



SYNTEC
TECHNOLOGY CO.,LTD.

60W-E Wood Router Machine Operation Manual

匯出日期：2020-12-08

修改日期：2019-11-27

[Manual](#)

Hardware



60W-E (Serial-bus controller)

SYNTEC

1 Software Spec.

Item	Content
Axes(Standard / Option)	4/6
Spindle Number	6
Axes Group Number	4
Workpiece Coordinate Number	100
BPT	1000
Software Version	10.116.54x, 10.118.12x, 10.118.x
Communication Protocol	M3/EtherCAT: 60W-E
Option	• 4-Axes RTCP (Option-29) • ROT (Option-32) • Auto Labeling Machine Function (Option-37) • Built-in CAM (Option-36)
Standard	• WorkingList • High speed high precision • Panel Nesting/Redo function • Support ZPL Printer to print Labels

SYNTEC

2 Manual Objective Machine

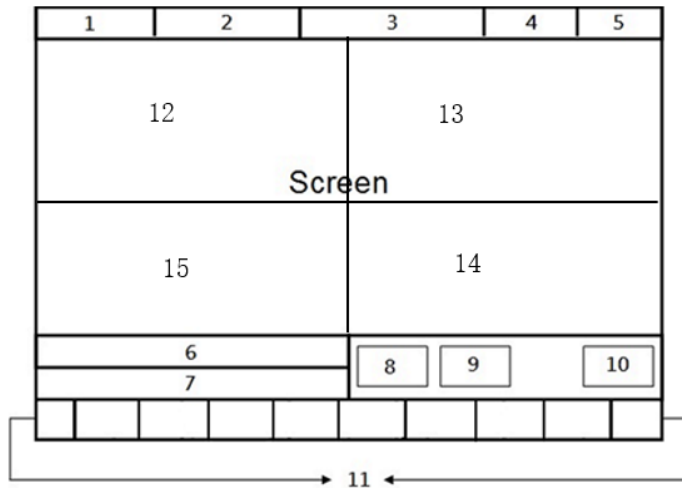
This Operation Manual is suitable for Syntec SCD-60WA and FC-60WA controller.



SYNTEC

3 Fenu and System Introduction

3.1 Main Screen

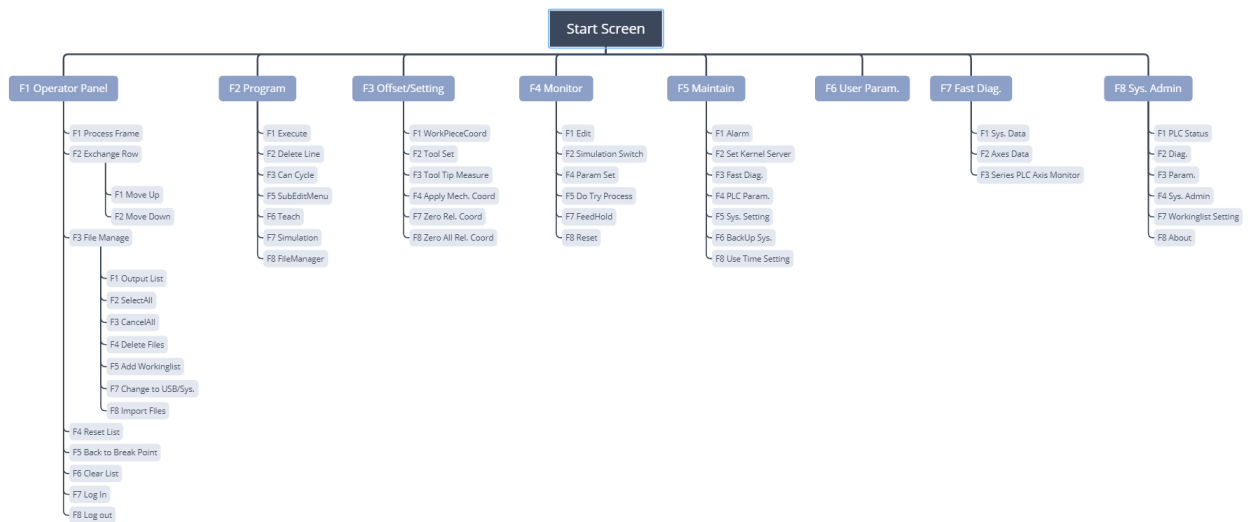


- **Screen component description**

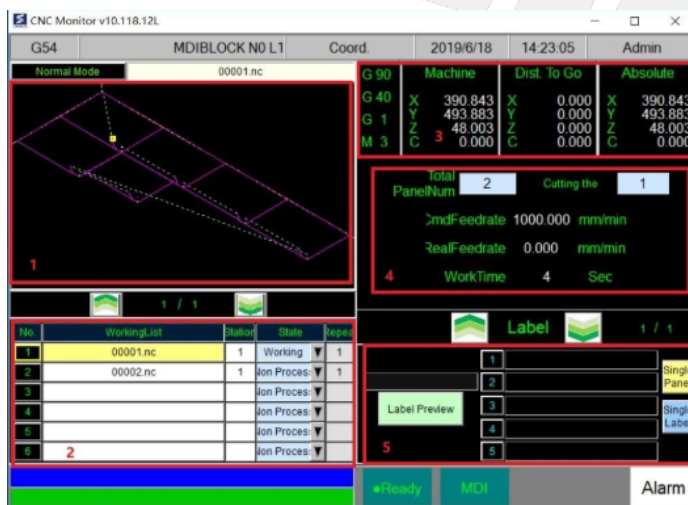
- Current Workpiece Coordinate**
- Current Processing file and process line**
- Current Screen**
- Date**
- Time**
- Data Input**
- Hint**
- Controller Status**
- Mode**
- Alarm**
- Fenu Selection**
- Process Simulation Previes**
- G-Code status and Coordinate**
- Label Information Previes**
- Workinglist**

SYNTEC

3.2 Function key Tree



3.3 Operator Panel (MainFenu: F1)



3.3.1 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
3	G-code State/Coordinate

Part	Item
4	Process Information
5	Tag List

- 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

3.3.2 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.

SYNTEC

Adjust selected item be later

Before



CNC Monitor v10.118.12L

G54 MDIBLOCK N0 L1 Coord. 2019/6/18 14:35:31 Admin

Normal Mode 00001.nc

G 90 Machine Dist. To Go Absolute

G 40 X 390.843 X 0.000 X 390.843

G 1 Y 493.883 Y 0.000 Y 493.883

M 3 Z 48.003 Z 0.000 Z 48.003

Total PanelNum 2 Cutting the 0

CmdFeedrate 1000.000 mm/min

RealFeedrate 0.000 mm/min

WorkTime 4 Sec

No.	WorkingList	Status	State	Panel
1	00001.nc	1	Working	1
2	00002.nc	1	Ion Procs	1
3			Ion Procs	
4			Ion Procs	
5			Ion Procs	
6			Ion Procs	

木工產品 Label List Material

1 2 3 4 5

Label Preview

Ready MDI Alarm

After



CNC Monitor v10.118.12L

G54 MDIBLOCK N0 L1 Coord. 2019/6/18 14:35:03 Admin

Normal Mode 00001.nc

G 90 Machine Dist. To Go Absolute

G 40 X 390.843 X 0.000 X 390.843

G 1 Y 493.883 Y 0.000 Y 493.883

M 3 Z 48.003 Z 0.000 Z 48.003

Total PanelNum 2 Cutting the 0

CmdFeedrate 1000.000 mm/min

RealFeedrate 0.000 mm/min

WorkTime 4 Sec

No.	WorkingList	Status	State	Panel
1	00002.nc	1	Ion Procs	1
2	00001.nc	1	Working	1
3			Ion Procs	
4			Ion Procs	
5			Ion Procs	
6			Ion Procs	

木工產品 Label List Material

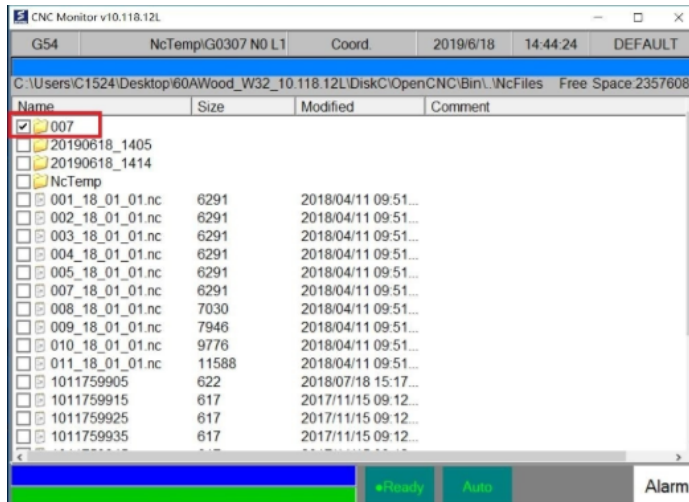
1 2 3 4 5

Label Preview

Ready MDI Alarm

SYNTEC

3.3.3 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.

3.3.4 Operator Panel-F4: Reset List

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

Reset List

Before

CNC Monitor v10.118.12L

G54 NcTempIG0307 N0 L1 Coord 2019/6/18 14:46:17 DEFAULT

Normal Mode 00001.nc

G 90 Machine Dist To Go Absolute

G 40 X 198.276 X 0.000 X 198.276

G 1 Y 250.550 Y 0.000 Y 250.550

M 3 Z 48.003 Z 0.000 Z 48.003

Total PanelNum 2 Cutting the 0

CmdFeedrate 1000.000 mm/min

RealFeedrate 0.000 mm/min

WorkTime 3 Sec

Label 1 / 1

1 2 3 4 5

Label Preview

Ready Auto Alarm

No	WorkingList	Status	State	Repeat
1	00001.nc	1	completed	0
2	00002.nc	1	completed	0
3			ion Proce	
4			ion Proce	
5			ion Proce	
6			ion Proce	

After

CNC Monitor v10.118.12L

G54 NcTempIG0307 N0 L1 Coord 2019/6/18 14:46:34 DEFAULT

Normal Mode 00001.nc

G 90 Machine Dist To Go Absolute

G 40 X 198.276 X 0.000 X 198.276

G 1 Y 250.550 Y 0.000 Y 250.550

M 3 Z 48.003 Z 0.000 Z 48.003

Total PanelNum 2 Cutting the 0

CmdFeedrate 1000.000 mm/min

RealFeedrate 0.000 mm/min

WorkTime 3 Sec

Label 1 / 1

1 2 3 4 5

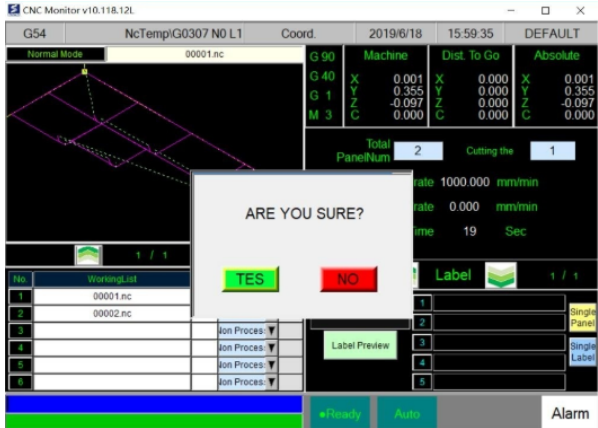
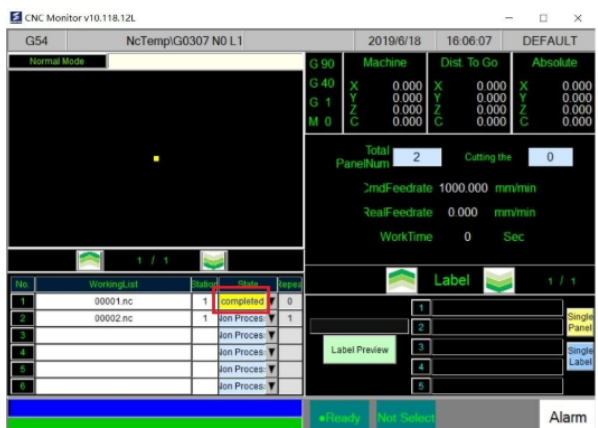
Label Preview

Ready Auto Alarm

No	WorkingList	Status	State	Repeat
1	00001.nc	1	ion Proce	1
2	00002.nc	1	ion Proce	1
3			ion Proce	
4			ion Proce	
5			ion Proce	
6			ion Proce	

SYNTEC

3.3.5 Operator Panel-NEXT-F1: Back to Break Point

Press NEXT-F1, 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 a unexpected event happen that cause a break to the processing file and you don't wanna do the whole process again

- users can press this button and the system will go back to the block before the break happen, and continue the working.
- **Hint: Please DO NOT press the button at normal condition, this might cause the panel process state to be wrong.**

3.3.6 Operator Panel-F5: Clear List

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

3.3.7 Operator Panel-NEXT-F4: Login

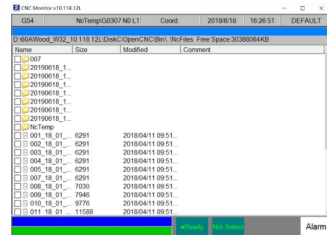
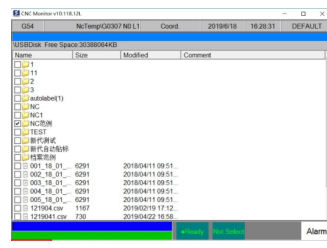
- Enter username and password to login to the system
- login different users can have different permissions
- Permissions can be set in "Authority mangement"

3.3.8 Operator Panel-NEXT-F5: LogOut

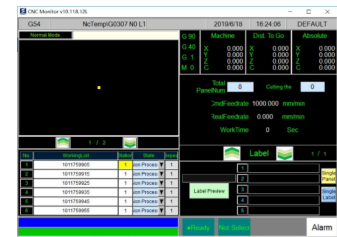
- logout the accessed account

3.3.9 Workinglist Manipulation

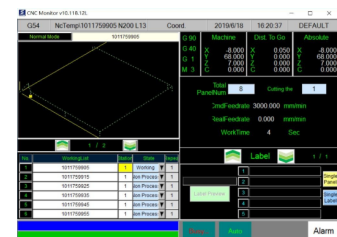
- Normal Mode
 - **Select Files/Folder, Output Workinglist**

• Operator Frame(F1) → F3 File Manage • F5: 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.

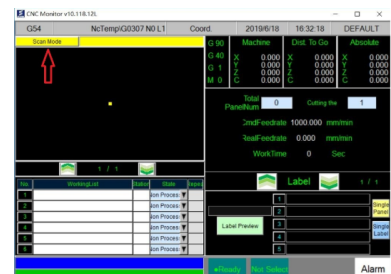


• BarCode Scanning Mode

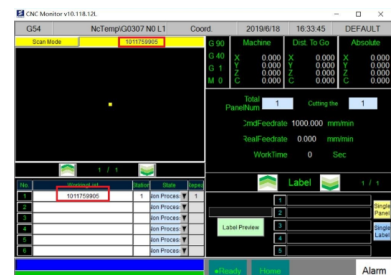
- Press The "Normal Mode" button, change to "Scan Mode", Press again back to "Normal Mode".
- Scan the barcode and the scanned NCfile will be added to workinglist by sequence, if no files found in system, there will show a pop-up hint

Figure

ScanMode

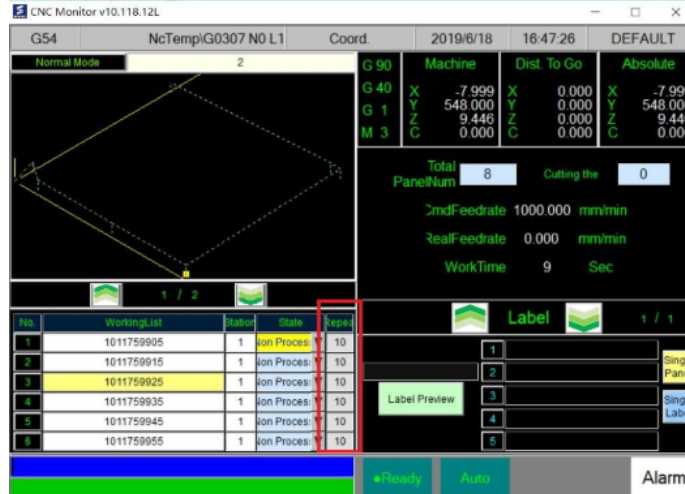


Scanning Barcode and add to Workinglist

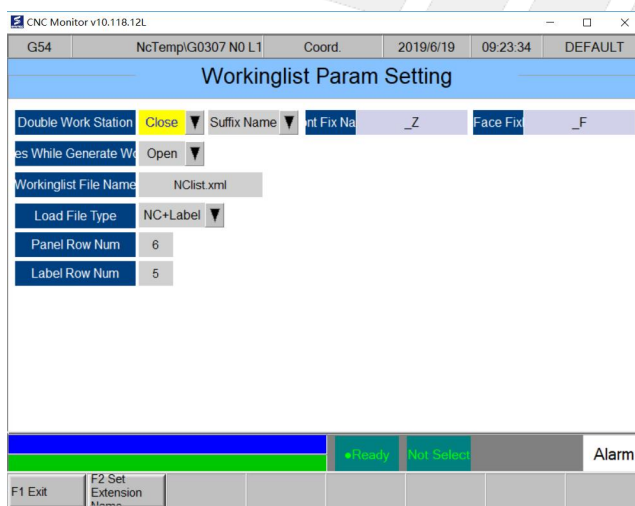


- Set Repeat Times for the Same File

- The default of repeat time is 1, every time the process done, it will minus 1 time.
- If set it be 10, this file will be repeat executed till the number be 0.

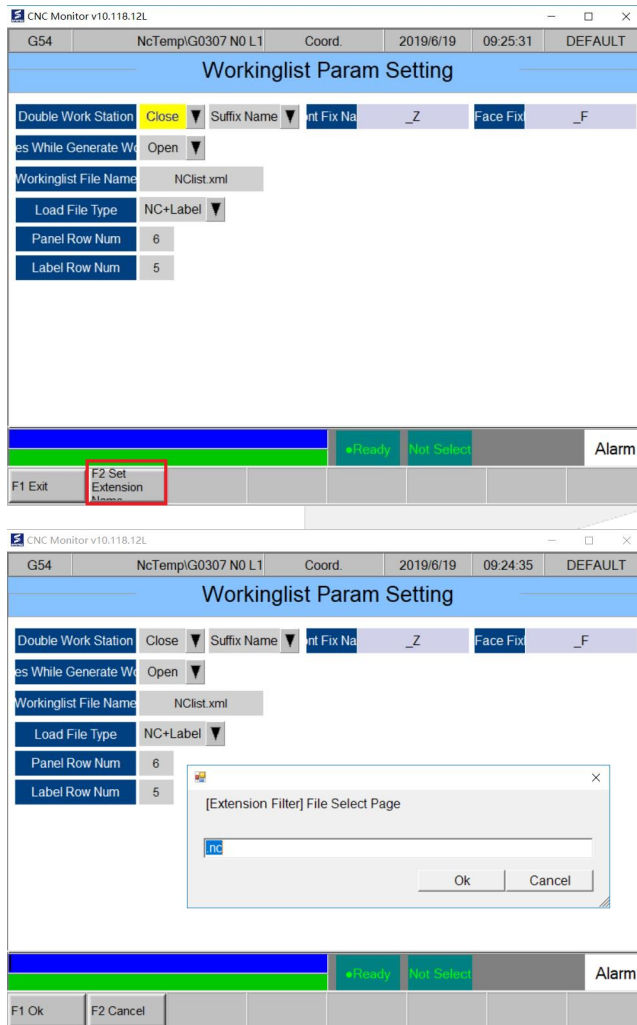


3.4 WorkinglistSetting (MainFenu F10 → F10 → F1)



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

3.4.1 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 xxxxx.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)

3.4.2 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

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																	

SYNTEC

	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																			

3.4.3 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.

3.4.4 Panel Row Number

- The number of rows in Workinglist

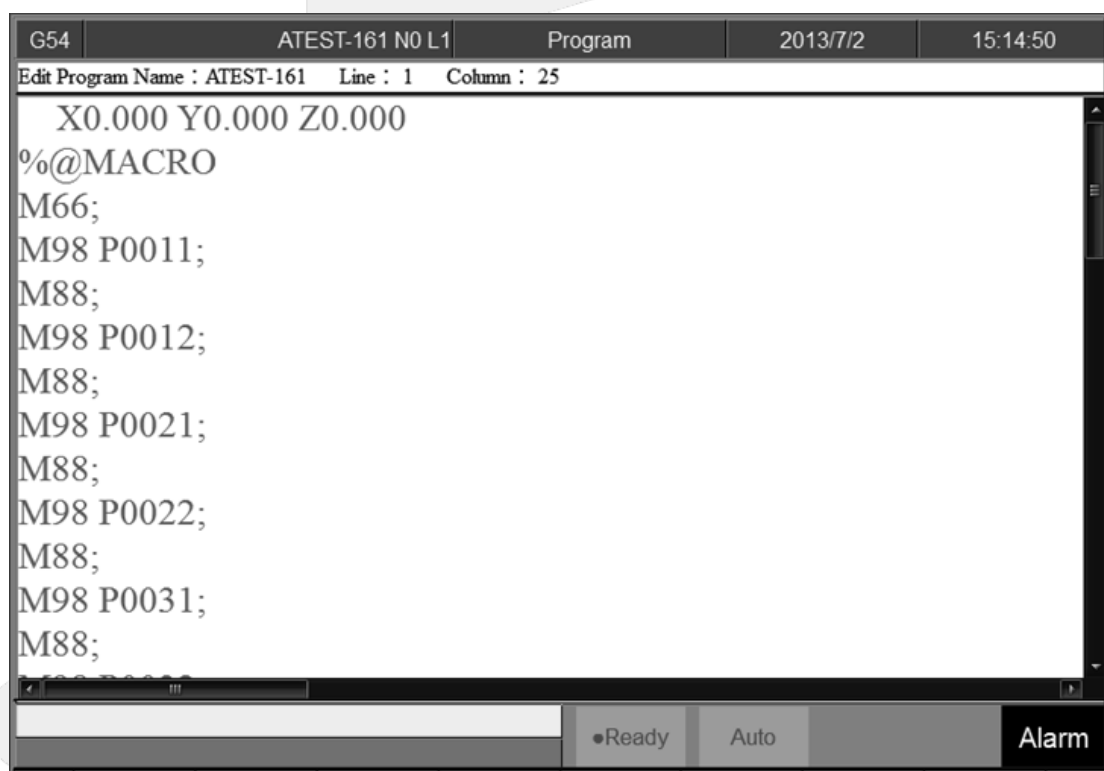
3.4.5 Label Row Number

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

3.4.6 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.

3.5 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」.

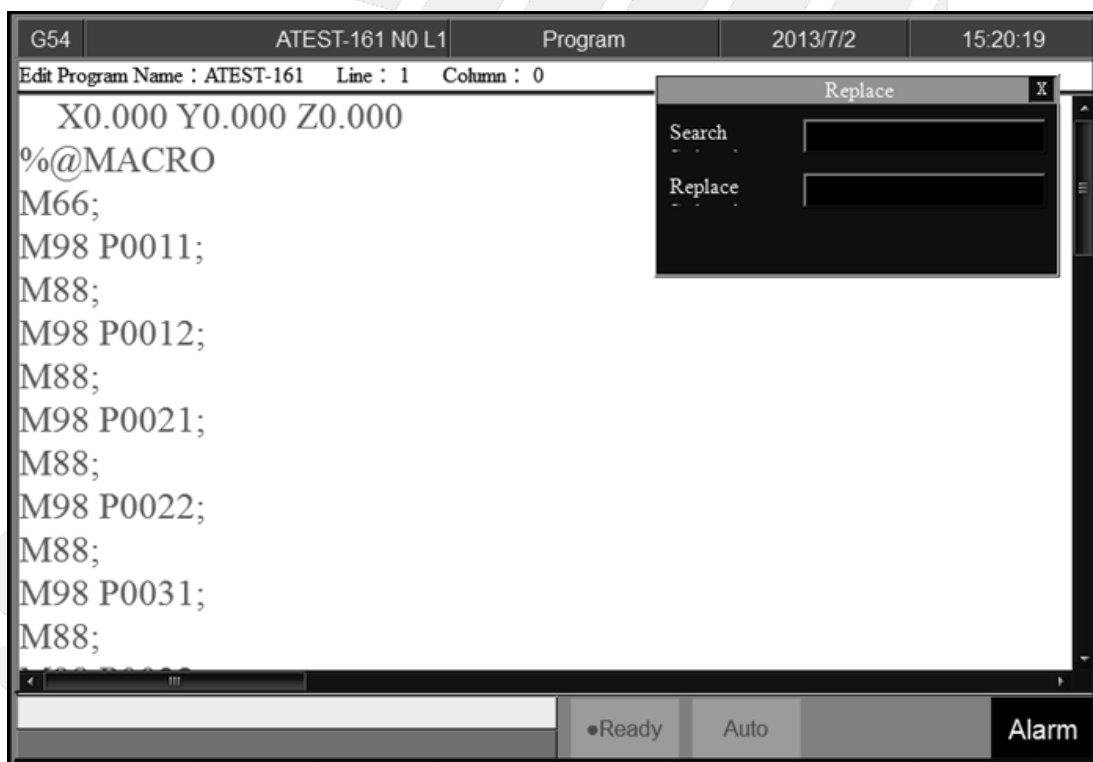
3.5.1 Execute

- 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.

3.5.2 Delete Line

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

3.5.3 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

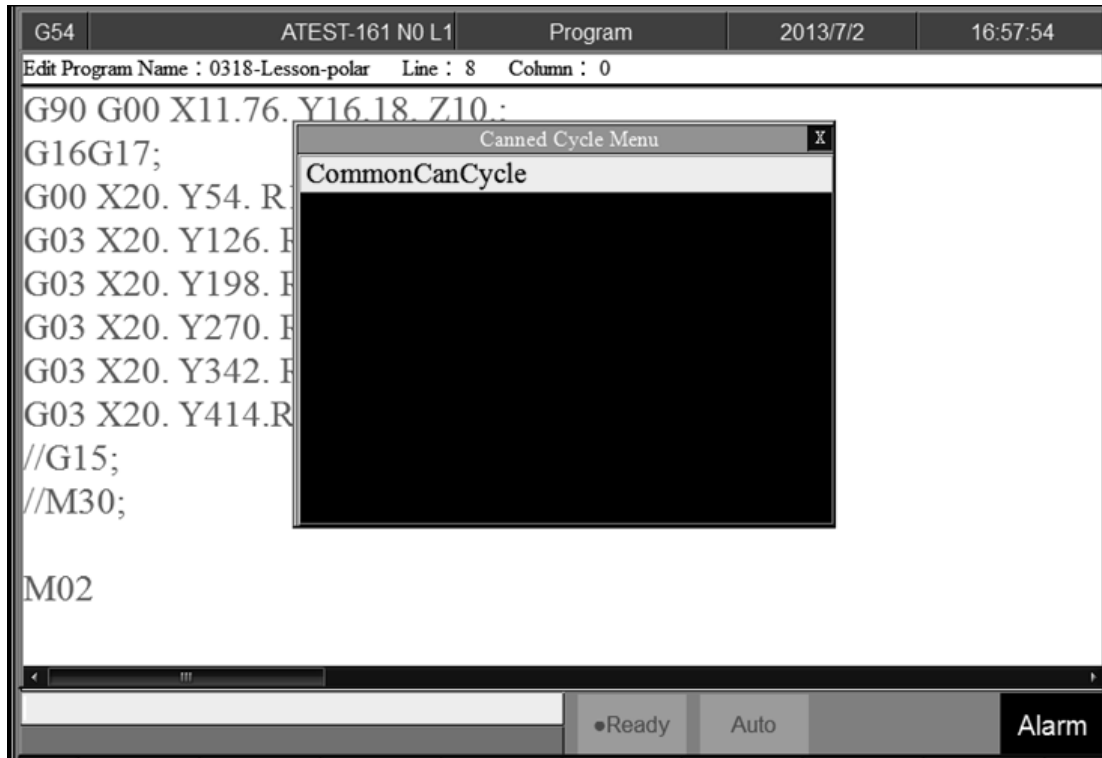
- Command
 - F2 Program → F3 Search/Replace → F3 Replace All
- Function
 - Replace all search text with input text.

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.

SYNTEC

3.5.4 Can Cycle



- Command
 - 5 Buttons type: F2 Program → Next → F1 Can Cycle
 - 8 Buttons type: F2 Program → F4 Can Cycle
- Function
 - Because SYNTEC system provides many kind of G code and different G code has each function. When editing the program, this function can help user to edit G code easily.

Insert Cycle

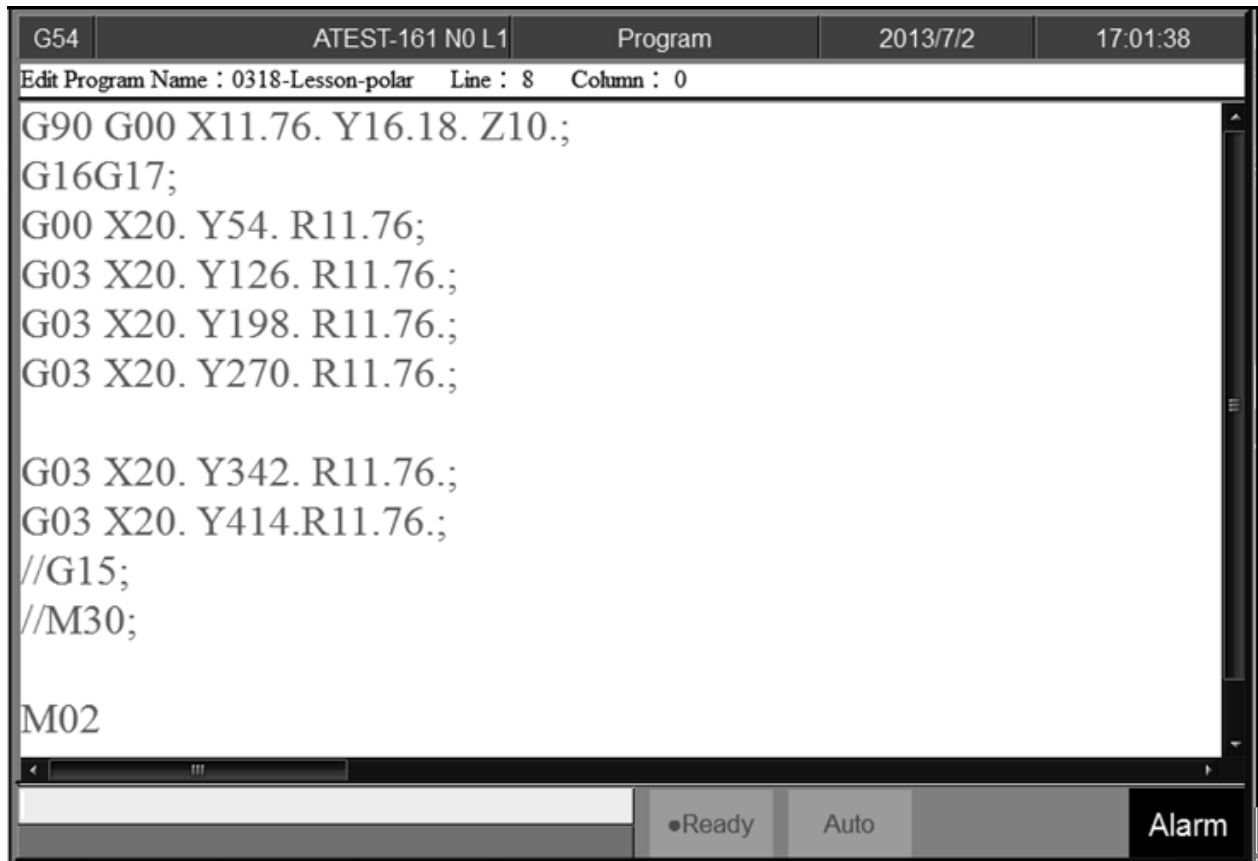
- Command
 - 5 buttons type: F2 Program → Next → F1 Can Cycle → F1 Insert Cycle
 - 8 buttons type: F2 Program → F4 Can Cycle → F1 Insert Cycle
- Function
 - Insert the required G code.
- Operation Method
 - Under the program edit mode, move the cursor to the desired location and press 「Insert Cycle」.
 - 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.

3.5.5 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.

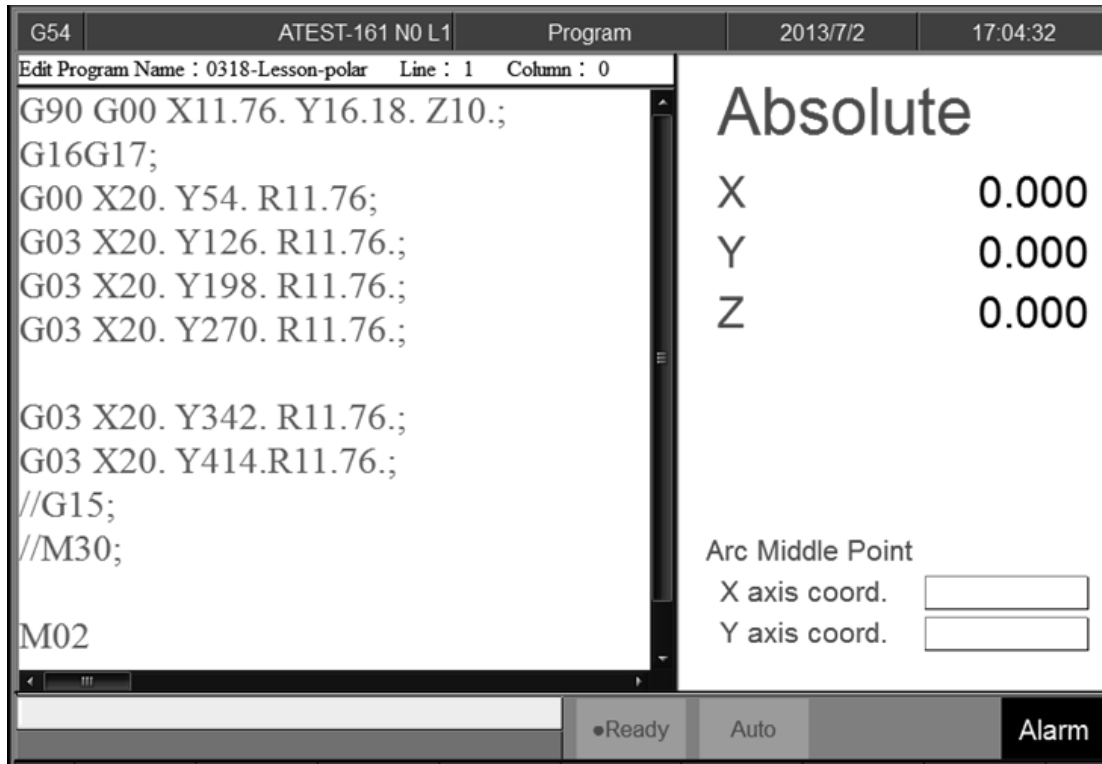
Block Copy

- 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.

3.5.6 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

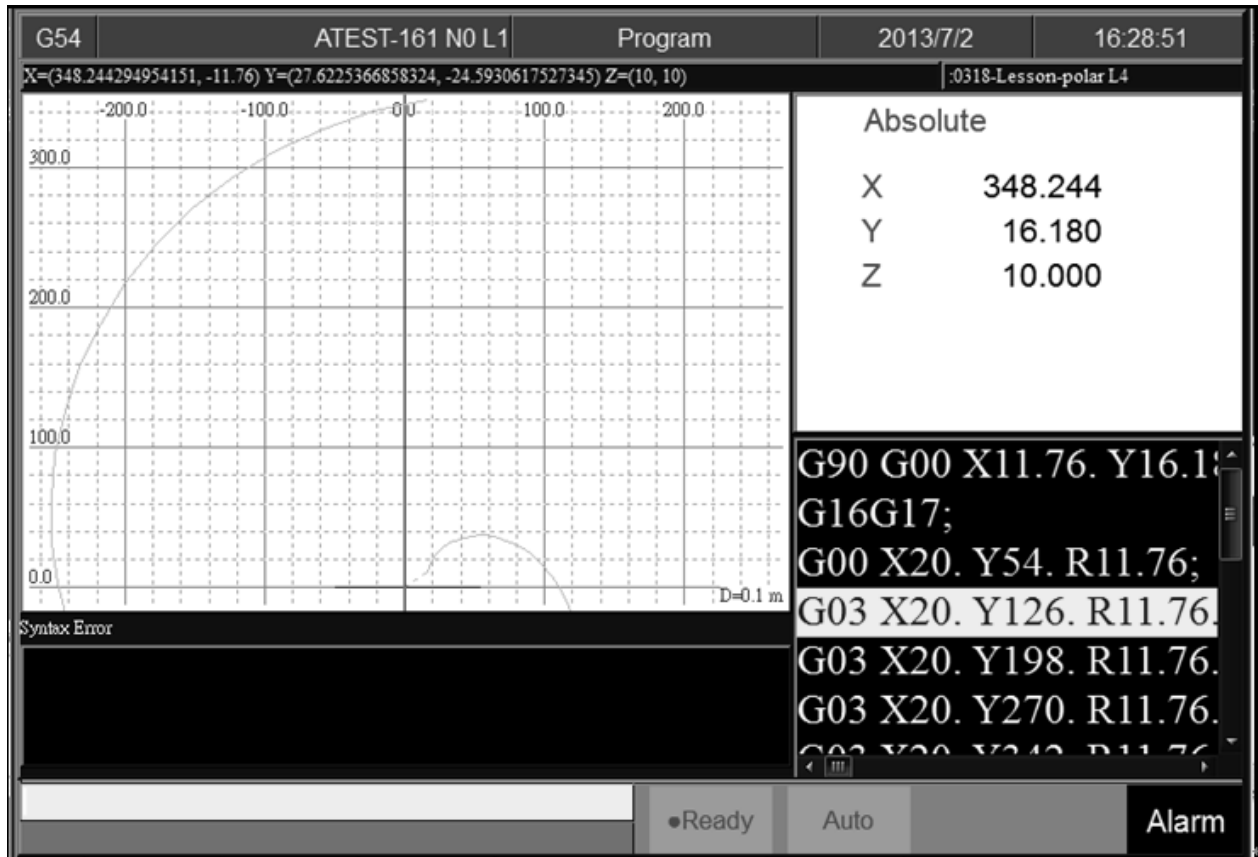
- Command
 - 5 Buttons type: F2 Program → Next → F3 Teach → F4 Cancel Arc Middle
 - 8 Buttons type: F2 Program → F6 Teach → F4 Cancel Arc Middle
- Function
 - Clear the arc middle that had been set.
 - If the arc middle are not being set, this function will not enable.

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.

SYNTEC

3.5.7 Simulation



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

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

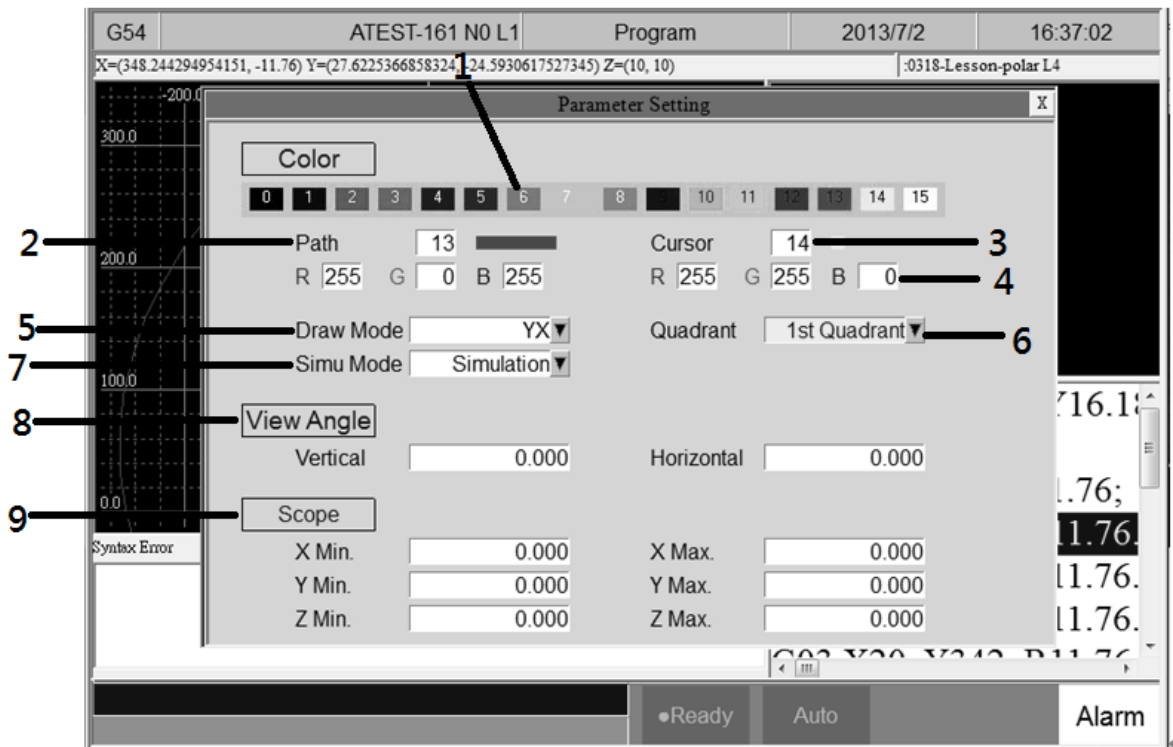
- Command
 - 5 Buttons type: F2 Program → F5 Simulation → F3 Zoom
 - 8 Buttons type: F2 Program → F7 Simulation → F3 Zoom
- Function
 - Zoom in/out the simulation window.
- Operation Method
 - Press F3「Zoom」,there will be a block show up. Use【↑】【↓】【←】【→】can move the window up, down, left and right.
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

SYNTEC

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

3.5.8 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	
kfp-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

3.6 Offset/Setting (MainFenu: F3)

- Path
 - 5 key, 8 Key : F3 Offset/Setting
 - 10+8 key : G3 Offset/Setting
- Explanation
 - Offset/Setting function setup are under this fenubar group
 - Able to switch between pages in the group with hot key【Offset/Setting】

3.6.1 Workpiece Cord.

G54	N0 L1		Offset/Setting		2013/8/22	17:12:48
External Shift		G54P1(G54)		G54P2(G55)		Machine
X	0.000	X	0.000	X	0.000	X 0.000
Y	0.000	Y	0.000	Y	0.000	Y 0.000
Z	0.000	Z	0.000	Z	0.000	Z 0.000
A	0.000	A	0.000	A	0.000	A 4.158
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

- Path
 - 5 key, 8key : F3 Offset/Setting → F1 Workpiece Coord.
 - 10+8 key : G3 Offset/Setting → F1 Workpiece Coord.
- Explanation
 - Switch to "Workpiece Coord." screen with this fenubar
 - System default will be G54 if there are no specific declaration with G54~G59.10 in the NC program.
 - External Coordinate Shift : Effective in all workpiece coordinates (G54~G59.10)
- Operation Method:
 - Move the cursor with[↑][↓][←][→].
 - Use[PageUp][PageDown]to switch the pages.
- Note
 - After setting the workpiece coordinate, please set the tool length compensation again.
- Modification Timing
 - External Shift :
 - i. Able to modify : Block stop(ex:M00/M01), ready or not ready not caused by C40
 - ii. Not able to modify : Block stop or feedhold caused by C40 during machining or pre-interpretation
 - Workpiece Coordinate (G54P1(G54), G54P2(G55), G54P100) :
 - i. Able to modify :
 1. Block stop, ready or not ready not caused by C40;
 2. Block stop or feedhold caused by C40, but not modifying the using machining coordinate.
 - ii. Not able to modify :

1. During machining or pre-interpretation;
2. Block stop or feedhold caused by C40 and modifying the using workpiece coordinate

Mach. Coord. Teach

- Path
 - 5 key, 8key : F3 Offset/Setting → F1 Workpiece Coord. → F1 Mach. Coord. Teach
 - 10+8 key : G3 Offset/Setting → F1 Workpiece Coord. → Mach. Coord. Teach
- Explanation
 - Set current cursor located workpiece coordinate value into the corresponding machine coordinate value
- Operation Method
 - a. Move machine to the target location.
 - b. Move cursor to the modifying workpiece coordinate
 - c. Press "Mach. Coord. Teach"
 - d. Original value of workpiece coordinate will then be replaced by the corresponding machine coordinate value
- Example
 - a. Current machine coordinate of X axis is 5.000
 - b. Current G54 X axis value is 0.000
 - c. Move cursor to the G54 X axis coordinate
 - d. Press "Mach. Coord. Teach"
 - e. The G54 X axis value will then become 5.000

Rel. Coord. Teach

- Path
 - 5 key, 8 key : F3 Offset/Setting → F1 Workpiece Coord. → F2 Rel. Coord. Teach
 - 10+8 key : G3 Offset/Setting → Workpiece Coord. → F2 Rel. Coord. Teach
- Explanation
 - Set current cursor located workpiece coordinate value into the corresponding relative coordinate value
- Operation Method
 - a. Move machine to the target location.
 - b. Move cursor to the modifying workpiece coordinate
 - c. Press "Rel. Coord. Teach"
 - d. Original value of workpiece coordinate will then be replaced by the corresponding relative coordinate value
- Example
 - a. Current relative coordinate of X axis is 5.000
 - b. Current G54 X axis value is 0.000
 - c. Move cursor to the G54 X axis coordinate
 - d. Press "Rel. Coord. Teach"
 - e. The G54 X axis value will then become 5.000

Aux. Coord. Teach

- Path

- 5 key, 8 key : F3 Offset/Setting → F1 Workpiece Coord. → F3 Aux. Coord. Teach
- 10+8 key : G3 Offset/Setting → F1 Workpiece Coord. → F3 Aux. Coord. Teach
- Explanation
 - Set the current cursor located workpiece coordinate value into the corresponding auxiliary coordinate value.
 - The Aux. coordinate value only appears after applying middle function.
- Operation Method
 - a. Apply the middle function to calculate the aux. coordinate value.
 - b. Move cursor to the modifying workpiece coordinate
 - c. Press 「Aux. Coord. Teach」
 - d. Original value of workpiece coordinate will then be replaced by the corresponding Aux. coordinate value.
- Example
 - a. Current Aux. coordinate of X axis is 5.000
 - b. Current G54 X axis value is 0.000
 - c. Move cursor to the G54 X axis coordinate
 - d. Press "Aux. Coord. Teach"
 - e. The G54 X axis value will then become 5.000

Inc. Input

- Path
 - 5 key, 8 key : F3 Offset/Setting → F1 Workpiece Coord. → F4 Inc. Input
 - 10+8 key : G3 Offset/Setting → F1 Workpiece Coord. → Inc. Input
- Explanation
 - Change the cursor located workpiece coordinate value into (cursor located coordinate value + keyed in teaching value)
- Operation Method
 - a. Move cursor to the modifying workpiece coordinate.
 - b. Input the teaching value.
 - c. Press "Inc. Input"
 - d. The workpiece coordinate value will then become the value of cursor located coordinate value + keyed in teaching value.
- Example
 - a. Current G54 X axis value is 0.000
 - b. Move cursor to the G54 X axis coordinate
 - c. Input 10.000
 - d. Press "Inc. Input"
 - e. The G54 X axis value will then become 15.000

Mach. Coord. Inc. Teach

- Path
 - 5 key : F3 Offset/Setting → F1 Workpiece Coord. → Next → F1 Mach. Coord. Inc. Teach
 - 8 key : F3 Offset/Setting → F1 Workpiece Coord. → F6 Mach. Coord. Inc. Teach
 - 10+8 key : G3 Offset/Setting → F1 Workpiece Coord. → F6 Mach. Coord. Inc. Teach
- Explanation
 - Change the current cursor located workpiece coordinate value into (corresponding machine coordinate value + keyed in teaching value)

- Operation Method
 - a. Move cursor to the modifying workpiece coordinate
 - b. Input the teaching value
 - c. Press "Mach. Inc. Teach"
 - d. The workpiece coordinate value will then become the value of its corresponding machine coordinate value + keyed in teaching value.
- Example
 - a. Current machine coordinate of X axis is 0.000
 - b. Move cursor to the G54 X axis coordinate
 - c. Input 10.000
 - d. Press "Mach. Inc. Input"
 - e. The G54 X axis value will then become 15.000

Middle Function

- Path
 - 5 key, 8key : F3 Offset/Setting → F1 Workpiece Coord. → F5 Middle Func.
 - 10+8 key : F3 Offset/Setting → Workpiece Coord. → Middle Func.
- Explanation
 - This function helps to calibrate the middle point of workpiece before machining.
 - The value calculated by middle function will be input to G54~G59 workpiece coordinates for coordinate setups.
 - Middle function provides 2 modes, manual mode and auto mode.

Manual Middle Function

- Explanation
 - Set middle function to mode 0
 - User controls the machine with MPG and moves the machine to the end point of workpiece in X, Y axis direction respectively to record the machine location. The system will automatically calculate the X, Y axis value of workpiece center point.
- Operation method
 - a. Move the tool with MPG, reach Px1 point in the figure and press "Px1 Set". The current location will be set as the machine coordinate of Px1 and be recorded on the screen. The machine coordinate of workpiece middle point in X axis direction will be calculated using Px1 and Px2, the result will be shown in the Pxm column and Aux. Coord. X column.
 - b. Move the tool with MPG, reach Px2 point in the figure and press "Px2 Set". The current location will be set as the machine coordinate of Px2 and be recorded on the screen. The machine coordinate of workpiece middle point in X axis direction will be calculated using Px1 and Px2, the result will be shown in the Pxm column and Aux. Coord. X column.
 - c. Move the tool with MPG, reach Py1 point in the figure and press "Py1 Set". The current location will be set as the machine coordinate of Py1 and be recorded on the screen. The machine coordinate of workpiece middle point in Y axis direction will be calculated using Py1 and Py2, the result will be shown in the Pym column and Aux. Coord. Y column.
 - d. Move the tool with MPG, reach Py2 point in the figure and press "Py2 Set". The current location will be set as the machine coordinate of Py2 and be recorded on the screen. The machine coordinate of workpiece middle point in Y axis direction will be calculated using Py1 and Py2, the result will be shown in the Pym column and Aux. Coord. Y column.
 - e. The value of Pxm and Pym is the coordinate of workpiece middle point.

- f. For controller software versions before 10.116.24x(included), first switch to workpiece coordinate setting screen, then move cursor to the required workpiece coordinate system and press F3 "Aux. Coord. Teach", the system will input the Aux. Coord. of the cursor located axis to the column.
- g. For controller software versions above 10.116.34(included), first move cursor to "Workpiece Coord. No. P" on middle function screen, then input the required workpiece coordinate number and press F5 "Aux. Coord. Auto Setting", the system will input the Aux. Coord. in X, Y axis direction to the corresponding workpiece coordinate.

Auto Middle Function

- Explanation
 - Set middle function to mode 1
 - Different from the manual middle function, user only needs to enter the workpiece dimension and set the boundary coordinate. After leading the tool to the starting point and press start, the system will search for the workpiece middle point automatically.
- Parameter description
 - Length I: Workpiece length in X axis direction
 - Width J: Workpiece width in Y axis direction
 - Safety Distance H: When the middle function starts, the initial position will be located near the center of workpiece, while moving from the center to the outer edge of workpiece, descending to Z axis coordinate P2 and searching for the border, a safety distance is required to 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.

3.6.2 Tool Set

G54	N0 L1		Offset/Setting		2013/8/22	17:42:01
Input Mode(Absolute (I)Increment (Z)Measure					Machine	
Absolute						
Diameter(D)		Length(H)				
Geometry	Wear	Geometry	Wear			
1	0.000	0.000	0.000			
2	0.000	0.000	0.000			
3	0.000	0.000	0.000			
4	0.000	0.000	0.000			
5	0.000	0.000	0.000			
6	0.000	0.000	0.000			
7	0.000	0.000	0.000			
8	0.000	0.000	0.000			
					Dist. To Go	
					X 0.000	
					Y 0.000	
					Z 0.000	
					A 0.000	
					Relative	
					X 0.000	
					Y 0.000	
					Z 0.000	
					A 4.158	
					●Ready	
					Auto	
					Alarm	

- Path
 - 5 key, 8 key: F3 Offset/Setting → F2 Tool Set
 - 10+8 key: G3 Offset/Setting → F2 Tool Set
- Function
 - Press the button to switch to the compensation setting page.
 - Actual G41/G42 compensation value = Tool nose + Tool radius wearing compensation
 - Actual G43/G44 compensation value = Tool length compensation + Tool length wearing compensation
 - Provides 200 sets of tool compensation setups (only 96 sets for versions before 10.116.36D)
- Parameter Explanation
 - Diameter Geometry: G41/G42 Cutter radius Dn compensation (please input radius, not diameter)
 - Diameter Wear: Minor modification of cutter radius
 - Length Geometry: G43/G44 Tool length Hn compensation.
 - Length wear: Minor modification of cutter length
- Operation method
 - a. Move the cursor with arrow keys【↑】【↓】【←】【→】
 - b. Use【PageUp】【PageDown】to switch the page.
 - c. Key in method:
 - Normally applies absolute mode for tool nose or tool length compensation
 - Normally applies incremental mode for cutter radius wearing compensation or tool length wearing compensation
 - d. Absolute input:
 - Press "A" and press "Enter"
 - The value of where the cursor is located will be set as the input value.
 - e. Incremental input:
 - Press "I" and press "Enter".

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

Clear Z Coord.

- Path
 - 5 key, 8 key: F3 Offset/Setting → F2 Tool Set → F1 Clear Z Coord.
 - 10+8 key: G3 Offset/Setting → F2 Tool Set → F1 Clear Coord.
- Function
 - Clear the current Z axis relative coordinate value.

Tool No.

- Path
 - 5 key, 8 key: F3 Offset/Setting → F2 Tool Set → F7 Tool No.
 - 10+8 key: G3 Offset/Setting → F2 Tool Set → F7 Tool No,
- Function
 - Please refer to chapter 3.4.4 for details

Tool Life Manag.

- Path
 - F3 Offset/Setting → F2 Tool Set → F8 Tool Life Manag.
- Function
 - Please refer to chapter 3.4.5 for details

3.6.3 Tool Tip Measure

- Command
 - F3 Offset/Setting → F3 Tool Tip Measure
 - G3 Offset/Setting → F3 Tool Tip Measure
- Function
 - When the machine is equipped with tool presetting and measuring machine, users can move the machine table to the measuring machine location to measure tool tip through parameter setting.
 - Under different conditions, here are four methods
 - Single tool, single workpiece
 - Single tool, multi workpiece
 - multi tool, multi workpiece
 - 5-axis (need to turn on Option12 or Option13)
 - Refer to chapter 3.4.3

3.6.4 User Parameter Setting

- 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.

3.7 Monitor (MainFenu: F4)

G54	TEST N0 L1	Monitor	2013/7/2	20:05:46
Absolute	Dist. To Go	G Code G1	Run Time Accum	0 : 0 : 0
X -20.000	X 0.000	G18 G90 G95	Run Time	0 : 0
Z 100.000	Z 0.000	G71 G40 G49	G00 Over	100 %
Y 100.000	Y 0.000		G01 Over	100 %
			Spd Over	100 %
Feedrate	0.0mm/min	T 0000	TotalAcumPar	0
Spindle	1000RPM	Start Block No. 1	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.

Fenubar Introduction

- F1 : Edit
Explanation : Change the screen to "Program Editor" and load current NC file to edit.
- F2 : Simulation Switch
Explanation : Display graphic simulation components.

F3 : MDI Input

Explanation : Edit MDI program under MDI state.

- F4 : Param. Set

Function : Used to switch between "Information" and "Setting"

- Screen 1 : Line No. Col. No., Parameter Set, Part Count

工件数设定	
总工件数	0 工件数 0
需求工件数	0
加工参数设定	
进给速率	1000.000 mm/min
主轴转速	1000 RPM
G120.1 P0	断点行号 0
	断点序号 0

- Screen 2 : G code, Time, Override

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

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

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

(support more than 8 axes)

SYNTEC

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

(support more than 6 axes)

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

- F5 : Tool Wear Set
Function : Show the tool wear set screen
ARM machine : Tool Wear Set and Information are at the same fenubar layer, the tool wear set page can be opened by pressing the fenubar and closed by pressing again (switch back to the Information page)
- F6 : Start MPG Shift
Function : Enable the MPG Shift function
- F7 :
 - 5 key, 8 key : Work Record
Function : Show the work record
 - 10+8 key : Graph Adjust
Function :
- F8 :
 - 5 key, 8 key : Clear Accum. Cycle Time
 - Function : Clear the accumulated machining time to zero
 - 10+8 key : File Manager
Function :
- F9 :
 - 5 key, 8 key : N/A
Function : N/A
 - 10+8 key : Work Record
Function : Show the work record
- F10 :
 - 5 key, 8 key : N/A
Function : N/A
 - 10+8 key : Clear Accum. Cycle Time
 - Function : Clear the accumulated machining time to zero

3.7.1 Monitor Area of Machining Information

G54		TEST N0 L1		Monitor		2013/7/2		20:05:46	
Absolute		Dist. To Go		G Code		Run Time		0 : 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 Status Monitor

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

Program Monitor Block

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

Machining Information

- 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

3.7.2 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.

3.7.3 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.

3.7.4 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.

3.7.5 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"

3.7.6 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.

3.7.7 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.

3.7.8 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

3.7.9 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」.

3.8 Maintain (MainFenu: F5)

- Path
 - 5 key, 8 key: F5 Maintain
 - 10+8 key: Maintain
- Explanation
 - This function page displays system alarms, network settings, fast diagnosis, PLC parameter settings, and controller software information.

3.8.1 Alarm

- Path
 - 5 key, 8 key: F5 Maintain → F1 Alarm
 - 10+8 key: G5 Maintain → F1 Alarm
- Explanation
 - Display alarm messages on the screen.

Pending Alarm

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

History Alarm

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

Save Alarm

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

- Default name of the exported files:
 - Pending Alarm: Actalm.txt
 - History Alarm: Histalm.txt

3.8.2 Set Kernel Server

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
 - 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
 - Subnet Mask
 - Enter the subnet mask of the IP address
 - Should be the same as the setting at PC end
 - PC Name
 - Name of the connecting PC
 - Should be the same as the setting at PC end
 - Dir Name
 - Name of the shared folder at PC end (should be the same as the setting at PC end).
 - 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.

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

- Path
 - 55 key, 8 key: F5 Maintain → F3 Fast Diagnosis → F1 System Data
 - 10+8 key: G5 Maintain → F3 Fast Diagnosis → F1 System Data
- Explanation
 - Display basic system and spindle information for diagnosis.

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

3.8.4 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	0	1	0	1	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

3.8.5 System Setting

[illegible]

- Path
 - 5 key, 8 key: F5 Maintain → F5 Sys. Setting
 - 10+8 key: G5 Maintain → F5 Sys. Setting
- Explanation
 - This page is used for system environment setting.
- Operation Method
 - Move the cursor with[↑][↓][←][→]
 - Switch pages with[PageUp][PageDown]

Input/Display Unit

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

System Date/Time Setting

- Explanation
 - Date format: YYYY/MM/DD
 - YYYY: year
 - MM: month
 - DD: day
 - Time format: HH/MM/SS
 - HH: hour
 - MM: minute
 - SS: second

Set MMIconfig

- Path
 - 5 key, 8 key: F5 Maintain → F5 Sys. Setting → F8 Set MMIconfig
 - 10+8 key: G5 Maintain → F5 Sys. Setting → F8 Set MMIconfig
- Explanation
 - This page is used for other system environment setting. (9 pages in total)
- Operation Method
 - Move the cursor with【↑】【↓】【←】【→】
 - Switch pages with【PageUp】【PageDown】

Set Default

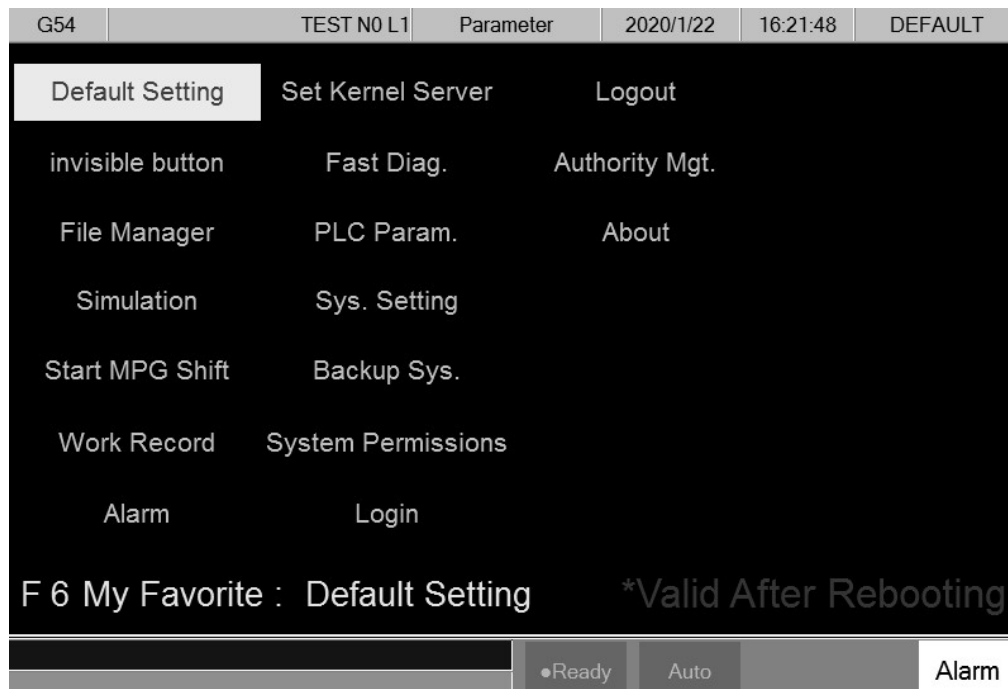
- Path
 - 5 key, 8 key: F5 Maintain → F5 Sys. Setting → F8 Set MMIconfig → F4 Set Default
 - 10+8 key: G5 Maintain → F5 Sys. Setting → F8 Set MMIconfig → F4 Set Default
- Explanation
 - Reset the value of where the cursor is located to default
- Attention
 - The setting only takes effect after reboot

Set Default All

- Path
 - 5 key, 8 key: F5 Maintain → F5 Sys. Setting → F8 Set MMIconfig → F5 Set Default All
 - 10+8 key: G5 Maintain → F5 Sys. Setting → F8 Set MMIconfig → F5 Set Default All
- Explanation
 - Reset all parameters to default.
- Note
 - The setting only takes effect after reboot

SYNTEC

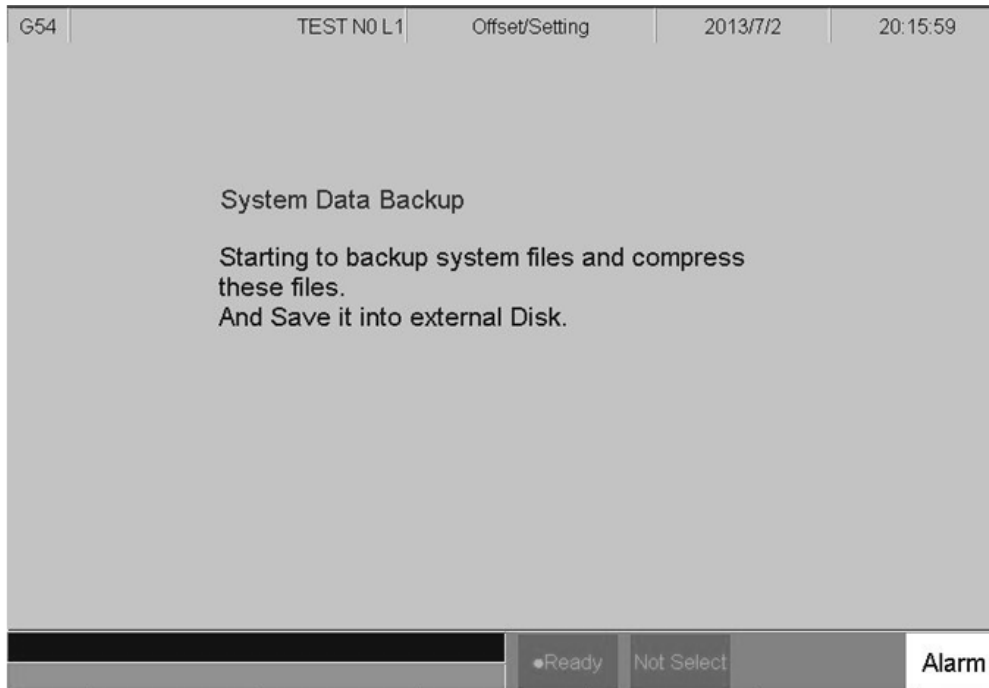
My Favor



- 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

3.8.6 Backup System



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

Online Backup

- Path
 - 5 key: F5 Maintain → Next → F1 Backup Sys. → F5 Online Backup
 - 8 key: F5 Maintain → F6 Backup Sys. → F8 Online Backup
 - 10+8 key: G5 Maintain → F6 Backup Sys. → F10 Online Backup
- Explanation
 - Upload the system backup data to the cloud for related technicians to help diagnose the problem.

3.8.7 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

3.8.8 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.

3.8.9 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.

3.8.10 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.

3.8.11 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.

3.8.12 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

3.8.13 About

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

3.8.14 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 → Pending Alarm
- Explanation
 - Display current system alarm.

History Alarm

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

Save Alarm

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

3.8.15 Network Setting

G54	TEST N0 L1	Set Kernel Server	2020/1/22	11:42:20	DEFAULT
IP Address Parameter 1 / 1					
IP Address Setting		Specify an IP Address ▼			
IP Address		Name Server Parameter			
Subnet Mask		Primary DNS			
Default Gateway		Primary WINS			
Network DiskRemote Host Path					
PC Name		Dir Name			
User Name		Password			
Net Status	Code : -1				
Resource Shared					
Shared Folder Path					
		●Ready	Auto	Alarm	

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

- Explanation
 - Set up the system network
- Parameters
 - a. IP Address Setting
 - Select "Specify an IP Address" when using a crossover cable
 - Select "Obtain an IP Address via DHCP" when using a normal cable, "IP Address" and "Subnet Mask" can then be skipped
 - b. IP Address
 - Enter the applicable IP address in the domain
 - c. Subnet Mask
 - Enter the subnet mask of the IP address
 - Should be the same as the setting at PC end
 - d. PC Name
 - Name of the connecting PC
 - Should be the same as the setting at PC end
 - e. Dir Name
 - Name of the shared folder at PC end (should be the same as the setting at PC end).
 - f. User Name
 - Can be skipped if no account and password is set to protect the folder shared by the Internet disk; if do, set the corresponding account and password.
 - g. Password
 - Same situation as "User Name"

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.

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

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

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

Axes Data

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

3.8.17 PLC Param Setting

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
 - 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

3.8.18 System Setting

[illegible]

- Path
 - 5 key, 8 key: F5 Maintain → F5 Sys. Setting
 - 10+8 key: G5 Maintain → F5 Sys. Setting
- Explanation
 - This page is used for system environment setting.
- Operation Method
 - Move the cursor with【↑】【↓】【←】【→】
 - Switch pages with【PageUp】【PageDown】

Input/Display Unit

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

System Date/Time Setting

- Explanation
 - Date format: YYYY/MM/DD
 - YYYY: year
 - MM: month
 - DD: day
 - Time format: HH/MM/SS
 - HH: hour
 - MM: minute
 - SS: second

Set MMIconfig

- Path
 - 5 key, 8 key: F5 Maintain → F5 Sys. Setting → F8 Set MMIconfig
 - 10+8 key: G5 Maintain → F5 Sys. Setting → F8 Set MMIconfig
- Explanation
 - This page is used for other system environment setting. (9 pages in total)
- Operation Method
 - Move the cursor with【↑】【↓】【←】【→】
 - Switch pages with【PageUp】【PageDown】

Set Default

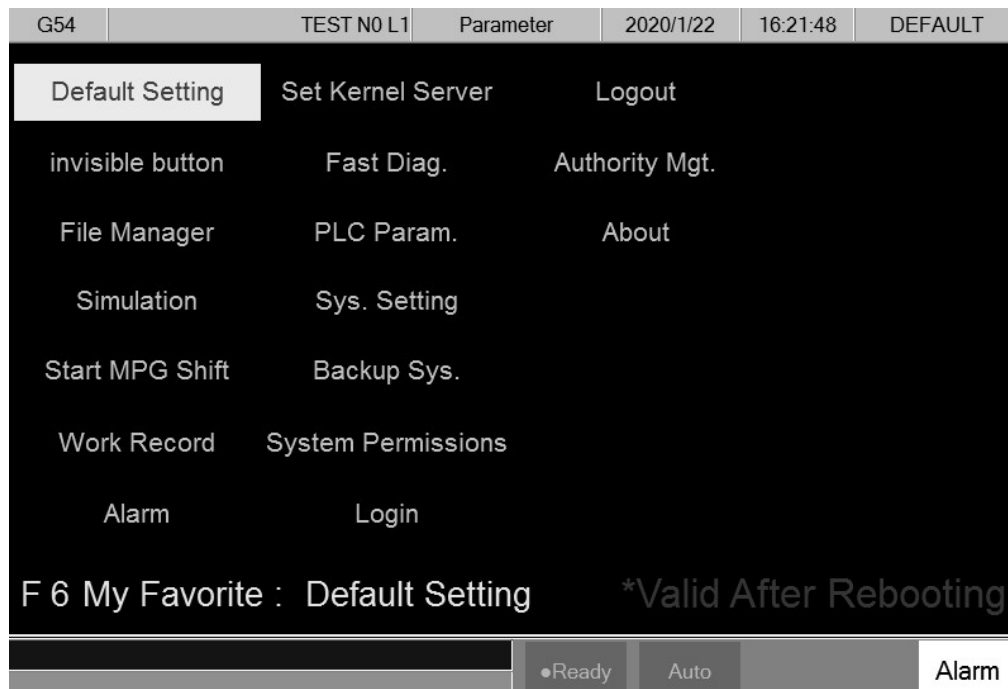
- Path
 - 5 key, 8 key: F5 Maintain → F5 Sys. Setting → F8 Set MMIconfig → F4 Set Default
 - 10+8 key: G5 Maintain → F5 Sys. Setting → F8 Set MMIconfig → F4 Set Default
- Explanation
 - Reset the value of where the cursor is located to default
- Attention
 - The setting only takes effect after reboot

Set Default All

- Path
 - 5 key, 8 key: F5 Maintain → F5 Sys. Setting → F8 Set MMIconfig → F5 Set Default All
 - 10+8 key: G5 Maintain → F5 Sys. Setting → F8 Set MMIconfig → F5 Set Default All
- Explanation
 - Reset all parameters to default.
- Note
 - The setting only takes effect after reboot

SYNTEC

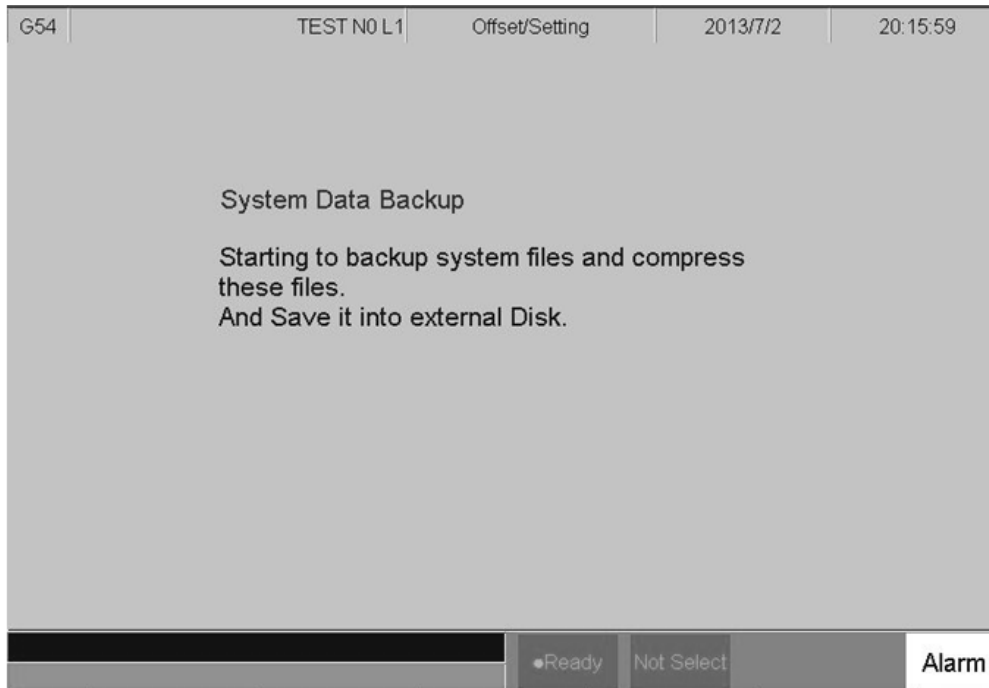
My Favor



- 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

3.8.19 Backup System



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

Online Backup

- Path
 - 5 key: F5 Maintain → Next → F1 Backup Sys. → F5 Online Backup
 - 8 key: F5 Maintain → F6 Backup Sys. → F8 Online Backup
 - 10+8 key: G5 Maintain → F6 Backup Sys. → F10 Online Backup
- Explanation
 - Upload the system backup data to the cloud for related technicians to help diagnose the problem.

3.8.20 About

- Command:
 - F8 Maintain -> F8 About
- Function
 - Provide controller software version

3.9 Parameter (MainFenu: F6)

Under the main menu, press F6 to enter this function as shown in the following figure.

[illegible]

Descriptions of sub functions are as below:

F1 : User param

- Function :
Drive the parameters of server system, for example: server parameter, mechanism parameter, range of program stroke, and max. feedrate speed..etc.
- Operation :
When system is in manual data input mode (MDI), user should press stop button before setting any parameter ("unready" shown on the state). Whenever the parameter is changed, system must be restart to effect the changes.

3.10 Diagnosis (MainFenu: F7)

This selection provides users with direct access to the memory area for parameter checking, parameter settings and NC diagnosis function. It can also be used to maintain and debug the control devices. Under the main menu, press F7 to access this function as shown in the following figure.

G54	TEST NO 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

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

G54	TEST NO 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

- 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		33		
2		18		34		34		
3		19		35		35		
4		20		36		36		
5		21		37		37		
6		22		38		38		
7		23		39		39		
8		24		40		40		
9		25		41		41		
10		26		42		42		
11		27		43		43		
12		28		44		44		
13		29		45		45		
14		30		46		46		
15		31		47		47		
16		32		48		48		
				●Ready	Auto			
							Alarm	

3.11 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

4 How to Operate 60 Series Controller

4.1 System Status

4.1.1 Not Ready

System cannot implement any operation under Not Ready state.

Conditions:

- Emergency stop button is pressed
- Serious alarm happens

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

4.1.2 Ready

System can implement all operations under Ready state.

Conditions:

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

4.1.3 Busy

System is executing NC program.

Conditions:

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

4.1.4 Feed Hold

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

Conditions

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

Note:

- Spindle will keep rotating under "Feedhold" state.

4.1.5 B-stop

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

Conditions:

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

Note:

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

4.2 Machine Preparation

4.2.1 Manual Function

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

JOG

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

Rapid JOG

- Function:
 - Control axis movement toward the selected direction with G00 speed
 - Can control more than one axis at the same time
- Condition:
 - System is on "Ready" state
 - JOG mode is selected
- Operation:
 - Press rapid key "~", then press axis direction key (X+,X-,Y+,Y-,Z+...) at the same time ,machine will move with G00- rapid speed
 - Hold the axis direction key to keep the axis moving uninterrupted
 - Rapid JOG speed can be adjusted by G00%
- Note:
 - Rapid JOG is usually much faster than JOG, so when operating, please confirm the axis position to ensure human and machine safety.

INC JOG (incremental JOG)

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

- Condition:
 - System is on "Ready" state
 - INC JOG mode is selected
- Operation:
 - Press axis direction key (X+, X-,Y+,Y-,Z+...) and machine moves toward the selected direction.
 - Cannot be constantly triggered like JOG or rapid JOG mode, INC JOG mode only moves once with a fixed distance when axis direction key is pressed once
 - The fixed distance in INC JOG mode can be selected by percentage movements as below:
 - X1 : Distance 0.001mm
 - X10 : Distance 0.010mm
 - X100 : Distance 0.100mm
 - These percentage movements are shared with MPG mode

MPG

- Function:
 - Control axis movement toward the selected direction
- Condition:
 - System is on "Ready" state
 - MPG mode is selected
- Operation:
 - Press axis direction key (X+, X-,Y+,Y-,Z+...) or turn the direction switch on the manual pulse generator for the direction to move.
 - Turn the dial on manual pulse generator clockwise / counterclockwise for forward / reverse direction.
 - The moving distance when turning one track in MPG mode can be selected by percentage movements as below:
 - X1 : Distance 0.001mm
 - X10 : Distance 0.010mm
 - X100 : Distance 0.100mm
 - These percentage movements are shared with INC JOG mode

4.2.2 Machining Process

AUTO

- 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

4.3 Workpiece Preparation

4.3.1 Workpiece Coord set

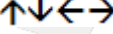
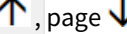
The setting of Workpiece Coordinate will be introduced on this section.

- Conditions
 - N/A
- Operation Procedure

SYNTEC

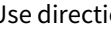
- a. Path: F3 Offset/Setting → F1 Workpiece Coord.

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
						●Ready
						Auto
						Alarm

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

4.3.2 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  move the cursor.
 - Use   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.

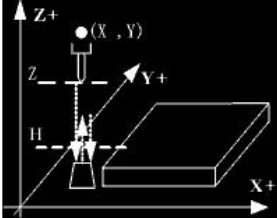
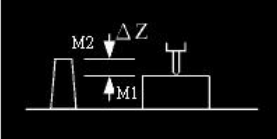
- Length compensate +length wear = real G43/G44 compensate.

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

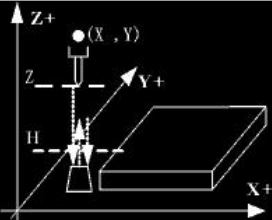
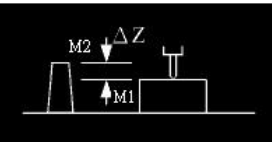
G54	N0 L1	Offset/Setting	2013/8/22	18:12:45
Auto Tool Function		WorkPiece No. P	0	Machine
AutoTool 1		Feedrate F	0.000	X 0.000
1:One Tool One Workpiece,2:On		Use Reference	0	Y 0.000
3:Many Tools Many Workpie		Ref Coord. X	0.000	Z 0.000
		Ref Coord. Y	0.000	A 4.158
		Start Coord. Z	0.000	
		Min. Z Mach. 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		Z 0.000
		to upper of measurement		A 4.158
		3:Press F1, Measure Start		
		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

- One Tool One Workpiece

G54		N0 L1	Offset/Setting	2013/8/22	18:12:45
Auto Tool Function AutoTool 1 1:One Tool One Workpiece,2:On 3:Many Tools Many Workpie			WorkPie No. P 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 			Delta Z Set 0.000 Do tool tip measure before do Delt 1:Take tool tip to top of good 2:Press F3, Delta Z Set		

- Set the Auto Tool number as 1(left-upper corner).
- Set the WorkPiece No. P as the workpiece coordinate.

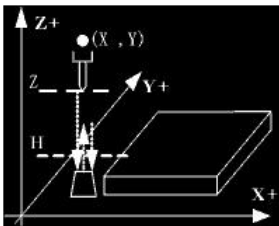
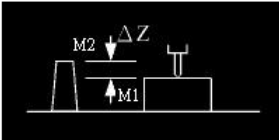
- Table X- Coordinate table

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

- Table X- Coordinate table

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

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

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.

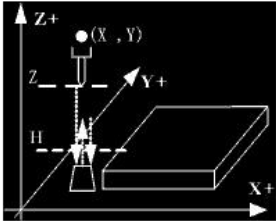
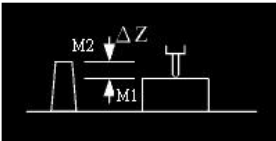
Workpiece coord No. P	Workpiece coord	Workpiece coord No. P	Workpiece coord
1	G54	2	G55

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 Auto Tool 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 WorkPoiece No. P 0 Do tool tip measure before do Delt 1:Take tool tip to top of good 2:Press F3, Delta Z Set	Machine X 0.000 Y 0.000 Z 0.000 A 4.158 Relative X 0.000 Y 0.000 Z 0.000 A 4.158 Aux. Coord X 0.000 Y 0.000 Z 0.000	
				
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.

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

G54	TEST N0 L1			Offset/Setting		2013/7/2	20:18:05
Tool Manager Function							
No	Turret group		Information	Cur. Life	Max. Life	Announce	Status
01	0	0	U N C -	0	0	0	No Managed
02	0	0	U N C -	0	0	0	No Managed
03	0	0	U N C -	0	0	0	No Managed
04	0	0	U N C -	0	0	0	No Managed
05	0	0	U N C -	0	0	0	No Managed
06	0	0	U N C -	0	0	0	No Managed
07	0	0	U N C -	0	0	0	No Managed
08	0	0	U N C -	0	0	0	No Managed
09	0	0	U N C -	0	0	0	No Managed
10	0	0	U N C -	0	0	0	No Managed
11	0	0	U N C -	0	0	0	No Managed
12	0	0	U N C -	0	0	0	No Managed

(0~96) Turret tool No.

●Ready

Not Select

Alarm

- Condition
 - Both auto and manual can be used.
- Operation
 - Pr.3228 is the on/off control of「Tool management」
- Description
 - Turret
 - Current tool case no. that tool located.
 - Group
 - Same kind of tool within in one group, if the first tool of that group is on lock state or「Tool life」is end, whenever user use T code to change the tool, system will skip the first tool and use the second one, when the second one is lock or「Tool life」is end, will use the third one, and so on.
 - Tool information (Status)
 - L—Lock / U—Unlock
 - If the status of tool is lock, that tool 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

4.4 Program Preparation and Execute Machining

4.4.1 Select Machining Program

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

4.4.2 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 or 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. F5(Simu. Setting): Set simulation parameter

4.4.3 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

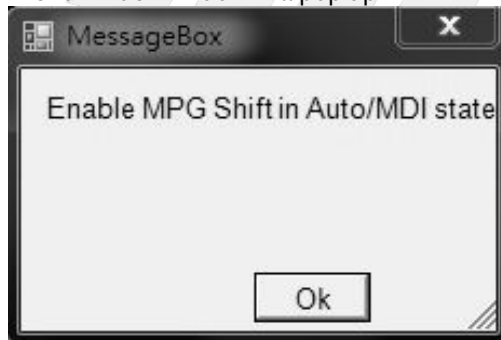
This chapter is going to introduce how to do machining test with single block

- 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.

4.4.4 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.

4.4.5 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.

4.4.6 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

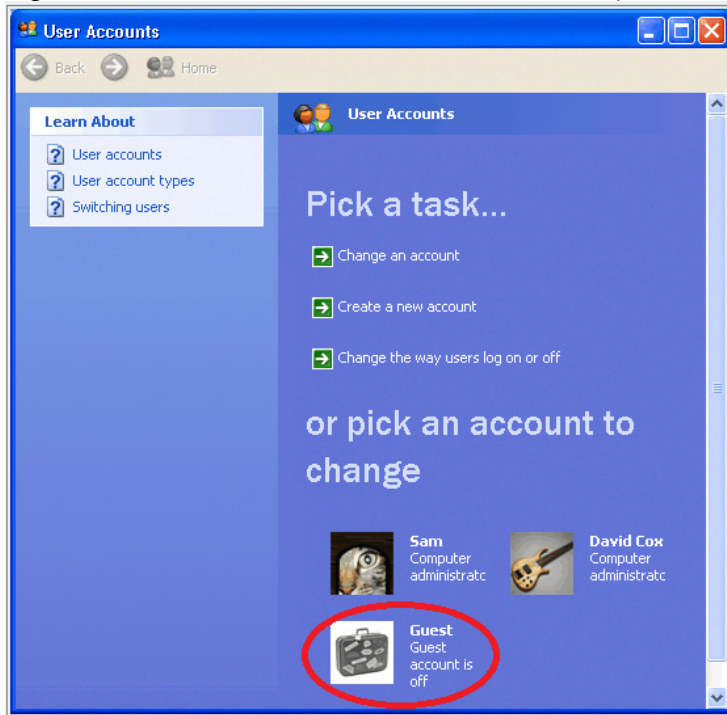
- Path
 - F5 Maintain → F2 Set Kernel Server → F5 Set Kernel Server → F1 Start Server
- Explanation
 - Start the kernel server immediately.

4.4.7 PC Setting

XP OS

1. Guest account setting

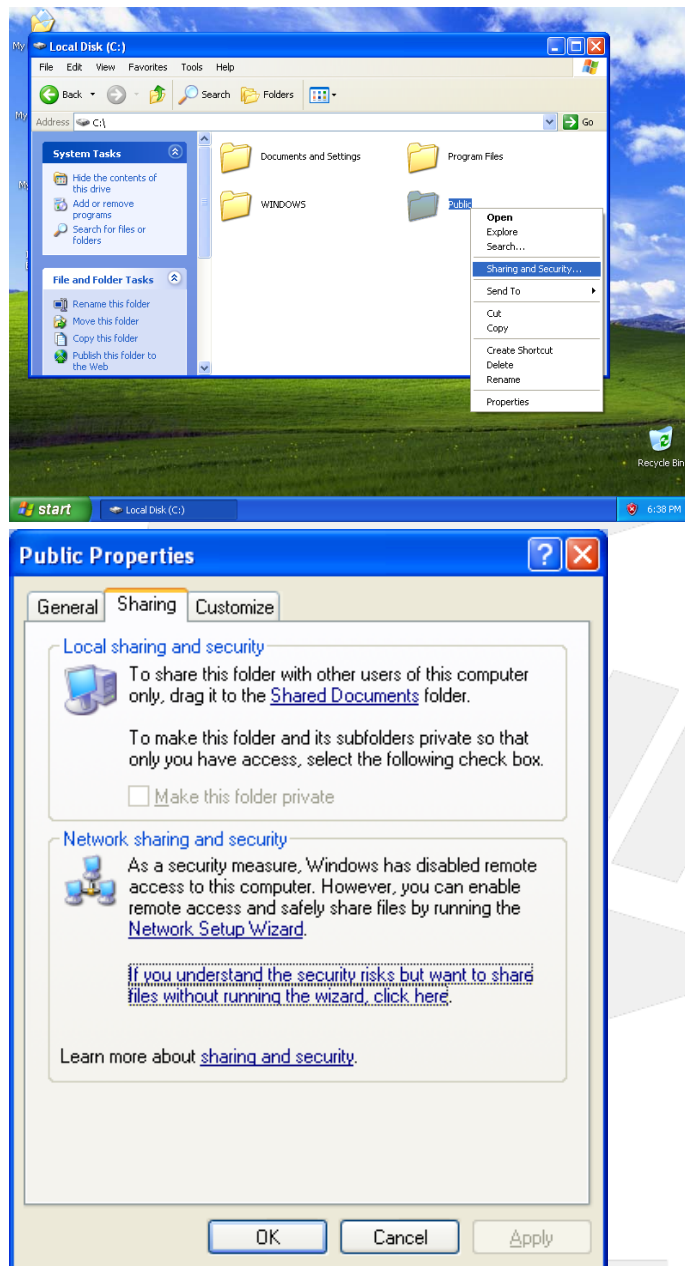
Log in as an Administrator and select "start"-> "control panel"->"user account"-> Guest



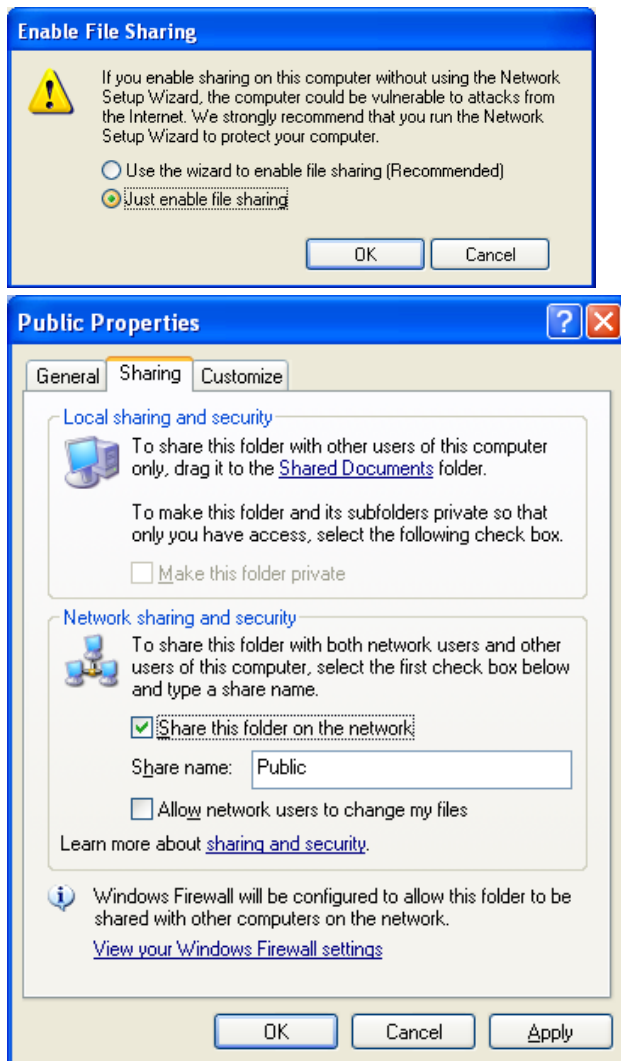
2. Sharing resource setting

- 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



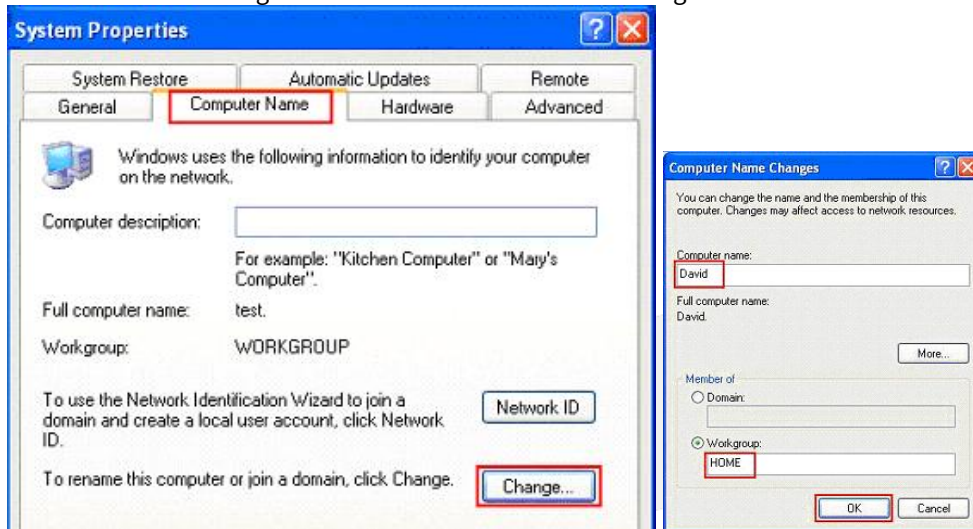
3. Click "OK" to confirm sharing setting; Select "Share this folder on the network", and "Allow network users to change my files".



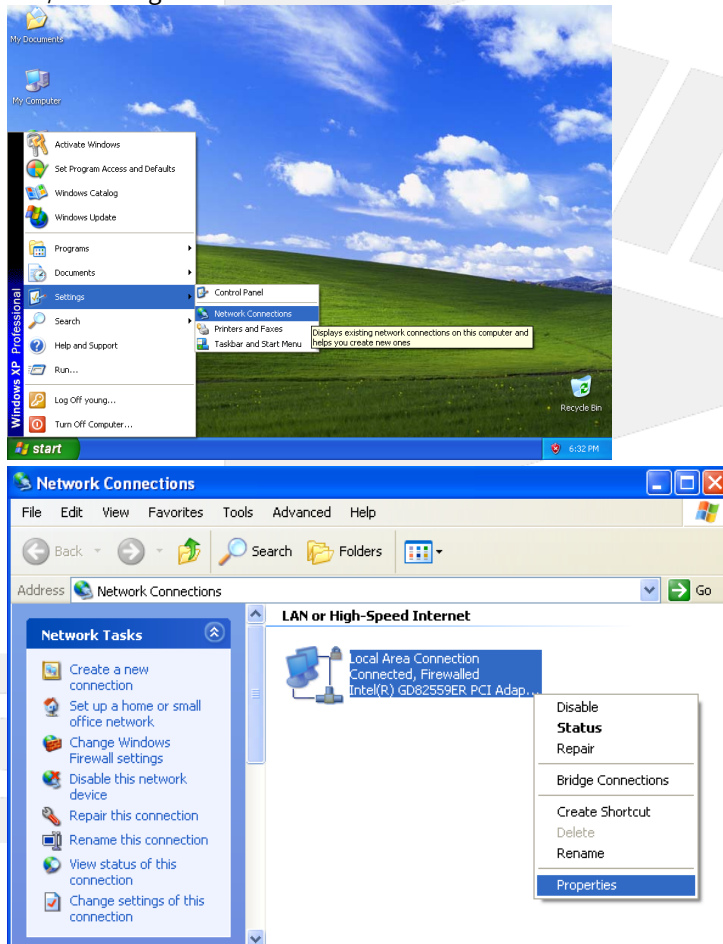
4. Setting PC name and workgroup
 "Start" -> "control panel" -> "System" -> "change" to set "Computer Name" and "Workgroup", and

SYNTEC

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



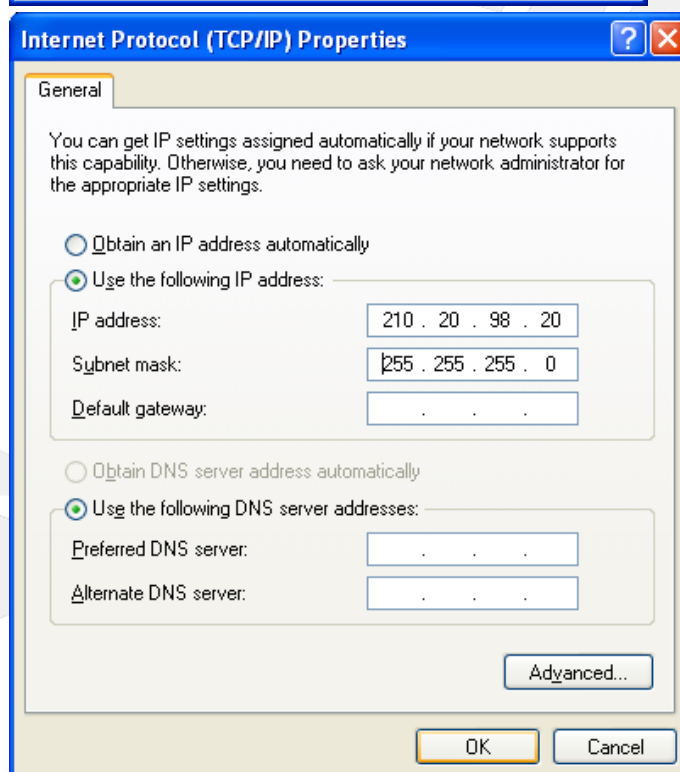
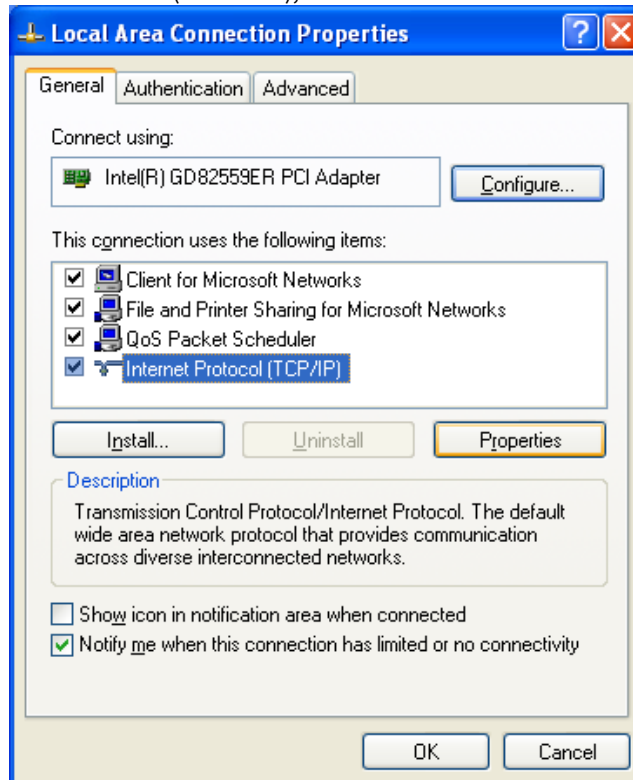
5. TCP/IP setting



"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)

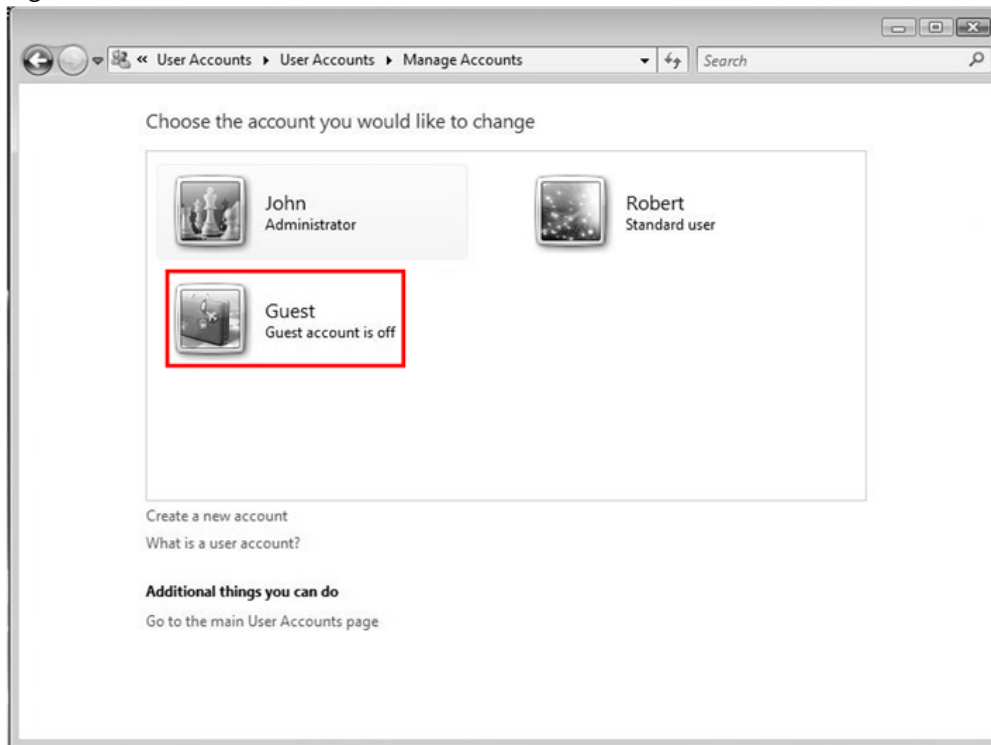
- Network cable (with HUB), select "Obtain an IP address automatically"



VISTA OS

1. Guest account setting

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



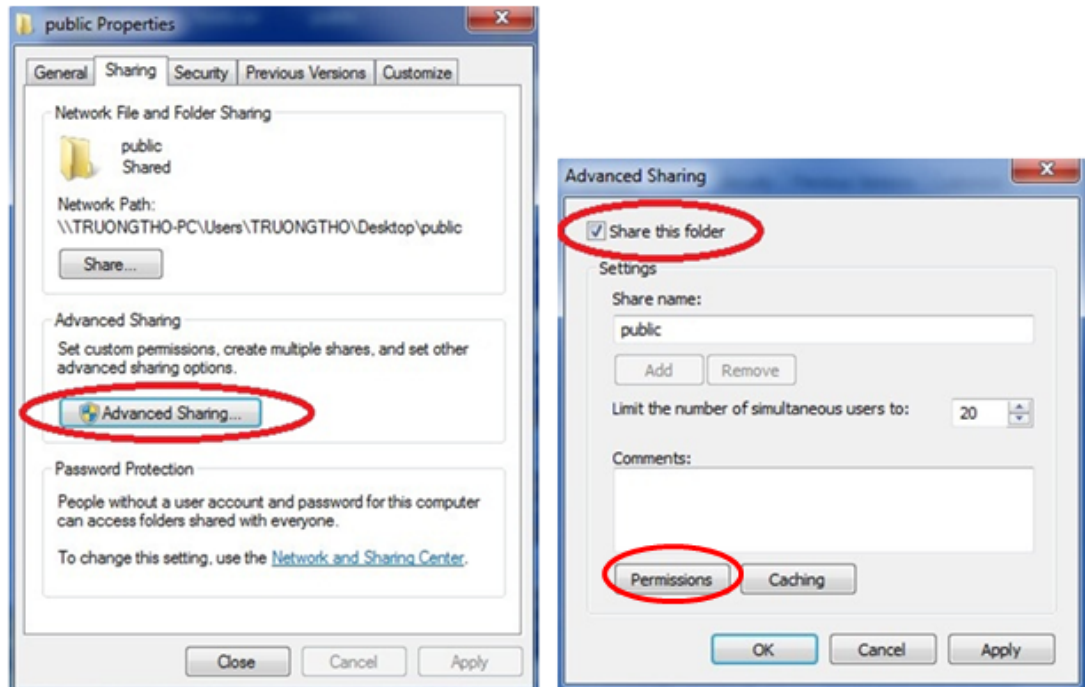
2. Sharing Resource Setting

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

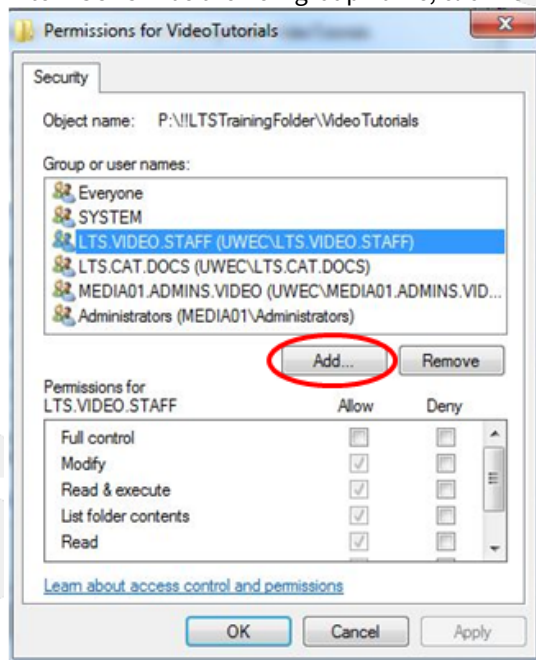
- a. Click on "advanced sharing"

SYNTEC

- b. Click on "share this folder"



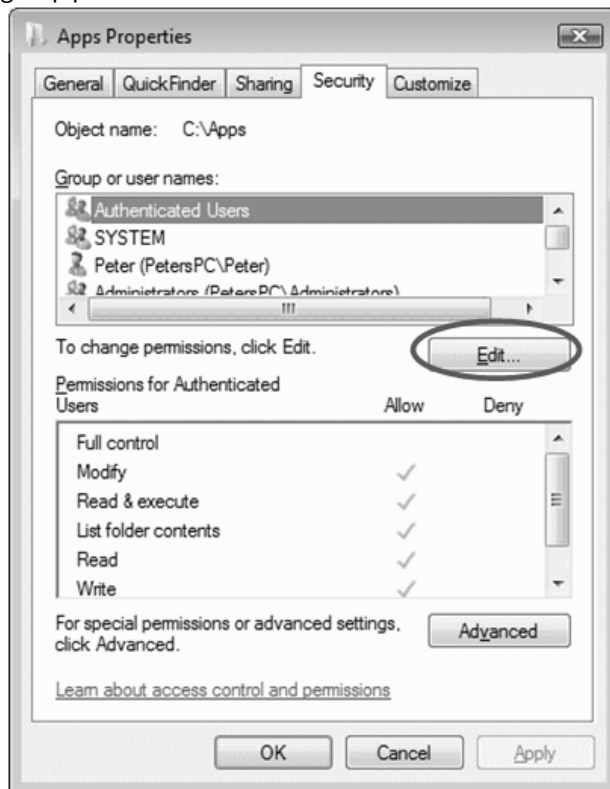
- c. Click on "permission"
 d. Click on "add"
 e. Enter "GUEST" as the new group name, click "OK" to complete setting



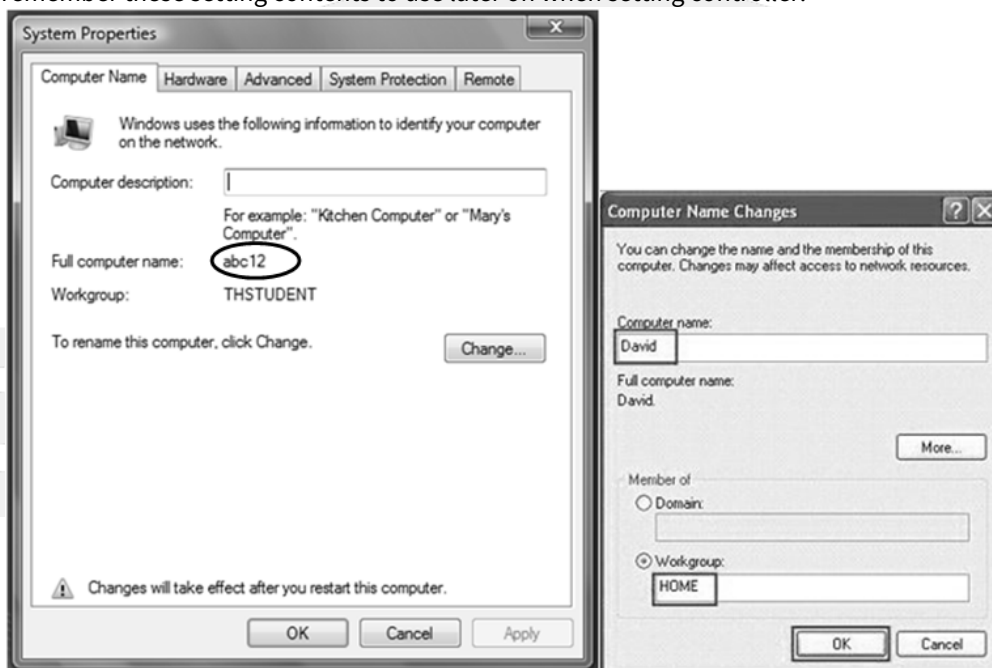
3. Security setting

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

group permissions to maximum.

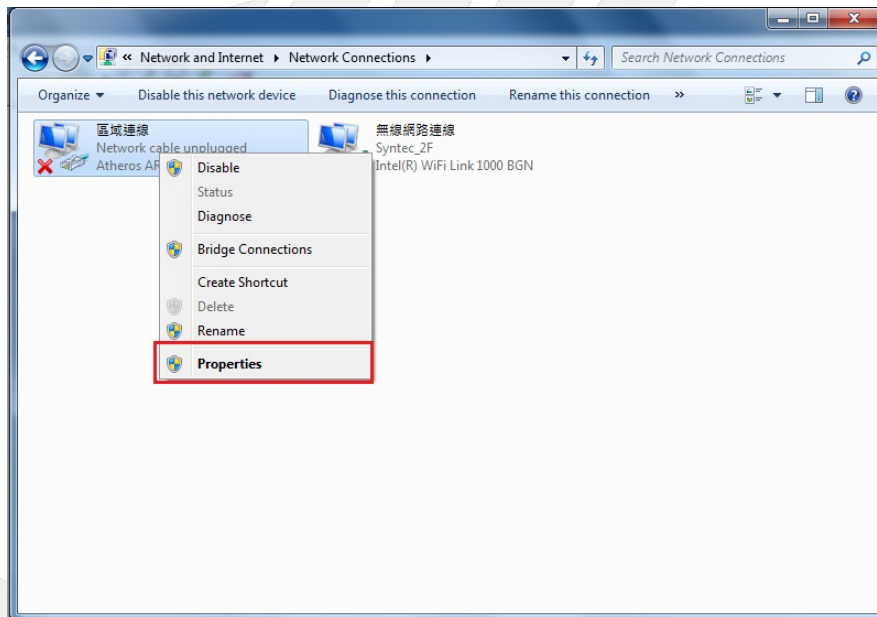
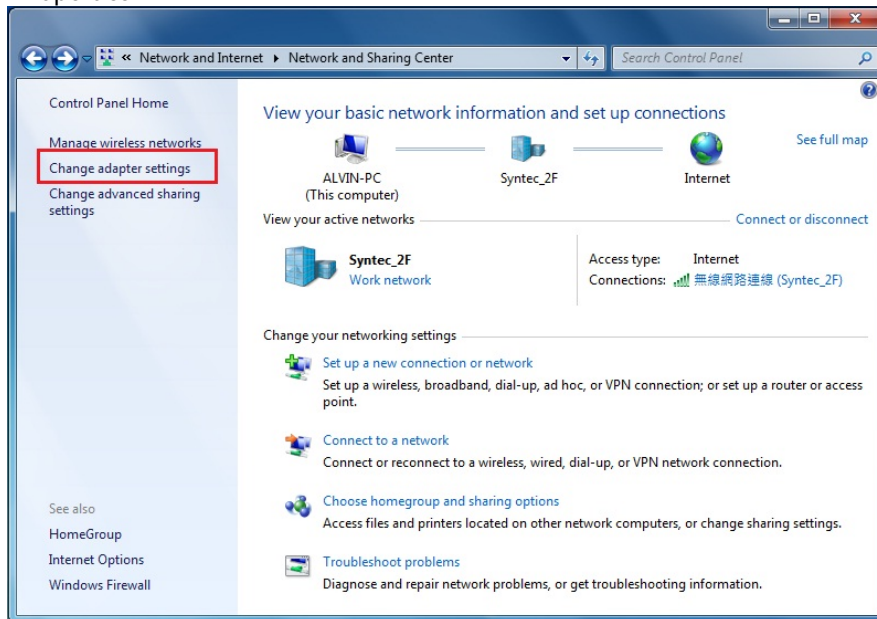


4. Setting PC name and workgroup
 "Start" -> "control panel" -> "System" -> "change" to set "Computer Name" and "Workgroup", and remember these setting contents to use later on when setting controller.



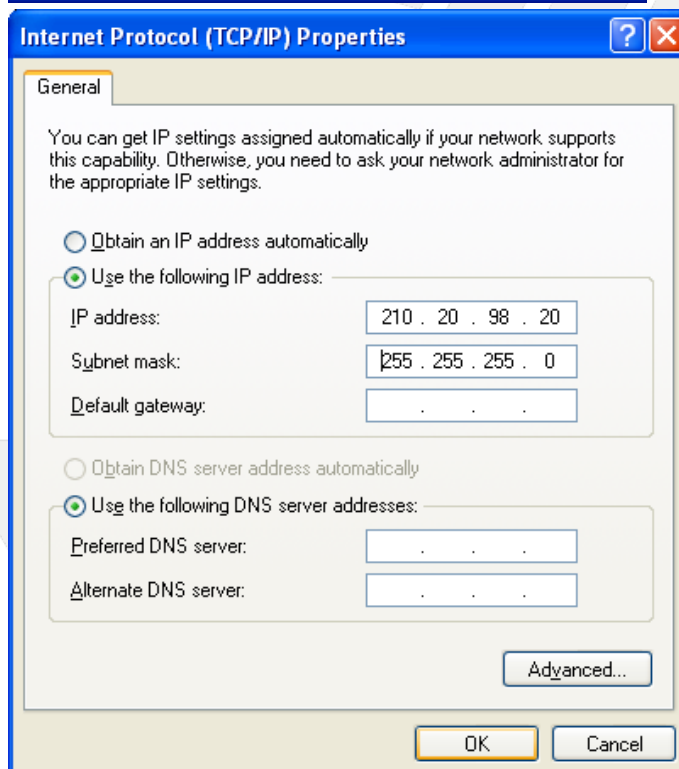
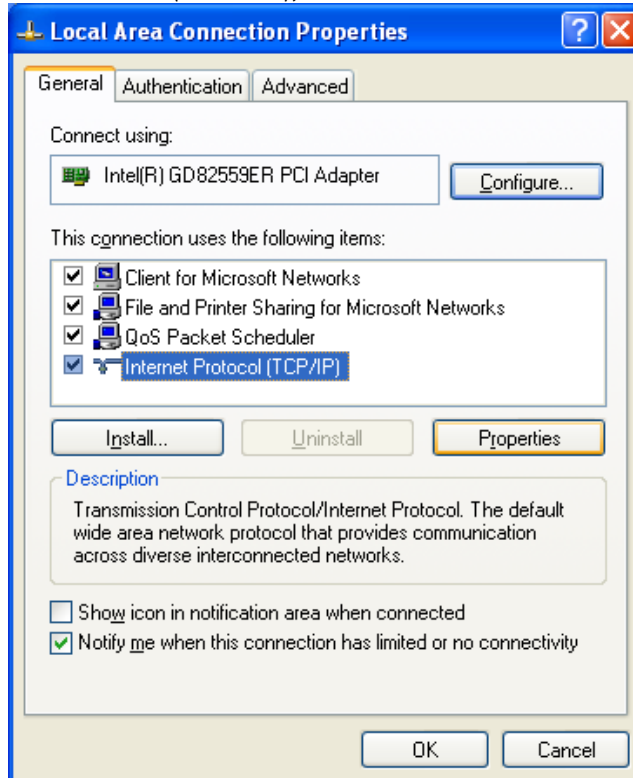
5. TCP/IP Setting

- 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)

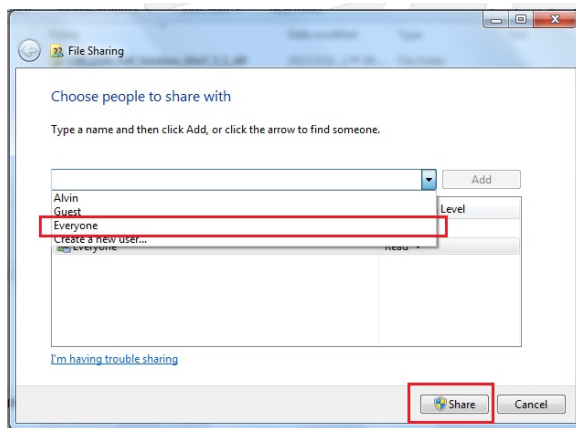
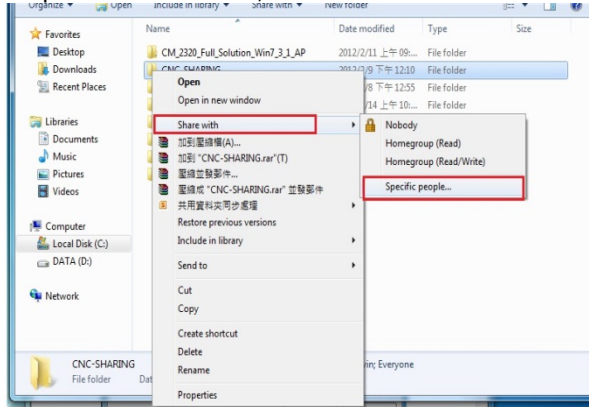
- Network cable (with HUB), select "Obtain an IP address automatically"



Win 7 OS

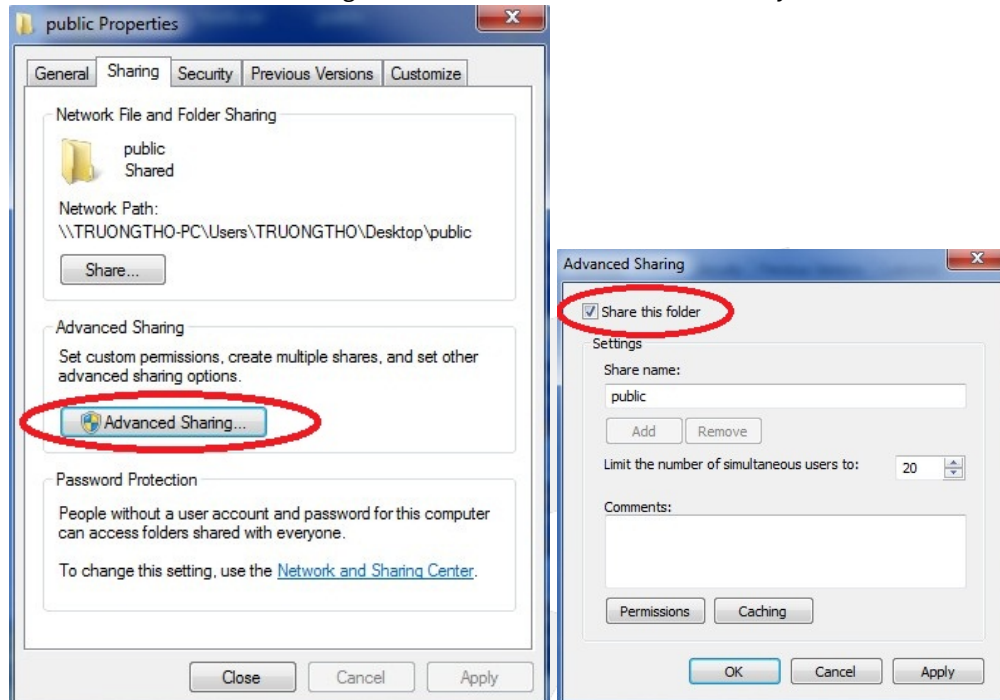
1. Sharing resource setting

- 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



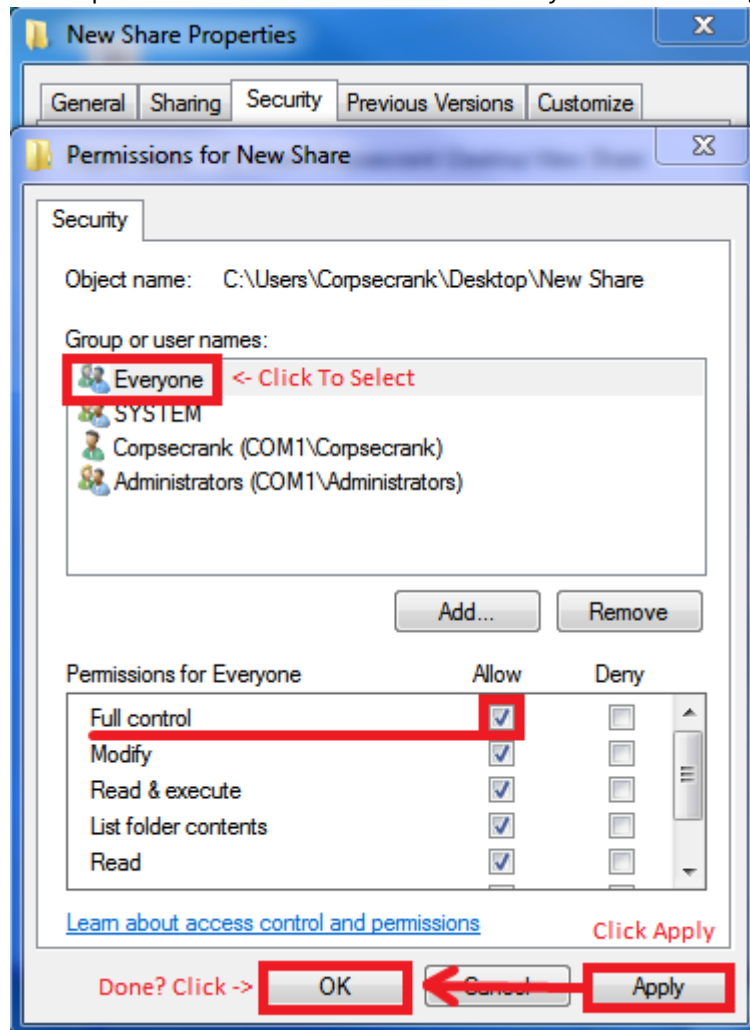
SYNTEC

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



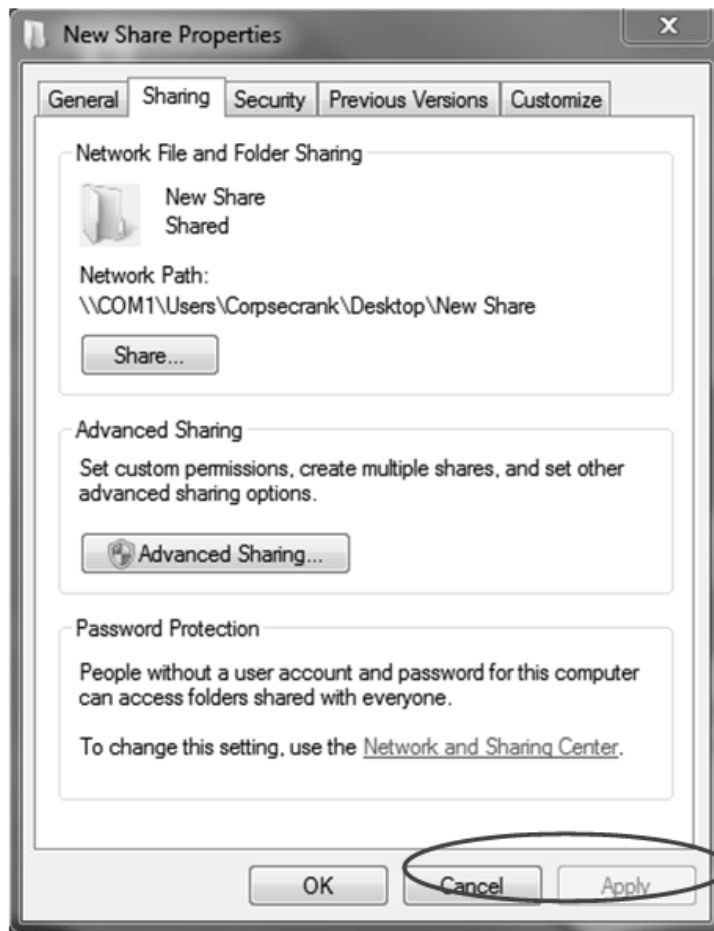
SYNTEC

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

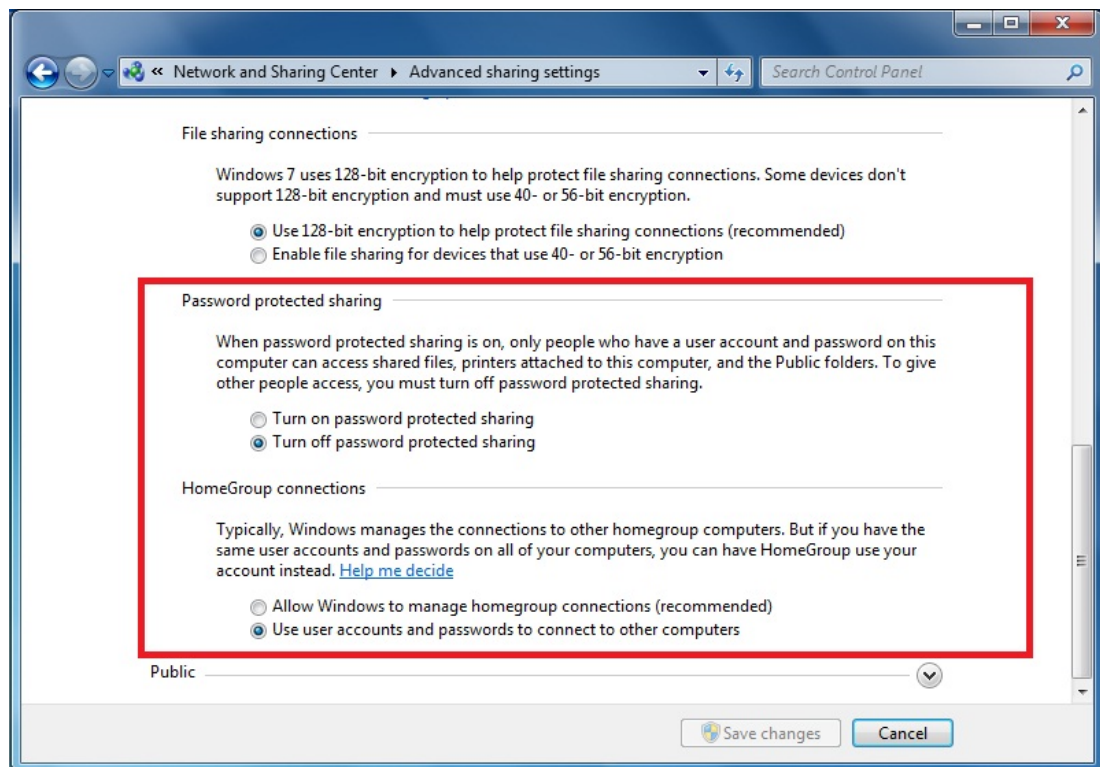


- Open "Network and sharing center", select "turn off password protected sharing" and "Open sharing....."

SYNTEC



SYNTEC

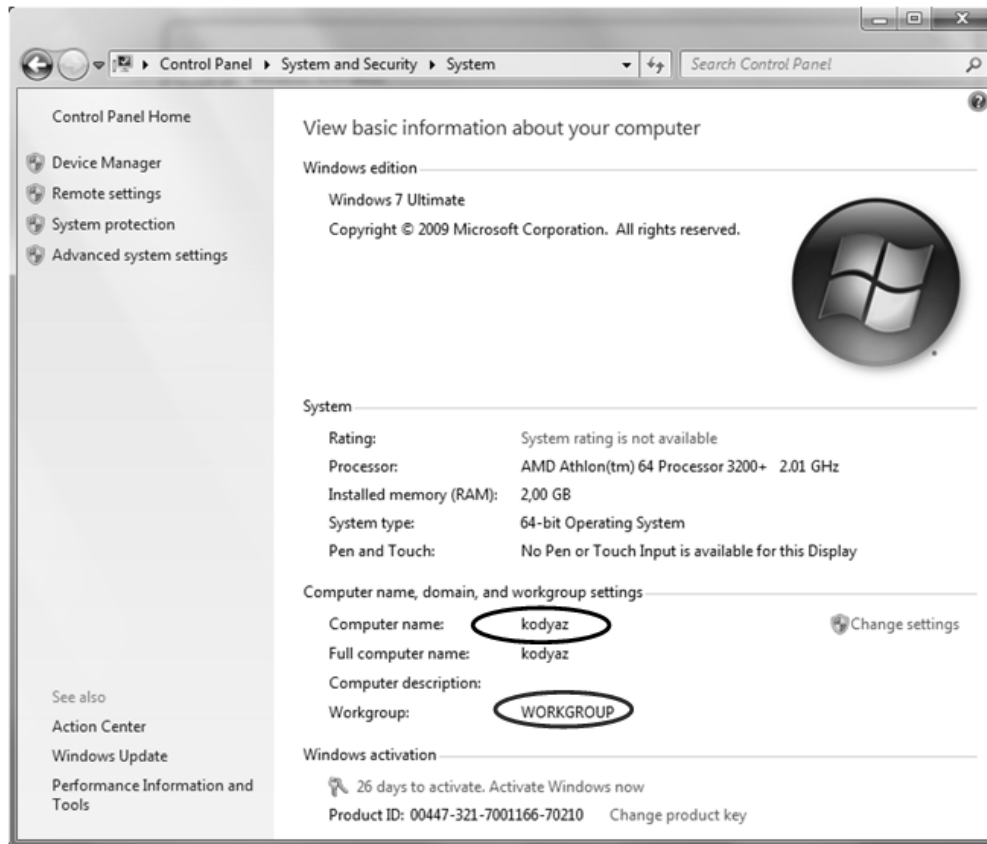


2. Setting PC name and workgroup

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

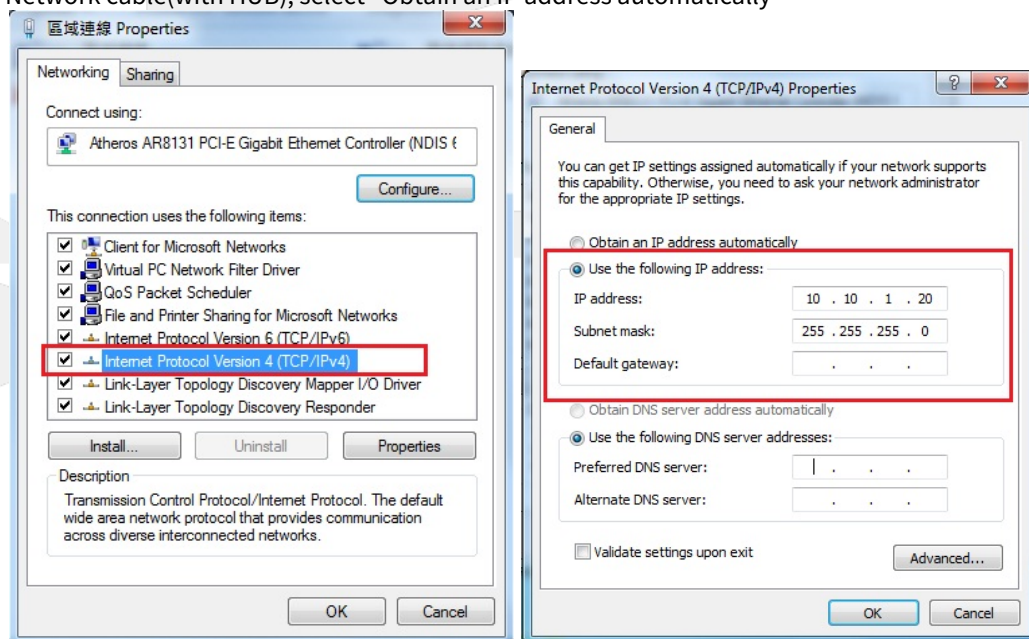
SYNTEC

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

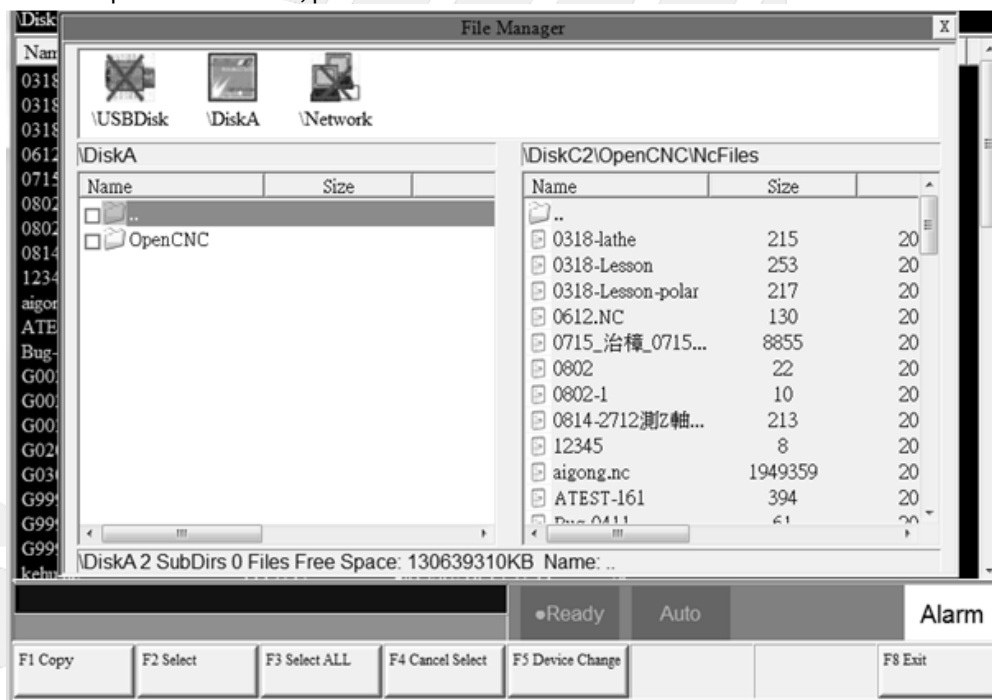


4.5 File Transfer

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.

4.5.1 File Import

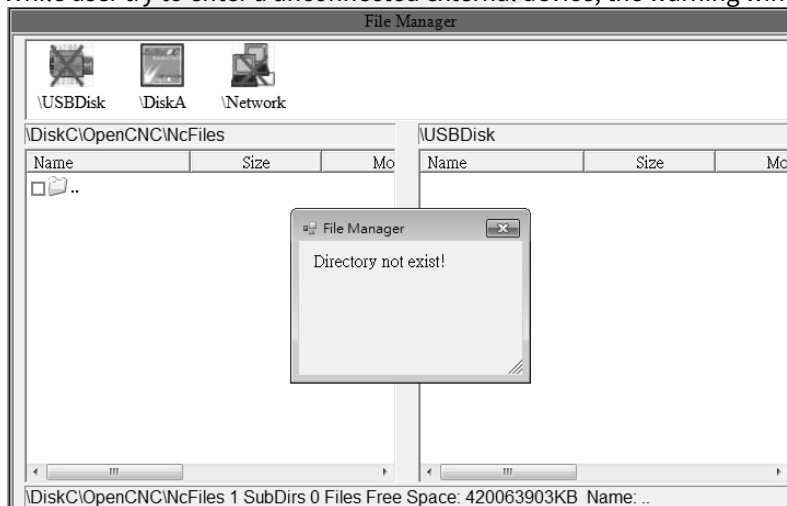
- Condition:
 - None.
- Operation:
 - a. Path: F2 "Program" => F8 "File Manager" => F4 "File Transfer" => F1 "File Import"
 - b. A window will show up on screen, press F5 "Device Select", and move cursor to the desired external device
 - c. Press "Enter" to access to the desired external device. If the 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



4.5.2 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

5 Appendix

5.1 Contact Window

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

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

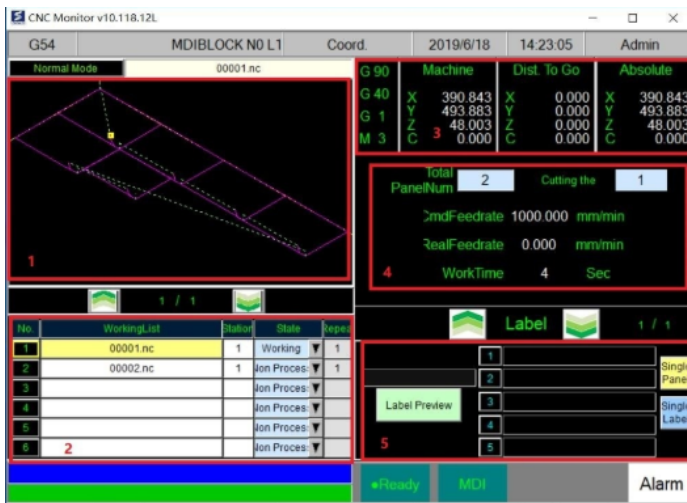
Jiangsu-Nanjing The Suzhou new generation of CNC Equipment Co., Ltd. Chongqing Branch TEL : 0512-69008860-300 Address : Room 505, Unit 3, Building 12, Wuyi Luzhou Guan Chu Yuan, No.99, Tianyuan Middle Road, Jiangning District, Nanjing	Hubei-Wuhan Suzhou new generation of CNC Equipment Co., Ltd. Wuhan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

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



SYNTEC

6 60WE Workinglist Manual



6.1 Operator Panel-F1: Process Frame

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

Part	Item
1	WorkingList
2	Process Path Simulation
3	G-code State/Coordinate
4	Process Information
5	Tag List

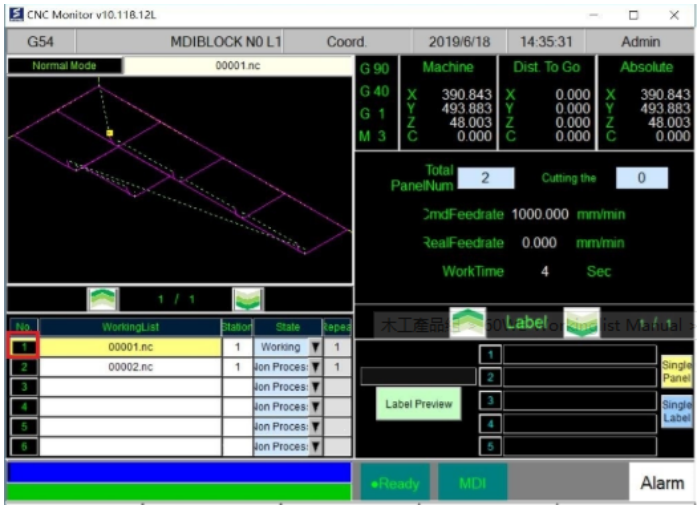
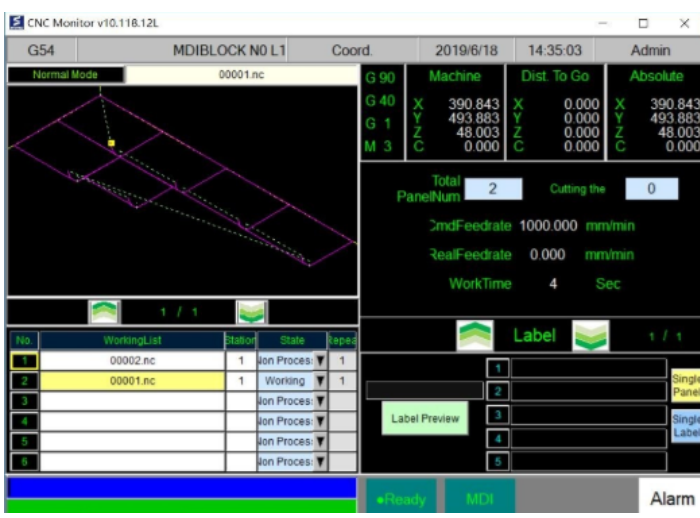
- Description

- 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:

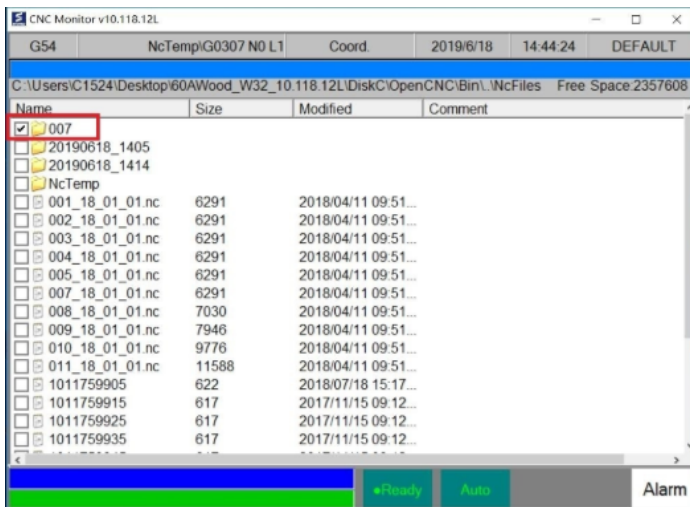
Sequence	NC files	Work Station	State	Repeat Times
1	00001.nc	1	Non Process	1
2	00002.nc	2	Non Process	1

6.2 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	

6.3 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.

6.4 Operator Panel-F4: Reset List

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

Reset List

Before

CNC Monitor v10.118.12L

G54 NcTemp/G0307 N0 L1 Coord. 2019/6/18 14:46:17 DEFAULT

Normal Mode 00001.nc

No	Working List	Status	State	Repeat
1	00001.nc	1	completed	0
2	00002.nc	1	completed	0
3			Ion Procs	
4			Ion Procs	
5			Ion Procs	
6			Ion Procs	

Ready Auto Alarm

After

CNC Monitor v10.118.12L

G54 NcTemp/G0307 N0 L1 Coord. 2019/6/18 14:46:34 DEFAULT

Normal Mode 00001.nc

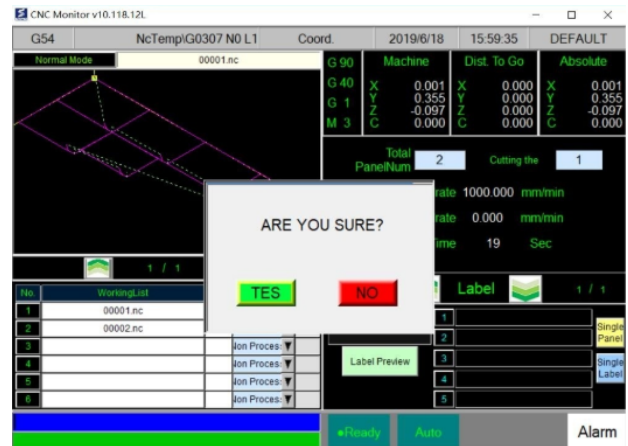
No	Working List	Status	State	Repeat
1	00001.nc	1	Ion Procs	1
2	00002.nc	1	Ion Procs	1
3			Ion Procs	
4			Ion Procs	
5			Ion Procs	
6			Ion Procs	

Ready Auto Alarm

SYNTEC

6.5 Operator Panel-NEXT-F1: Back to Break Point

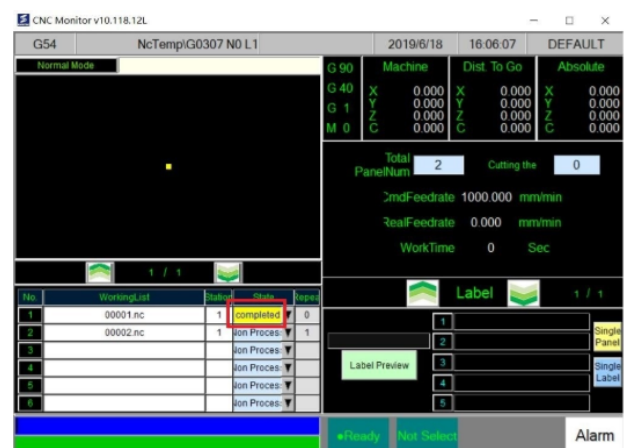
- Press NEXT-F1, a pop-up show on screen



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

6.6 Operator Panel-F5: Clear List

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

6.7 Operator Panel-NEXT-F4: Login

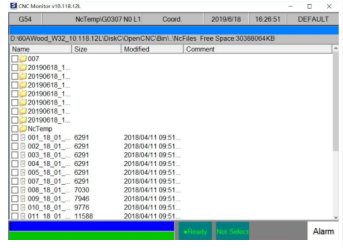
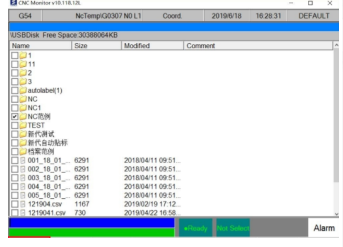
- Enter username and password to login to the system
- login different users can have different permissions
- Permissions can be set in "Authority mangement"

6.8 Operator Panel-NEXT-F5: Logout

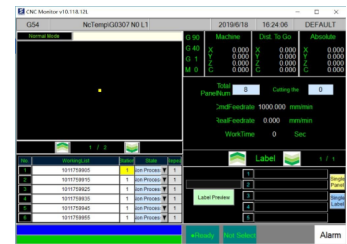
- logout the accessed account

6.9 Workinglist Manipulation

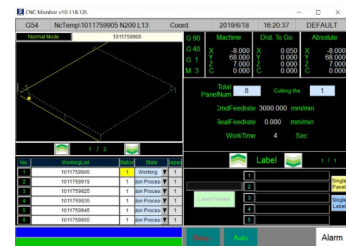
- 6.9.1 Normal Mode
 - **Select Files/Folder, Output Workinglist**

• Operator Frame(F1) → F3 File Manage • F5: 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.

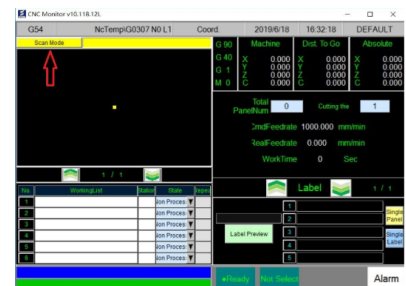


6.9.2 BarCode Scanning Mode

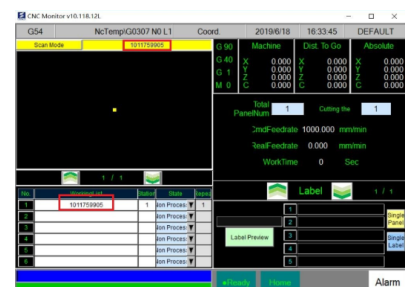
- Press The "Normal Mode" button, change to "Scan Mode", Press again back to "Normal Mode".
- Scan the barcode and the scanned NCfile will be added to workinglist by sequence, if no files found in system, there will show a pop-up hint

Figure

ScanMode



Scanning Barcode and add to Workinglist



6.9.3 Set Repeat Times for the Same File

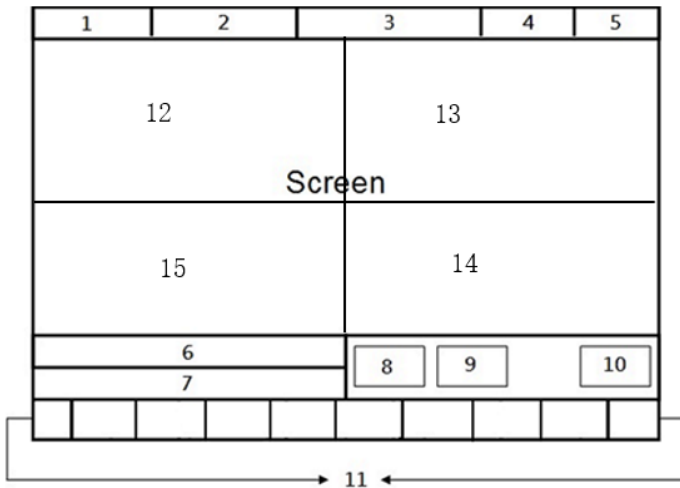
- The default of repeat time is 1, every time the process done, it will minus 1 time.
- If set it be 10, this file will be repeat executed till the number be 0.



SYNTEC

7 Fenu/Sys Introduction-60WE

7.1 Main Screen

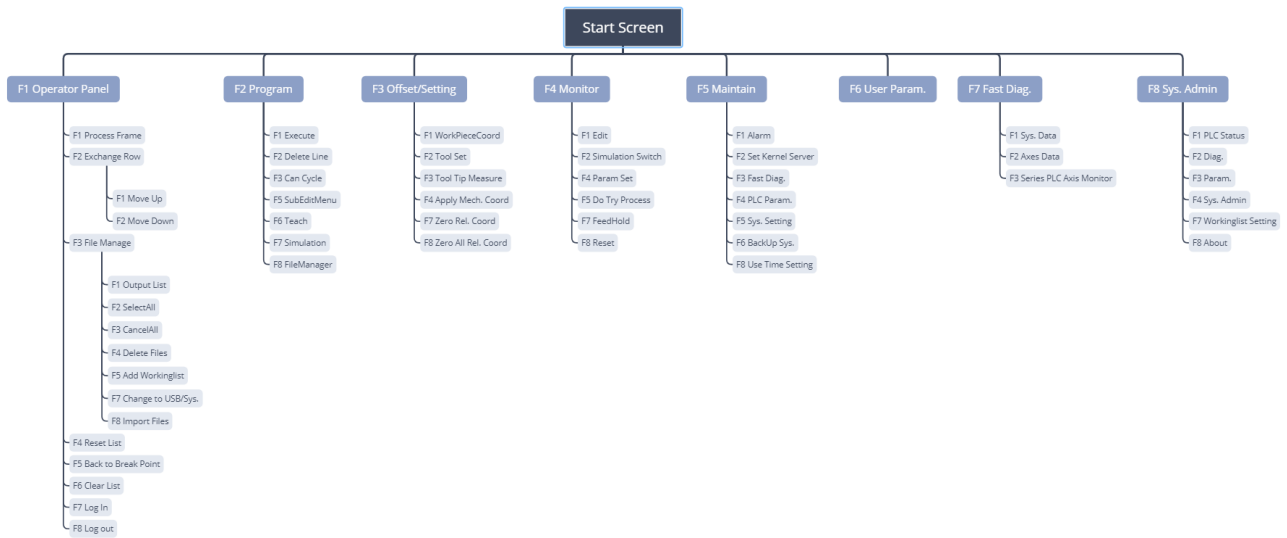


• Screen component description

- a. Current Workpiece Coordinate
- b. Current Processing file and process line
- c. Current Screen
- d. Date
- e. Time
- f. Data Input
- g. Hint
- h. Controller Status
- i. Mode
- j. Alarm
- k. Fenu Selection
- l. Process Simulation Previes
- m. G-Code status and Coordinate
- n. Label Information Previes
- o. Workinglist

SYNTEC

7.2 Function key Tree



SYNTEC

8 Hardware-60WE



60W-E (Serial-bus controller)

SYNTEC

9 Software Spec.-60WE

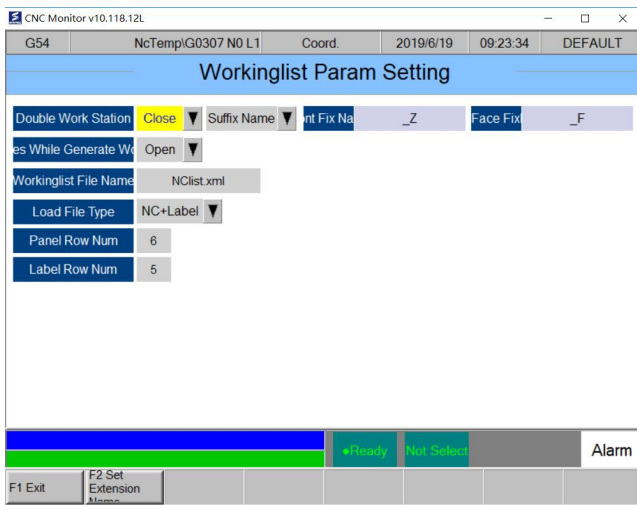
Item	Content
Axes(Standard / Option)	4/6
Spindle Number	6
Axes Group Number	4
Workpiece Coordinate Number	100
BPT	1000
Software Version	10.116.54x, 10.118.12x, 10.118.x
Communication Protocol	M3/EtherCAT: 60W-E
Option	• 4-Axes RTCP (Option-29) • ROT (Option-32) • Auto Labeling Machine Function (Option-37) • Built-in CAM (Option-36)
Standard	• WorkingList • High speed high precision • Panel Nesting/Redo function • Support ZPL Printer to print Labels

9.1 Manual Objective Machine

This Operation Manual is suitable for Syntec SCD-60WA and FC-60WA controller.

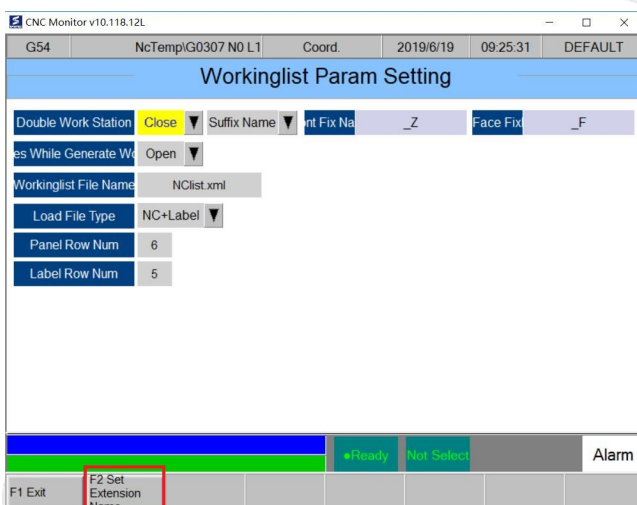
SYNTEC

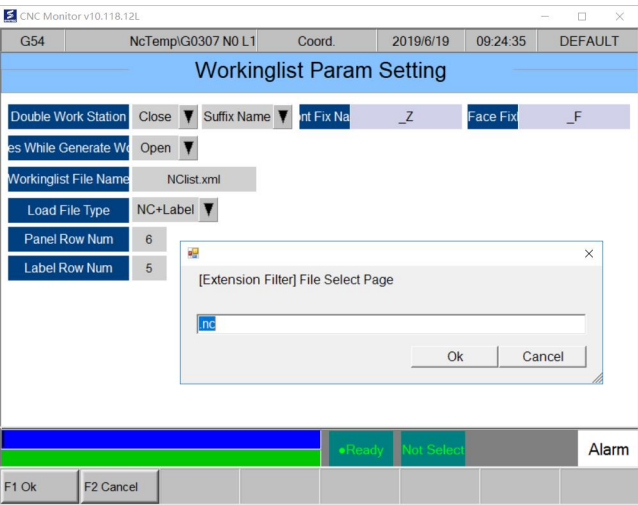
10 Workinglist Param Setting-60WE



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

10.1 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)

10.2 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,repeatly, 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

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																	

SYNTEC

	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																			

10.3 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.

10.4 Panel Row Number

- The number of rows in Workinglist

10.5 Label Row Number

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

10.6 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.



SYNTEC