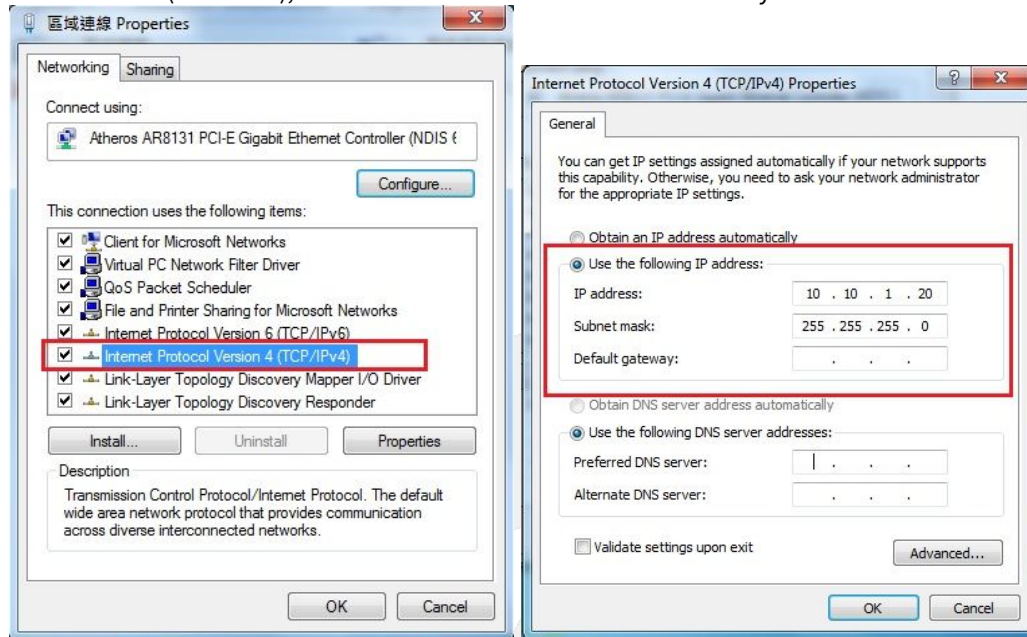


3. TCP/IP Setting

- Double click "Internet Protocol Version 4 (TCP/IPv4)"
- Jumper cable(without HUB), select "use the following IP address" and enter IP address(the forth number is different from controller setting) and Subnet mask(same with controller setting)

SYNTEC

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



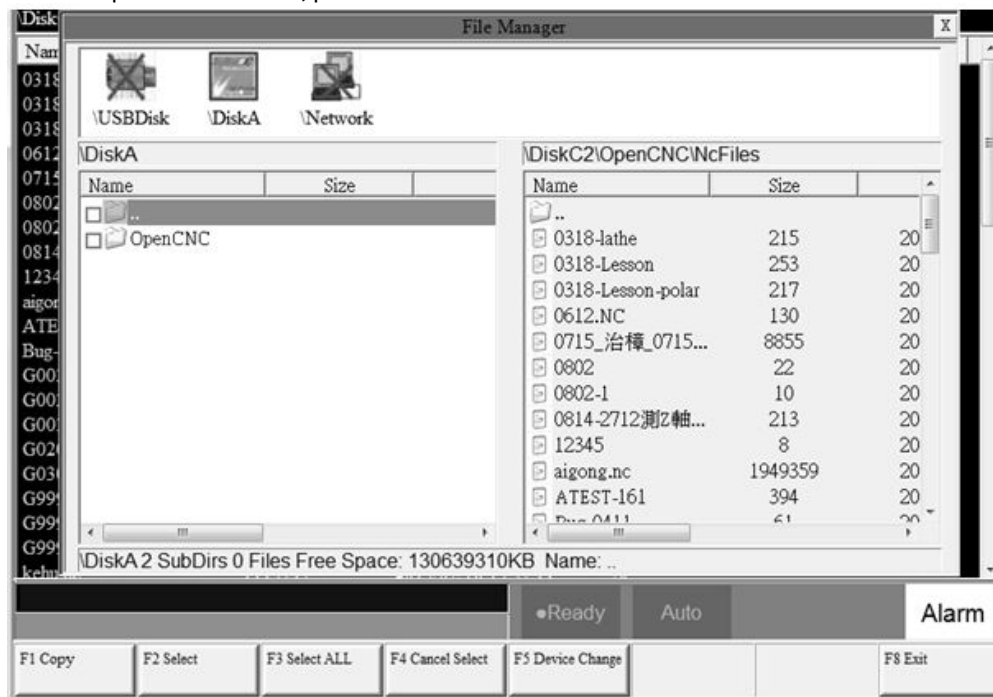
5.5 File Transfer

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

5.5.1 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.

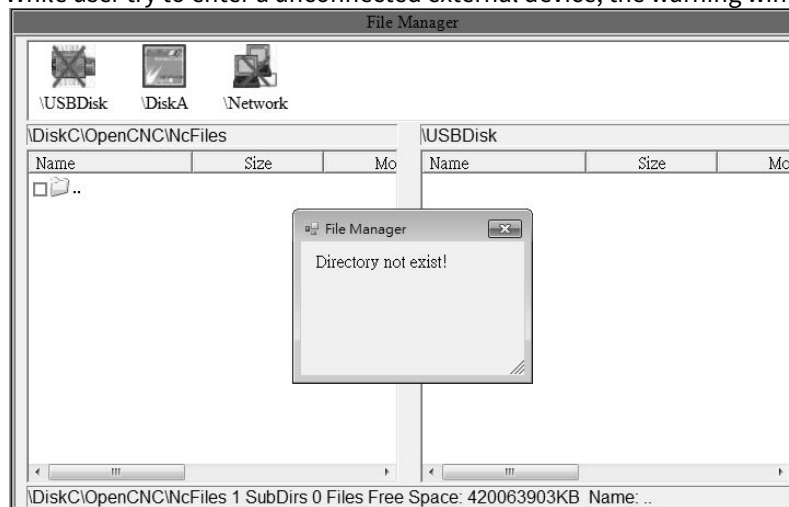
- j. After complete file transfer, press "Exit" or ESC to leave this window



5.5.2 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.



SYNTEC

6 **Appendix**

6.1 Contact Window



SYNTEC