SYNTEC TECHNOLOGY CO.,LTD.

200 Series Lathe Operation Manual.

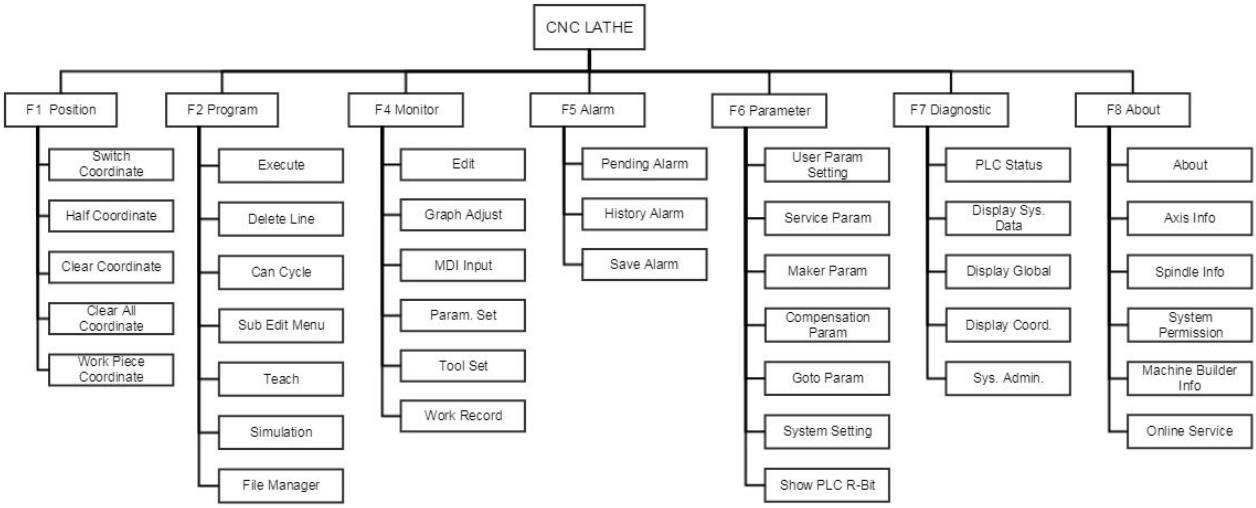
匯出日期：2023-05-10
修改日期：2019-05-29

—

SYNTEC

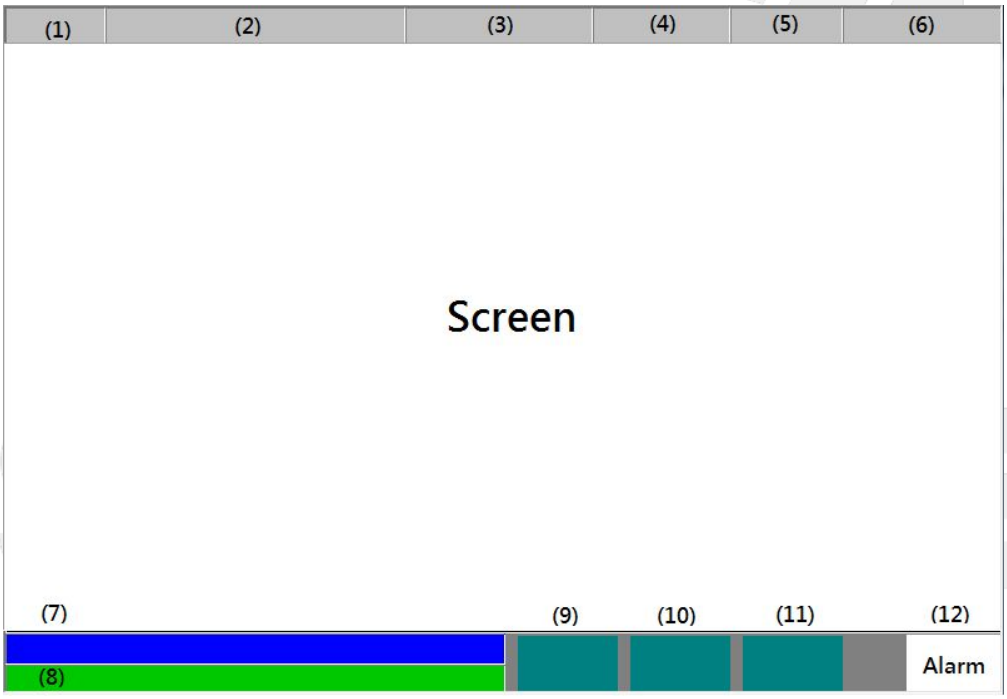
1 Lathe Controller Interface

1.1 CNC System Configuration



1.2 Screen Sections

The screen sections are shown below:



Meanings for Fields on the Display:

- (1) Work Piece Coordinate No.
- (2) Program Name / Line No.

- (3) Screen Title
- (4) Date
- (5) Time
- (6) User
- (7) Input
- (8) Hint
- (9) Status
- (10) Mode
- (11) Dipole Status (Only visible under Dipole)
- (12) Alarm

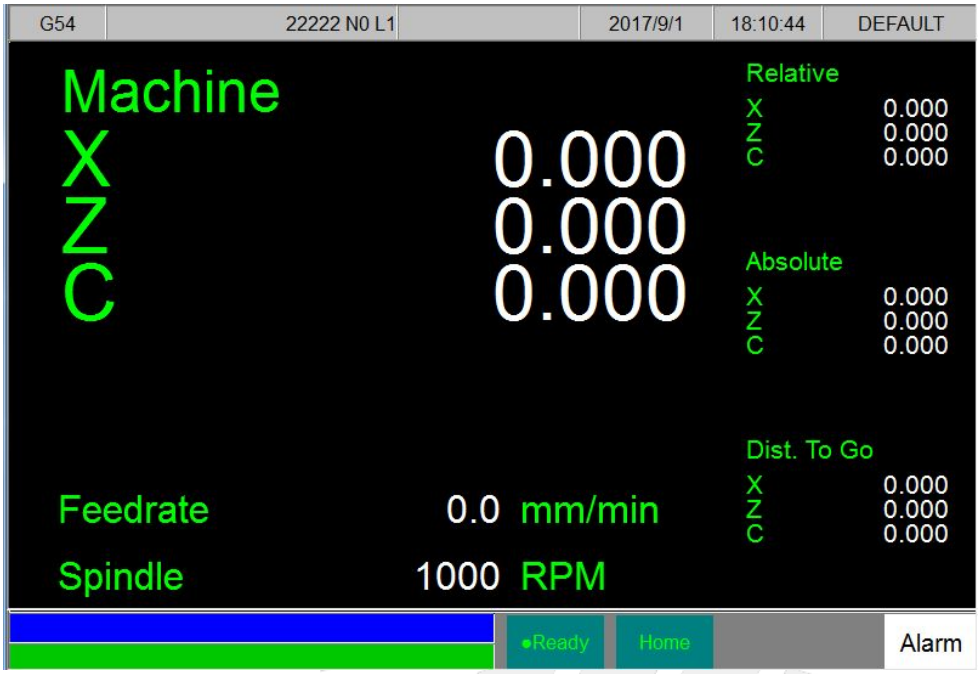
1.3 Main Menu Selections

The following diagram is the main menu selections for SYNTEC Lathe controller. To operate SYNTEC Lathe controller, users simply make the selections by pressing function keys, F1 ~ F8 located on the bottom of the screen.



1.3.1 F1: Position

This selection displays coordinate settings of current position. It can also be used to reset the position of relative coordinate. Pressing function key, F1, under the main menu to enter this selection
(Note:This is the first screen when the system is booted up)



Screen Layouts:

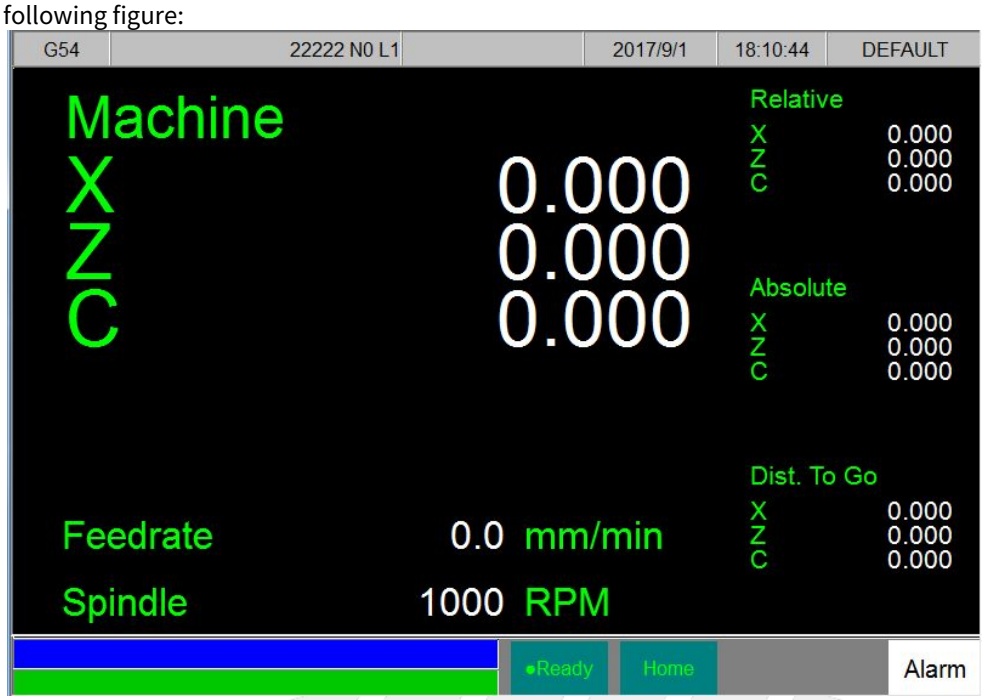
- X:X axis coordinate.
- Z:Z axis coordinate.
- C:C axis coordinate.
- Feedrate: Cutting Feedrate which unit is mm/min or inch/min.
- Spindle: Spindle speed which unit is RPM.
- Machine coordinate: (Relative coordinate of working platform)
  The current position relative to machine zero point is shown as machine coordinate on the display.
- Relative coordinate:
  The current position relative to the previous location.
- Absolute coordinate:(Programming Coordinate)
  The current position relative to the origin of user defined coordinate is shown as an absolute position on the display.
- Distance To Go:
  The remaining distance relative to the next position with sign +/- to display direction.

Descriptions of Function key selections:

- F1: Switch Coordinate

  Function: Switch Coordinate Displays.
  Operation: Under the Position submenu, whenever users press F1 key, the values and coordinate on the left corner of the display will toggle among the four different coordinates with bigger fonts as shown in the

following figure:



- F2: Half Coordinate

  Function: Set the center point of work piece as coordinate origin.
  Operation: Under the Position submenu, when the message line shows "X Input"(or "Y Input" or "Z Input"), press "F2 Half Coordinate" and the relative coordinate value will become half in order to easily calculate center of a workpiece.

- F3: Clear Coordinate

  Function: Reset the value of X(or Y or Z) axis relative coordinate to zero.(No effect on other axis)
  Operation: Under the Position submenu, when "X Input" is entered in the Data Input row (or "Y Input" or "Z Input"), press F3 to reset the value of X(or Y or Z) axis relative coordinate to zero.

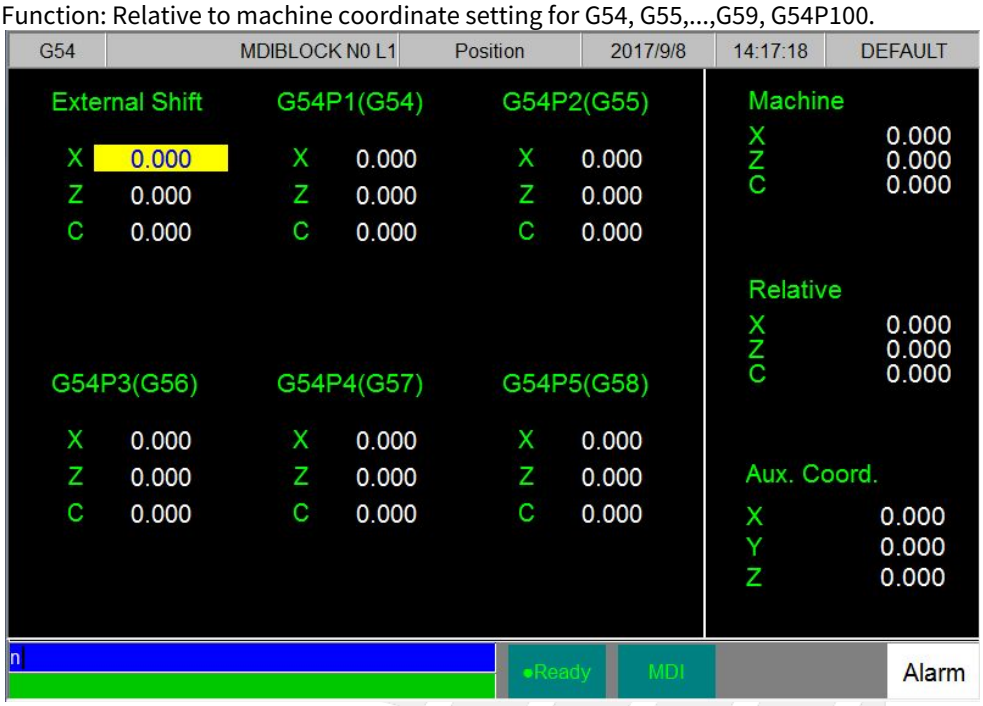
- F4: Clear All Coordinate

  Function: Reset all relative coordinate to zero.
  Operation: Under the Position submenu, pressing "F4" will reset all relative coordinate to zero.

SYNTEC

- F5: Workpiece Coor.

Function: Relative to machine coordinate setting for G54, G55,...,G59, G54P100.



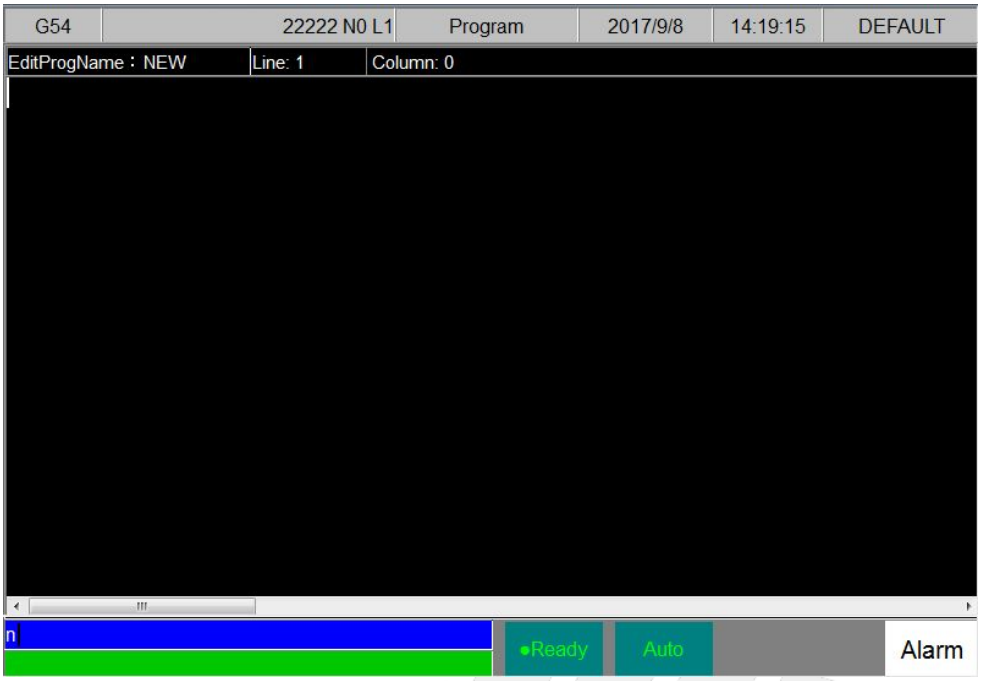
Operation: Under the Position submenu, press "F5" key and the following screen will show up. By pressing "F1" key, then users can begin to set the auto machine coordinate settings of G54~G59 one by one. (The system needs to be in manual date input (MDI) mode.)

a. "External Shift": operator can set the all coordinate G54..G59 at the same time .
b. CNC default G54, if user don't set any G54…G59 in the NC file

1.3.2 F2: Program

This selection provides users with program file management and editing. With a full screen editor, users can use arrow keys（↑、↓、←、→）to move the cursor to anywhere on the screen for editing purpose. Press F2 under the main menu to enter this selection. The full screen editor is shown as follows:

SYNTEC



Descriptions of Program Sub menu Key Selections:

- F1: Execute

  Function: Load the current program and jump to Run Menu
  Operation: Under Program submenu, press F1 to load current program and jump to Run menu.

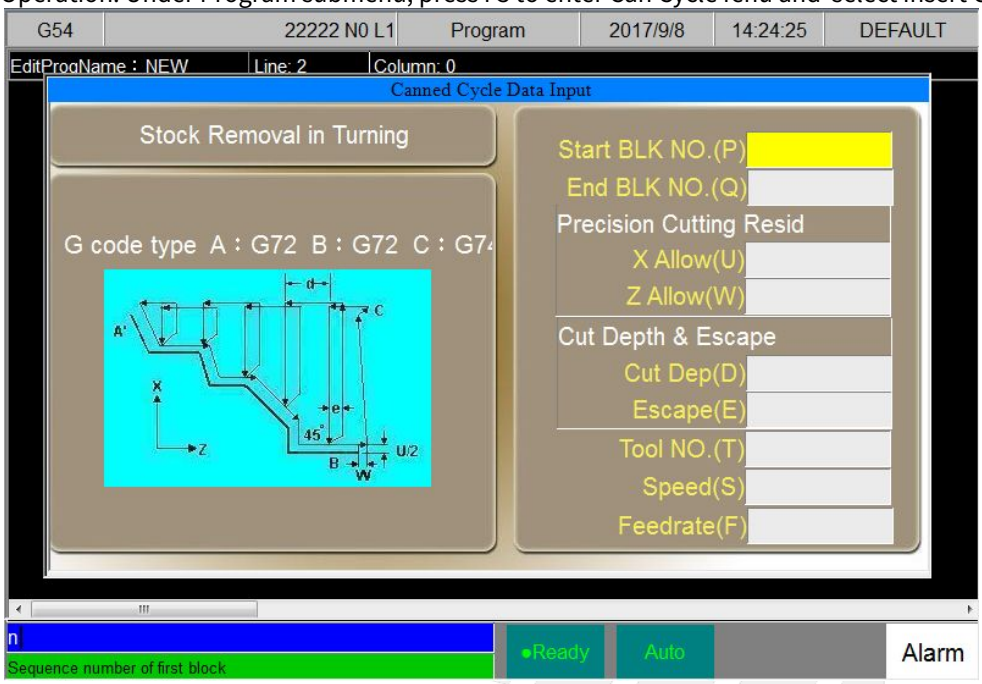
- F2: Delete Line

  Function: Delete a line at cursor position.
  Operation: Under Program submenu, press F2 to delete a line where the cursor is located.

SYNTEC

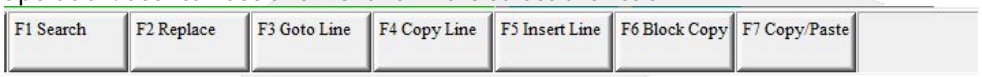
- F3: Can Cycle

  Function: Enter Can Cycle fenu to add or edit a cycle.
  Operation: Under Program submenu, press F3 to enter Can Cycle fenu and select Insert Cycle or Edit Cycle.



- F5: SubEdit Menu

  Function: Edit sub function "Search" "Replace" "Goto line" "Copy line" "Insert line" "Teach" .
  Operation: user can use this menu for more edit sub function



  - F1: Search

    Function: Search String
    Operation: Under Program submenu, press F5"SubEdit Menu" and then F1 "Search" to search string. A dialog box will pop up asking users to input a string as shown in the following figure. After keying in a string, press F1 to start searching.

  - F2: Replace

    Function: Replace string.
    Operation: Under Program submenu, press F5 "SubEdit Menu" and then F2"Replace" to replace string. An dialog box will pop up asking users to input the replacing string and the new string as shown in the following figure. After keying in a string, press F1 to start replacing.

  - F3: Go To Line

    Function: Go to a line number
    Operation: Under Program submenu, press F5"SubEdit Menu" and then F3"GOTO line" to go to the line number. A dialog box will pop up asking users to input a line number. After keying in a number, press F1 to go to the desired line.

- F4: Copy Line

  Function: copy a line from current cursor to next line
  Operation: Under Program submenu, press F5 "SubEdit Menu" and then F4"Copy line" to go to the next line.

- F5: Insert Line

  Function: Insert a space line above current cursor line
  Operation: Under Program submenu, press F5 "SubEdit Menu" and then F5"Insert line" to Insert a new space line.

- F6: Block Copy

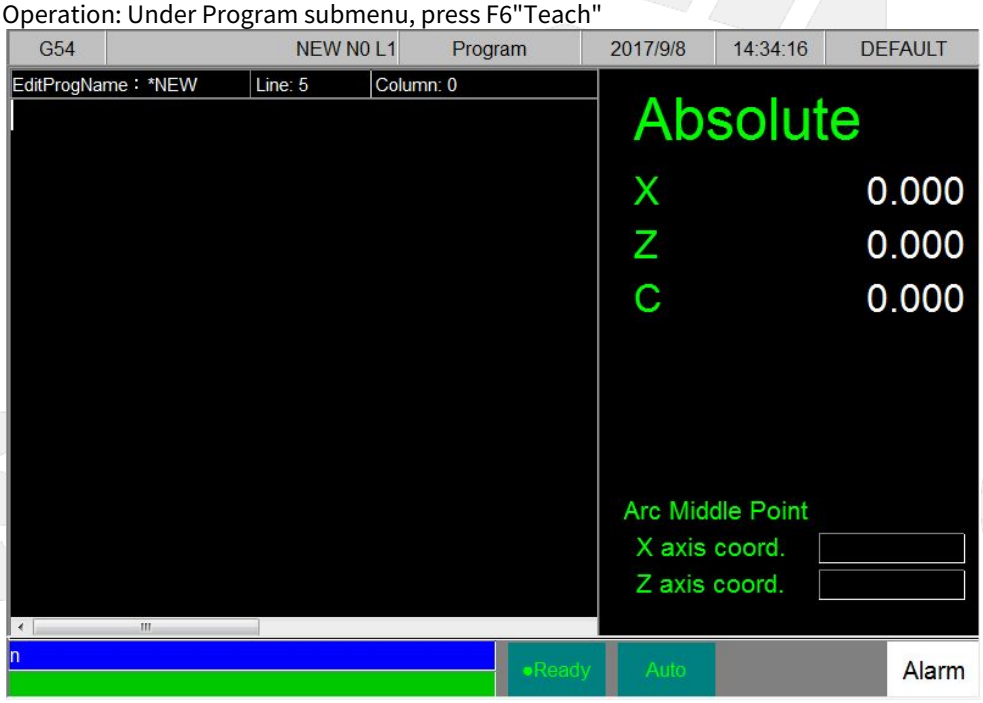
  Function: support functions of "Start Line", "End Line", "Block Cut", "Block Copy", & "Block Paste" to edit multiple lines.
  Operation: Under Program submenu, press F5 "SubEdit Menu" and then F6"Block edit" to edit blocks through available functions.

- F7: Copy/Paste

  Function: support functions of "Cut", "Copy", "Paste", "Undo", "Select All" to edit multiple lines.
  Operation: Under program menu, press F5 "SubEdit Menu"and F7 Copy/Paste to use available functions

- F6: Teach

  Function: Teach current absolute coordinate to NC files
  Operation: Under Program submenu, press F6"Teach"



  Descriptions of Teach sub Key Selections:

  - F1: Rapid Teach

    Function: Add "G00" code to NC files. Perform G00 to current absolute coordinate.

- F2: Linear Cut Teach

  Function: Add "G01" code to NC files. Perform G01 to current absolute coordinate.

- F3: Arc Cut Teach

  Function: Add "G02" or "G03" code to NC files
  Press(1st time) this key "arc teach ": CNC will automatically put current value to Arc middle point.
  Press(2nd time) this key "arc teach ": CNC will automatically calculate G02 or G03, and filled the complete code to NC files.

- F4: Cancel Arc Middle Point

  Function: When arc teach function performs, user can use this key to abort middle point teach.

- F6: Delete Line

  Function: When user uses Teach function, user can use this key "Delete line" to delete an entire command line.

- F7: Next Coor. Page

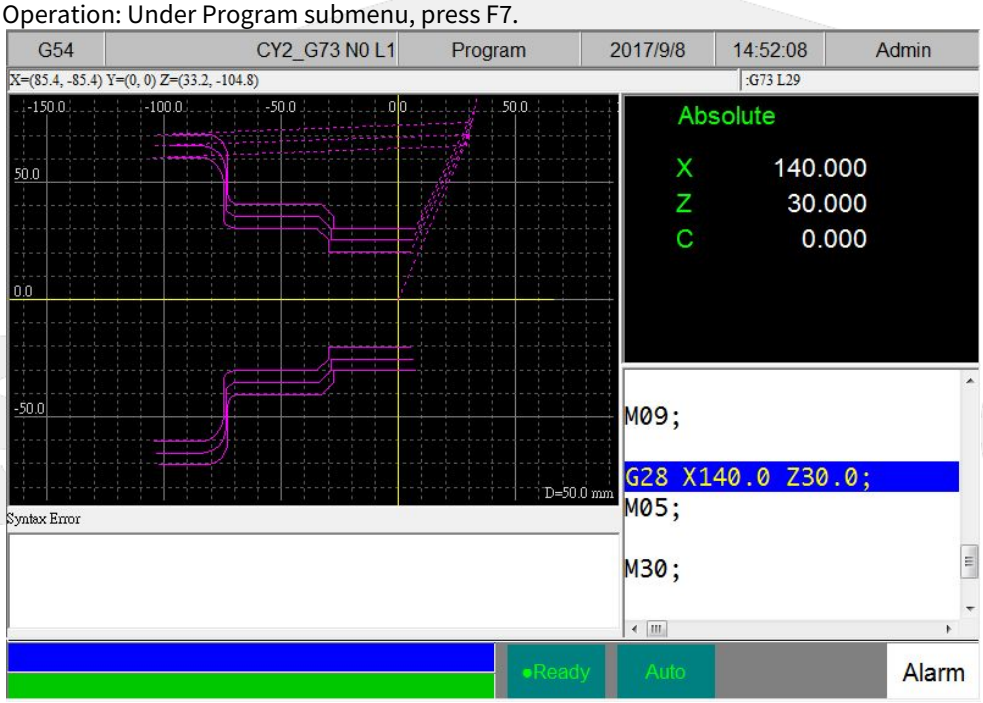
  Functions: Switch the coordinate displays when more than 3 axes exist.

- F8: Point Teach

  Function: Add point position to the NC files by the current absolute coordinate.

- F7: Simulation

  Function: Simulation program can verify the accuracy of the editing program.
  Operation: Under Program submenu, press F7.



Descriptions of F7 "simulation" sub Key Selections:

- F1: STEP

  Function: To simulate and check NC files STEP by STEP
  Operation: Under Program submenu, press F4"Simulation" and then F1"Step".The operator can use this function ,to check NC file Step by STEP

- F2: Continue

  Function: To simulate NC file once.
  Operation: Under Program submenu, press F4 "Simulation" and then F2"Continue". The operator can use this function to check NC file whole picture, when push button.

- F3: Zoom

  Function: To enlarge the workpiece graph.
  Operation: Under Program submenu, press F4 and then F3.The operator can use the "←","↑","→", "↓" cursor to move the frame to the determined area. And use "PageUp" "PageDn" to Enlarge this area .

- F5: Graph Reset

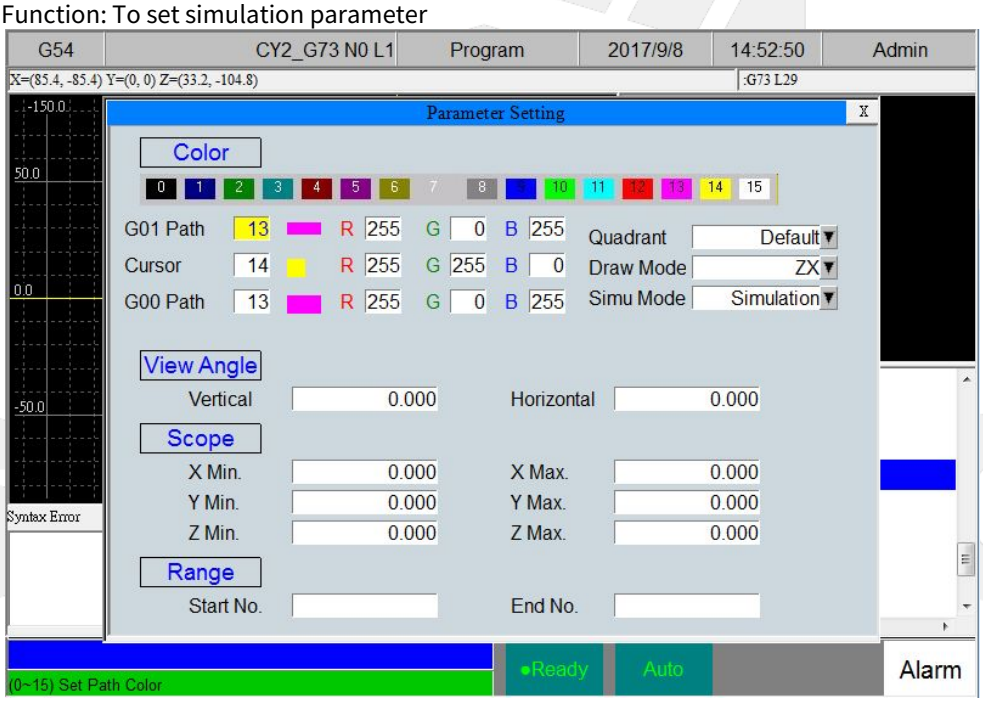
  Function: To recover the zoomed workpiece graph.

- F6: Abort

  Function: To abort simulation action

- F8: Simu. Setting

  Function: To set simulation parameter



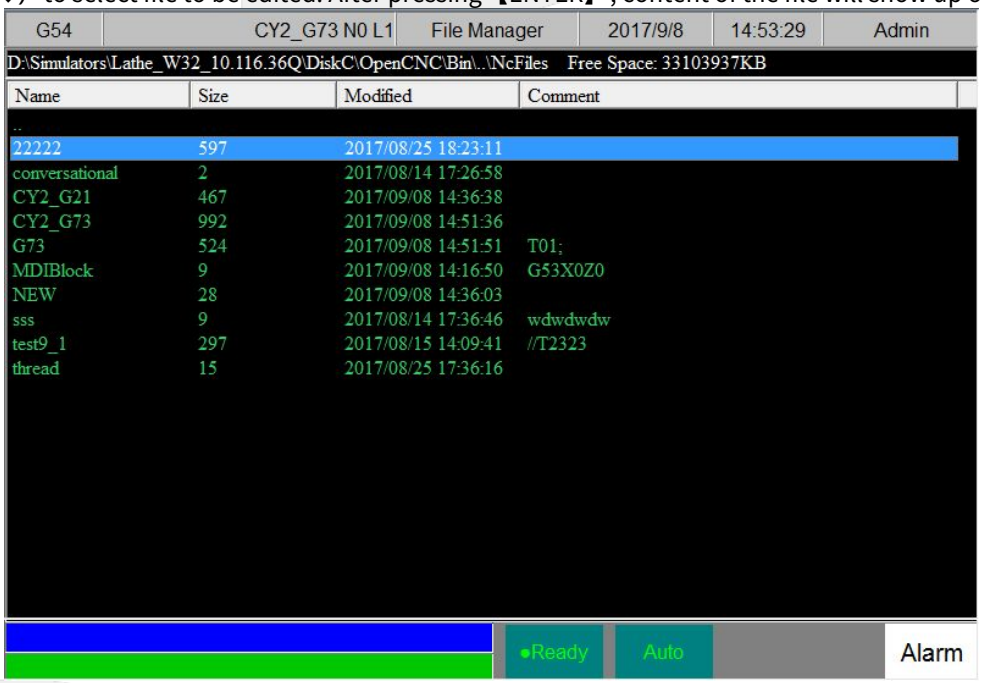
Simulation Parameter Description:

- Color: Total 15 colors can be used in simulation.
- G01 & G00 Path Color: user can select cutting path color by enter color codes. The RGB code will change as color code is selected.
- Cursor Color: user can select cutting point color by entering color codes.

- Quadrant: user can define the quadrant to display when simulating path in 3D view.
- Draw Mode: user can select simulation plane by this parameter (XY, YX, XZ,ZX, YZ, ZY, XYZ)
- Simulation Mode:
  Simulation: When operator enters "Run" screen, CNC would automatically simulation at that screen
  Direct Rraw: When operator enters "Run" screen, CNC would not simulate the path but start drawing path after cycle start.
  Not SImulation: When operator enters "Run" screen, no simulation path will be displayed.
- View Angle: when XYZ drawing mode, user can rotate 3D View angle verically and horizontally.
- Scope: user an setup simulation window scope. Value 0 means automatic.
  Under "direct draw" mode, operator must set drawing scope to match where the simulated path position.
- Range: Setup the range of program lines to simulate partial of the program path.

- F8: File Manager

Under Program submenu, press F5 and the following diagram will show up. Users can use arrow keys（↑、↓）to select file to be edited. After pressing【ENTER】, content of the file will show up on the screen



Descriptions of Key Selections:

- F1: New File

  Operation: Step 1: A dialog box will prompt users with "New File ". Type in the new file name and press 【ENTER】. Step 2: An empty screen shows up waiting users to type in a new program.

- F2: Copy File

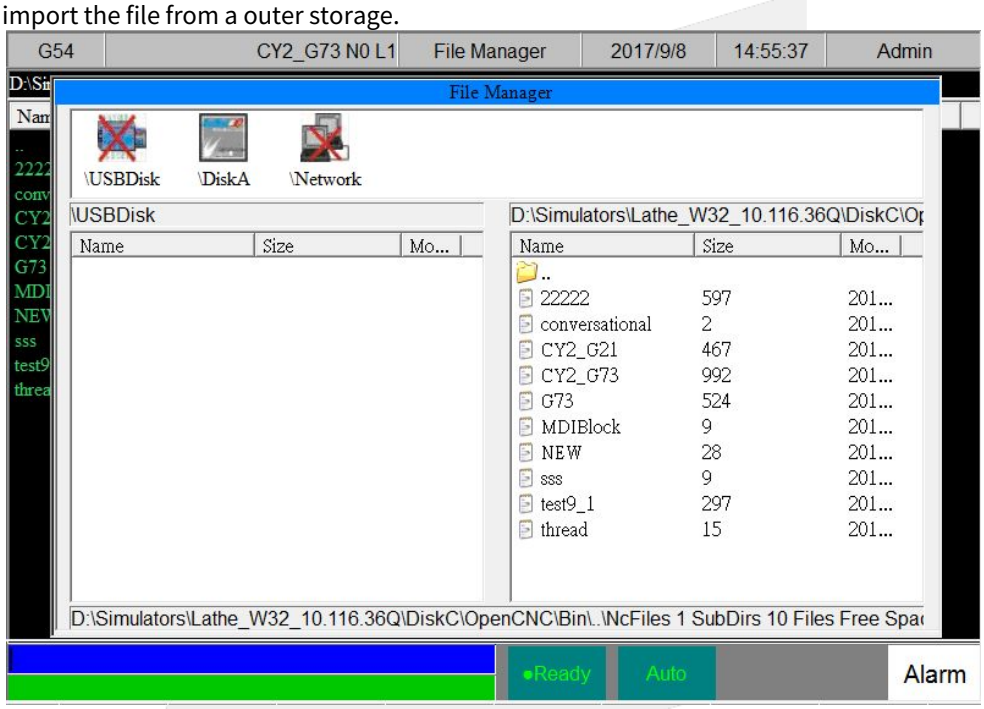
  Operation: After pressing F2, a dialog box will prompt users to type in a file name and press 【ENTER】. The current file is then copy to the hard disk with a different file name.

- F3: Delete File

  Operation: Select a file to be delete by pressing （↑、↓）. A dialog box will pop up to confirm this operation.

- F4: File Transfer

  Function: To do Import/Export file from outer storage.
  Operation: Insert a USB disk to the control or connect to a network folder, then press F4→F1 File Import or F2 File Export. Select a file name by pressing（↑、↓、←、→）. Press【ENTER】to import the file from a outer storage.



  P.S.
  - USB drive format must be FAT32, and no larger than 32GB.
  - The operation of transfering files on USB disk or network folder are identical.

- F5: Execute

  Function: To load the file that cursor is currently on, and jump to Run screen
  Operation: Use ↑↓←→ keys to select file, and press F5 to load it and jump to Run screen.

- F6: Select DNC File

  Function: To run a large NC file without copying it into controller memory
  Operation: Press F6 and use ↑↓ keys to select file on USB disk or network folder, then press F1 to load the file and jump to Run screen. When opening a DNC file, the title row displays as "DNC:+file name".

1.3.3 F3: Custom Function

Pressing "F3" under the main menu to open the customized interface. Geranally, this button will be empty.

1.3.4 F4: Run

This selection displays machining speed, time, manual data input (MDI) and other machine information such as coordinate, range or program at the run time. Press F4 under the main menu to select this function.



Each Key Selections are described below:

- F1: Edit

  Function: Open the currently loaded program to edit.
  Operation: Under Run submenu, press F1 to open this program and jump to Edit screen.

- F2: Graph Adjust

  Each Key Selections are described below:

  - F1: Zoom

    Function: To enlarge the workpiece graph.
    Operation: Under Monitor submenu, press F2 and then F1.The operator can use the cursor to move the frame to the determined area.

  - F2: Graph Reset

    Function: To recover the zoomed workpiece graph.
    Operation: Under Monitor submenu, press F2"Graph Adjust" and then F2"Graph Reset"

- F3: MDI Input

  Function: Manual Data Input
  Operation: Users can operate SYNTEC Lathe Controller manually in MDI mode.

a. Press F3 under Monitor submenu and type in single-line G or M code.
b. Press F1 (OK) to confirm the input command.
c. The typed-in command line will show on right upper corner of the screen.
d. Press 【CYCLE START】 on the machine panel to execute the single-line command.
e. The following figure shows an example of this function.

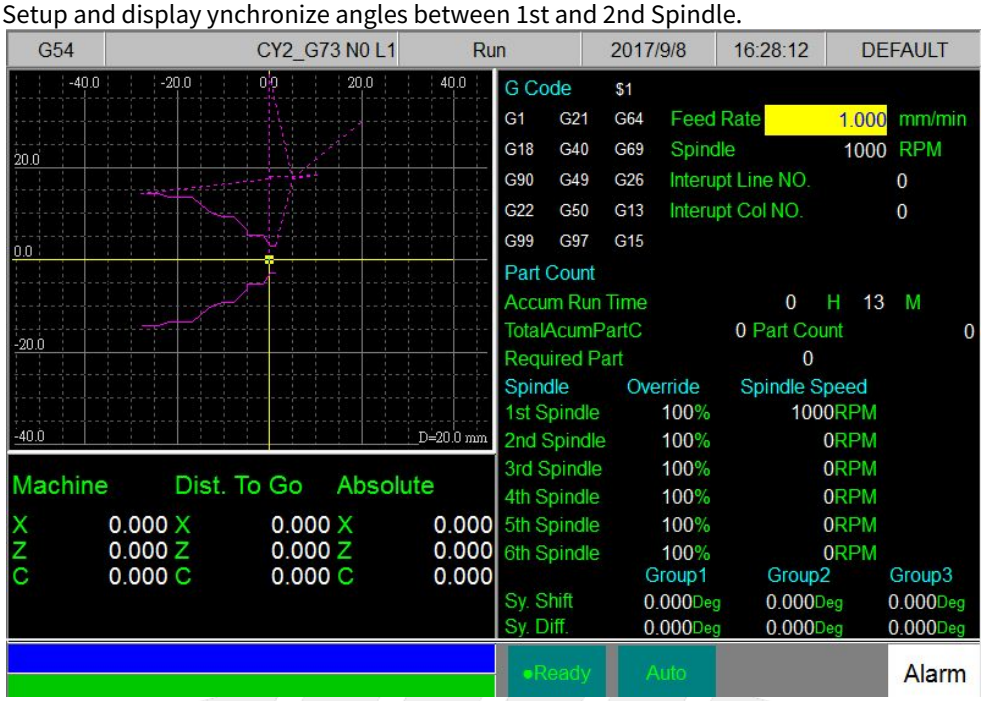


- F4: Param. Set

  Function: Display modal G codes, spindle status, setup part count, and setup required count number.
  Operation:
  - Users can set the part count number and target number
    - When CNC execute M02, M30, or M99, part count would add 1 automatically
    - When part count reaches required numbert, CNC would stop executing.
  - Manually setup spindle RPM and Feedrte as standard F and S.
  - Displays all modal G Codes that are currently running.
  - Display instantaneous load of spindles.

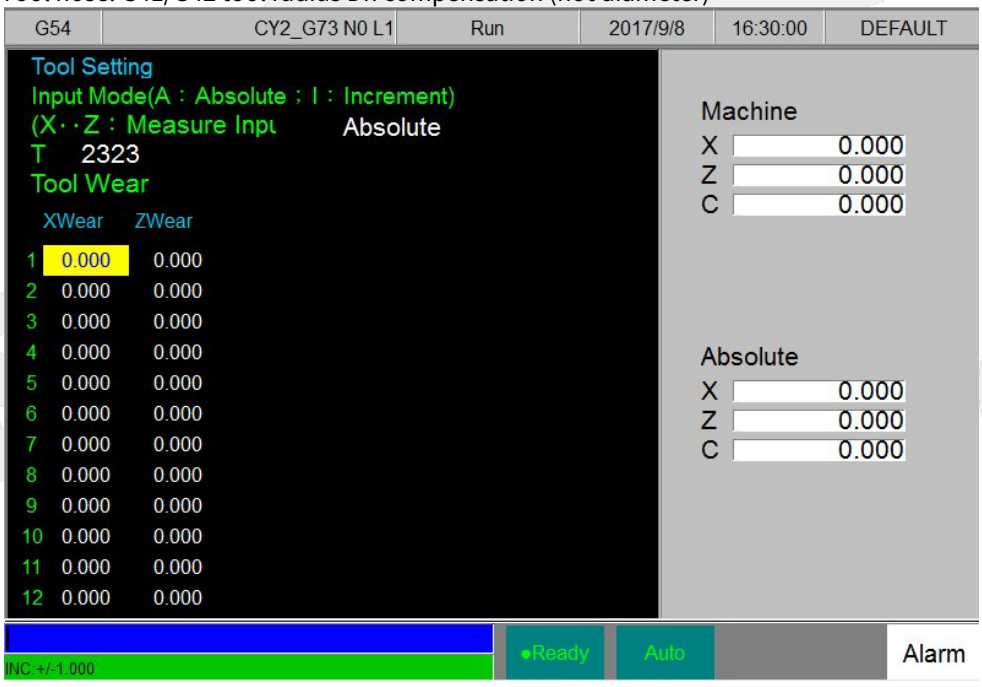
SYNTEC

- Setup and display ynchronize angles between 1st and 2nd Spindle.



- F5: Tool Set

Function: To set the tool compensation value, including tool length , tool wear , tool nose , describe as below :
Tool length: G43/G44 tool length Hn compensation
Tool Length wear: for small length dimension adjust
Tool nose: G41/G42 tool radius Dn compensation (not diameter)

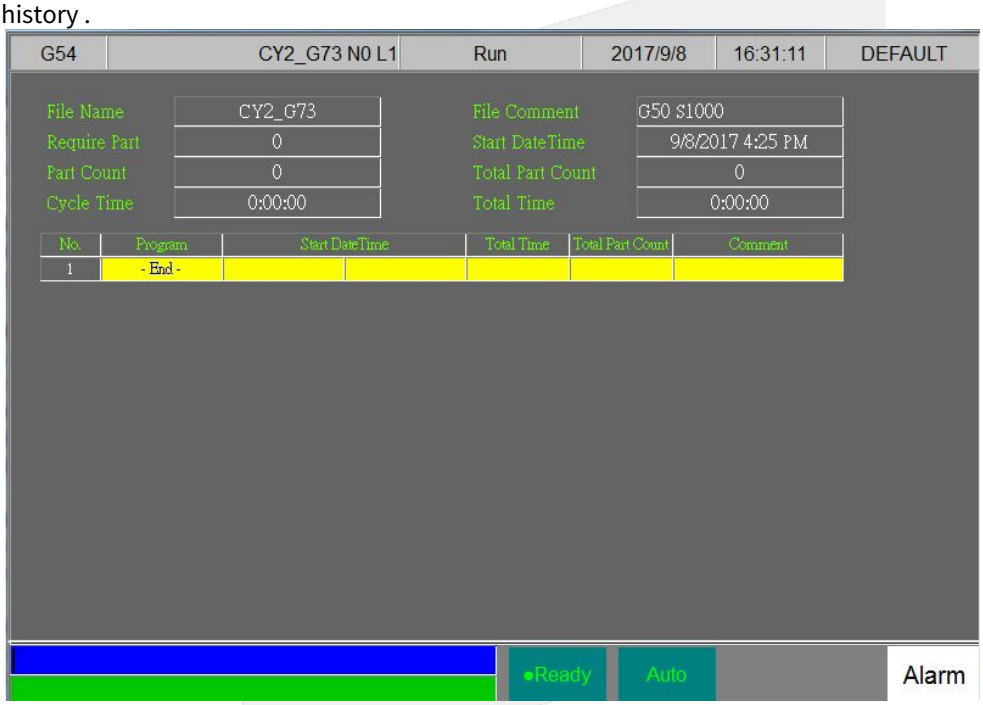


Operation procedure:

a. Main screen press F4 "monitor "
b. Press F5 "Tool Setting"
c. F1 " Tool Wear Set"
d. F2 " Tool Length Set "
e. F3 " Tool Nose Set"
f. F4 "Working Shift"
g. Use " Page Up "," Page Down ","↑", "↓ ", "← ", "→ "select tool No ,and X,Z item

- F8: Work Record

Function: This table can record 300 sets executed NC file ,this is very helpful to know end user working history .

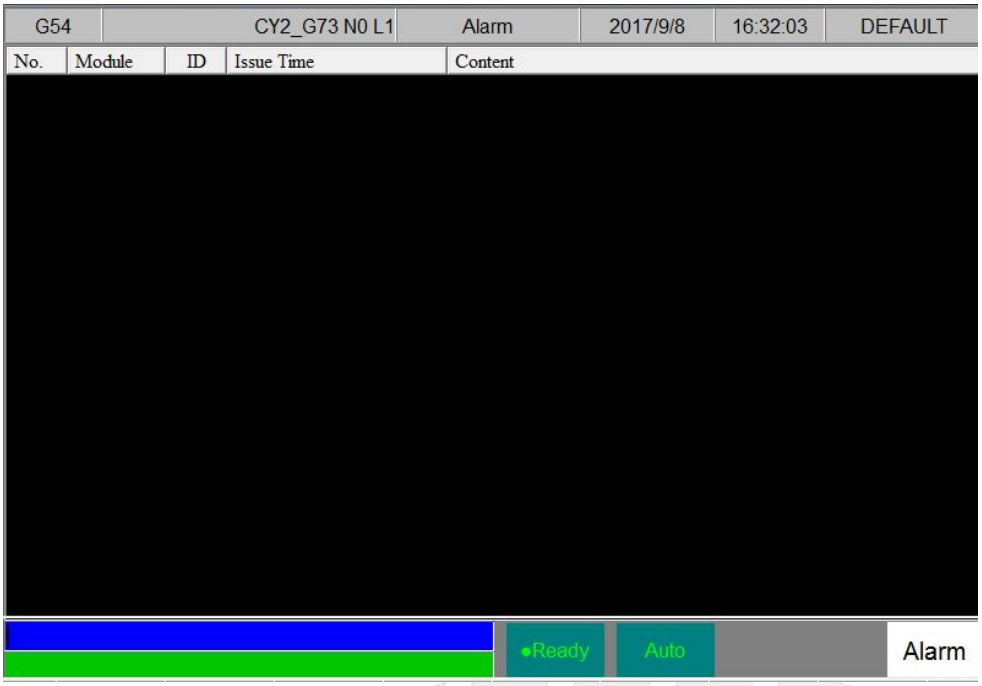


Operation Procedure:

a. Press F4 "Run" at Main screen.
b. Press F8 "Work Record".

1.3.5 F5: Alarm

Whenever the system or the program stops running due to some errors, there will be an alarm message shown on the screen. In order to clear the errors, users can press F5 in the main menu for Alarm submenu as shown in the following figure.



Each Key Selections are described below:

- F1: Pending Alarm

  Operation: Under the Alarm submenu, press F1 to show current alarm situation.

- F2: History Alarm

  Operation: Under the Alarm submenu, press F2 to show the alarm history of the system.

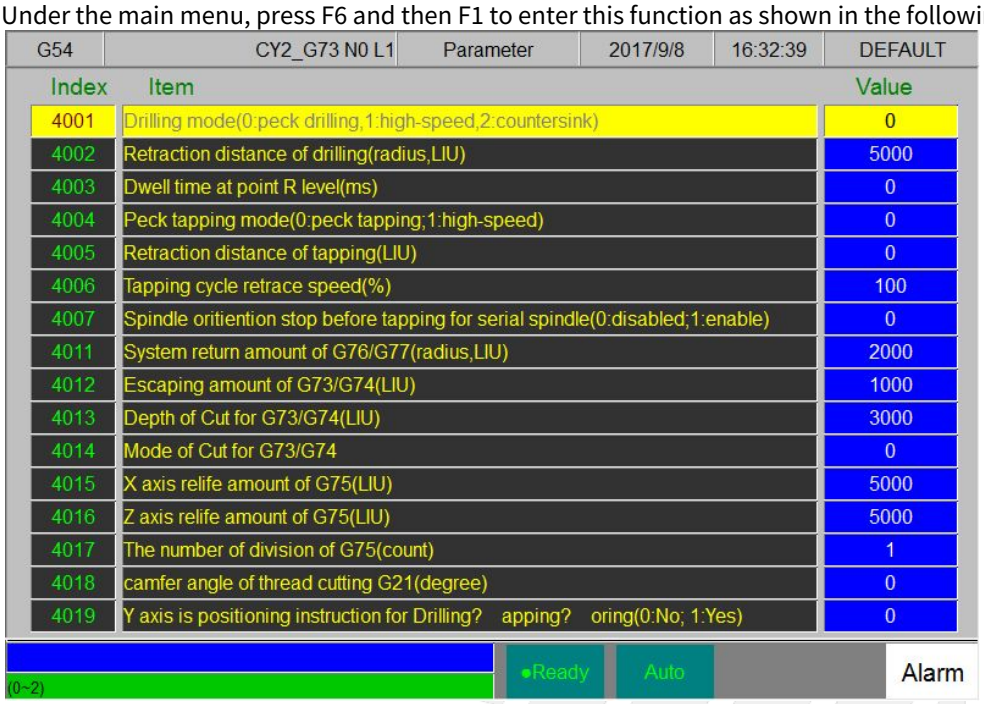
- F5: Save Alarm

  Function: Save Alarm History To File.
  Operation: Under the Alarm submenu, press F5 to save alarm history to a file as shown in the following figure. A dialog box will prompt users to type in file name to be saved. After selecting a disk drive by using (↑、↓、←、→), users press 【ENTER】 to confirm this operation.

SYNTEC

1.3.6 F6: Parameter

Under the main menu, press F6 and then F1 to enter this function as shown in the following figure.



Function:
Parameter setup for all settings of system such as servo parameters, mechanical parameters, programmable travel range, maximum machining speed and so on.
Operation:
User can change parameters anytime, but will not become effective until RESET is pressed. Some parameter with start sign* needs reboot system to become effective.

Function keys are described below:

- F1: User Param.

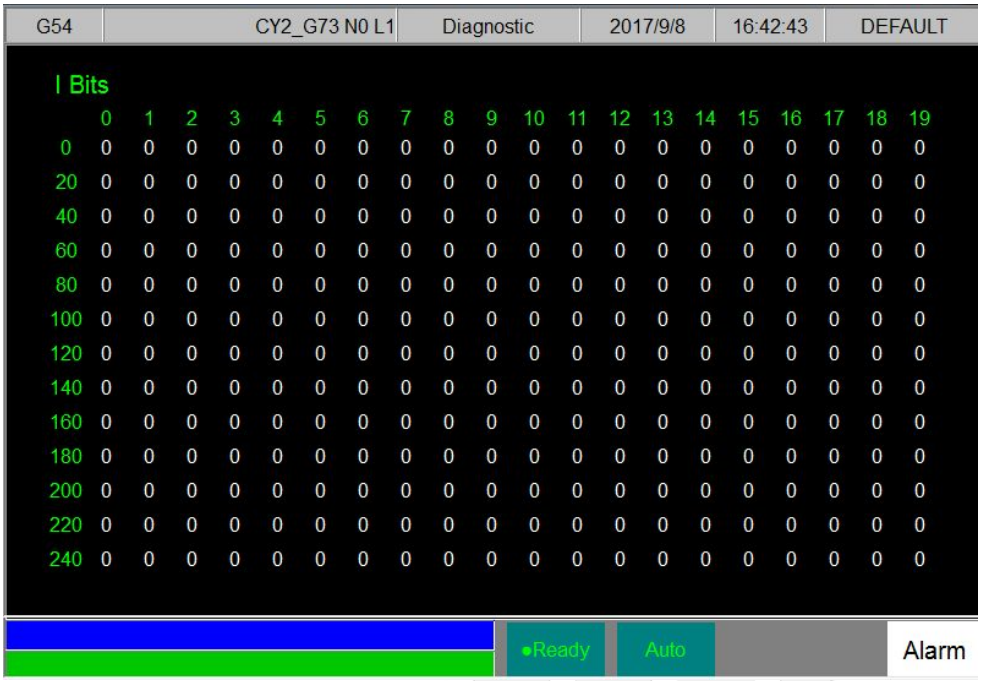
  This parameter group lists out all maching and cycle related settings.

- F5: Go To Param

  If the operator or engineer wish to jump to a specific parameter number, press F5 to cakll pop-out window. Enter parameter number then press F1 to jump.

1.3.7 F7: Diagnosis

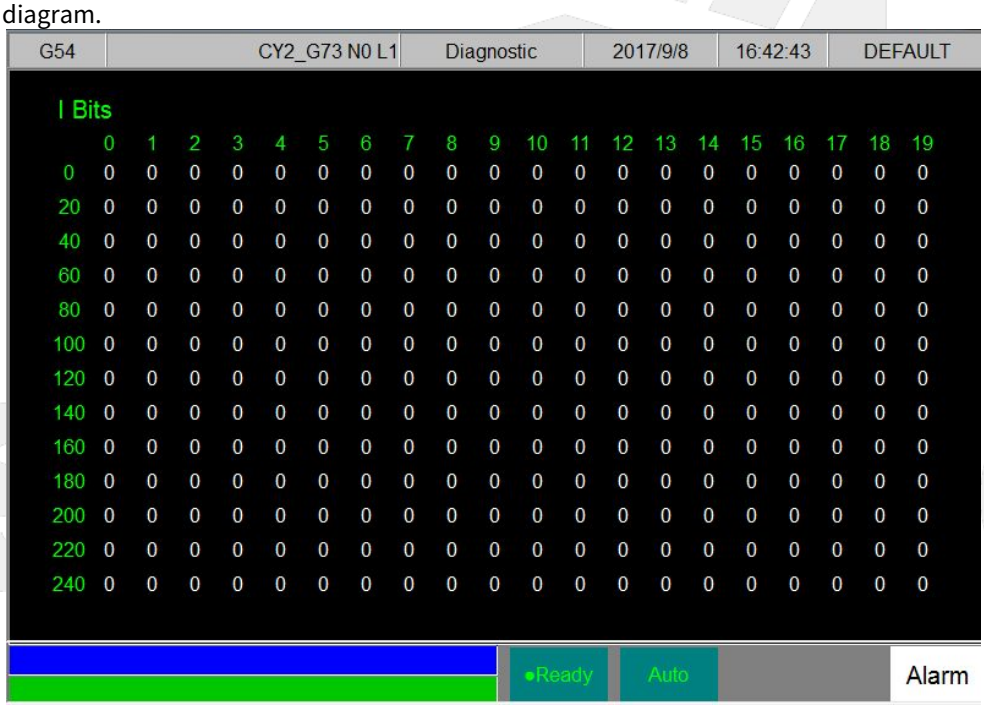
This selection provides users with direct access to the memory area for parameter checking, parameter settings and NC diagnosis function. It can also be used to maintain and debug the control devices. Under the main menu, press F6 and then F2 to access this function as shown in the following figure.



Key Selections:

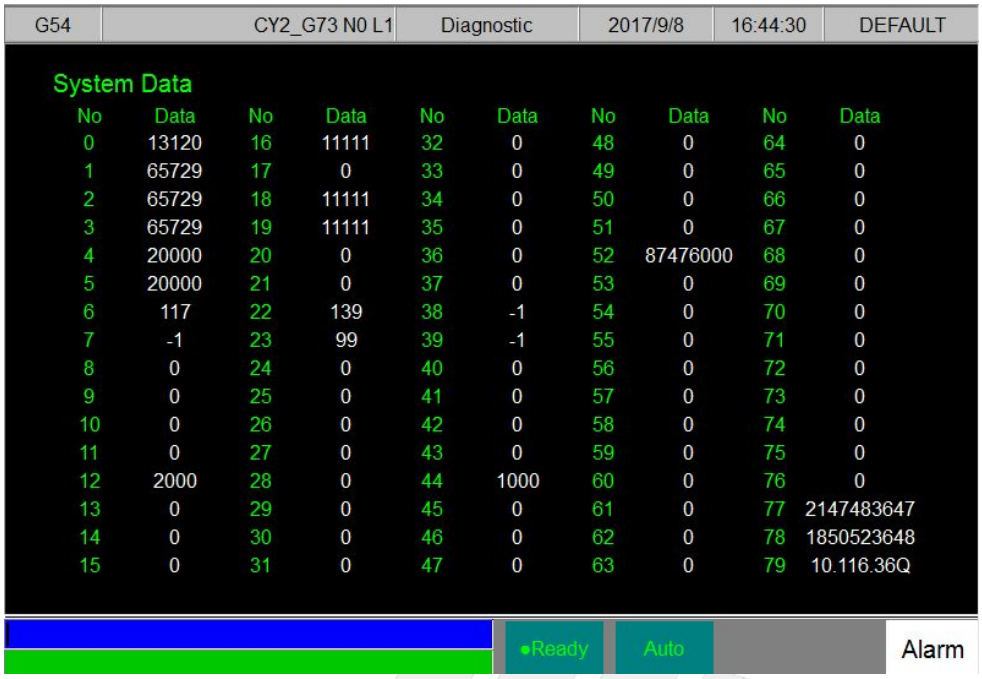
- F1: PLC State

  Display states and information of individual PLC internal-devices such as IOCSA bits, Registers, and Ladder diagram.



- F2: Display System Data

  Function: Display current system data. (For the definitions of each data, refer to the application manual)



- F3: Display Global

  Function: Display the values of Global Data (Global Variables, @ variables). The use and availability of each variable can be found in application manual.

- F4: Display Coord.

  Function: Display the values of Coordinate Data (Local Variables, # variables).

- F8: System Manager

  Function: Manage all system settings. Including backup/restore system, install software updates and so on.

  - F1 Program Install

    Function: Install software update package from external storage.

  - F2 Backup System

    Function: Backup system setting data (parameters, ladder, etc ...) to external storage.

  - F3 Restore System

    Function: Restore system setting data (parameters, ladder, etc ...) from a backup file in external storage..

- F4 Export/Import System Files

  Function: Export/Import individual system setting data and Macros.

1.3.8 F8: About

Function: Display controller information including software version, image version, PLC version, CPU board, serial number, and machine type.

2 Machine Operation Panel

2.1 Operation Panel

This section list out important keys on the control panel. The actual panel can have more keys depending on machine builder.

2.1.1 POWER ON

Turn on main power

2.1.2 POWER OFF

Turn OFF power

2.1.3 Emergency STOP

For safety reason, press this button ,CNC would stop all motor movement and break up all servo drive main circuit to guarantee safety of operator and machine.

2.1.4 Home Mode and Home Function



Description : For Machines with absolute encoder, Home mode has a fast-return function.
Operation :

1. Select HOME mode
2. Set Pr1001~ =1 to enable fast-return funcion for each axis.
3. Press any axis manual key X+,X-,Y+,Y-,Z+,Z-:
4. CNC would move the select axis back to Machine zero with G00 speed(Pr461~).

2.1.5 Continus JOG (Rapid JOG)



Description : User can use this function to move the machine by press JOG key
Operation :

1. Select JOG mode
2. Press axis manual key X+,X-,Y+,Y-,Z+,Z- ,work table would move
3. Operator can use JOG% or G01% adjust Jog federate
4. When operator press manual key and rapid Key "〰" at the same time
5. CNC would move the work table "RAPID speed"
6. Operator can use G00% Rapid Jog federate
   G00 % : Adjust G00 % (F0 .25% .50% .100%)
   G01 % : Adjust G01/G02 /G03 feedrate override %:

2.1.6 Incremental JOG



Description: User can use this function to move the machine by press JOG key
Operation:

1. Select INC JOG mode
2. Press axis manual key X+,X-,Y+,Y-,Z+,Z- ,work table would move a fixed distance
3. Operator can set the incremental distance by G00 rotary switch **1 : 1um *,10 : 10um *,***100 : 100um

2.1.7 MPG



Description: User can use this function to move the machine by MPG(Manual Pulse Generator)
Operation:

1. Select MPG mode
2. Select axis by hand box
3. Select incremental distance
4. Press axis manual key X+,X-,Y+,Y-,Z+,Z- ,work table would move a fixed distance *1 : 1um , *10 : 10um , *100:100um , *1000 : 1000um

2.1.8 AUTO Mode



Description: User use this function to execute NC file

Operation:

1. mode select to AUTO mode
2. After Home function .AUTO mode is available
3. Set workpiece coor.(G54..G59) ,CNC default G54 ,if user don't set any G94..G59 in the NC file
4. Set Tool setting to select tool radius and tool length.
5. Press "START" key to start the NC program.
6. Press "Feedhold" key to feedhold the NC program ,if necessary

2.1.9 MDI Mode Single Block Execute

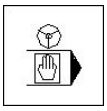


Description: User use this function to execute a block without NC file
Operation:

1. mode select to MDI mode
2. After Home function .MDI mode is available
3. Main function select F4"Monitor"

4. Press F3 "MDI Input" ,screen would pop up a window.
5. After key in date ,Press "ENTER" key to input the data
6. Press "START" key to start the MDI block.
7. If MDI block SYNTAX is correct ,data in MDI menu would disappear

2.1.10 MPG Simulation

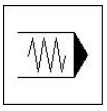


Description: User can use this function to check NC file
Operation:

1. Select AUTO mode
2. Press this button ,and button led ON
3. Press "START" key to start the NC file.
4. CNC would change machine status from "READY" to "BUSY"
5. Machine is still not moving
6. Operator can use rotate MPG to start the NC file
7. MPG rotate more fast ,machining speed is more fast
8. MPG stop, CNC stop too .
9. This function can "Enable" " Disable" immediately

P.S. this function is very user friendly for test runs.

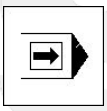
2.1.11 Dry Run



Description: User can use this function to check NC file
Operation:

1. mode select to AUTO mode
2. Press this button ,and button led light "ON"
3. Press "START" key to start the NC file.
4. CNC would change machine status from "READY" to "BUSY"
5. this function can "Enable" " Disable" immediately

2.1.12 Single Block

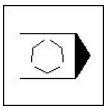


Description: User can use this function to check NC file
Operation:

1. mode select to AUTO mode
2. Press this button ,and button led light "ON"
3. Press "START" key to start the NC file.
4. CNC would execute NC file only one block and STOP
5. CNC would change machine status from "BUSY " to "B_STOP"
6. Press "START" again ,then CNC execute next block

7. This function is for user to check his NC file Block by Block

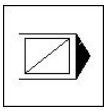
2.1.13 Option Stop



Description: User can use this function to decide NC file M01 is STOP or not
Operation:

1. mode select to AUTO mode
2. Press this button ,and button led light "ON"
3. Press "START" key to start the NC file.
4. When CNC execute "M01" ,CNC would STOP
5. CNC would change machine status from "BUSY " to "Feedhold"
6. This function use to change tool or check workpiece

2.1.14 Option Skip

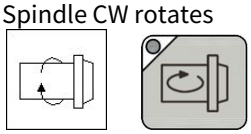


Description: User can use this function to decide NC file '/' skip or not
Operation:

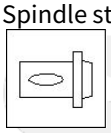
1. mode select to AUTO mode
2. Press this button ,and button led light "ON"
3. Press "START" key to start the NC file.
4. When CNC execute "/" ,CNC would Skip this block
5. If this key is not pressed ,CNC would execute this block

2.1.15 Spindle Control

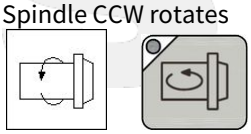
Spindle CW rotates



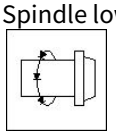
Spindle stop

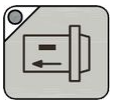


Spindle CCW rotates



Spindle low speed: When spindle is rotating, press this key and then spindle will rotate in low speed



Spindle speed increase by 10%		
Spindle speed decrease by 10%		

SYNTEC

3 How to Operate SYNTEC 200TB

This chapter is written for user task by task ,when user operates this controller ,operator can follow task description as below STEP by STEP ,So very easy to use this controller ,TASK description as below :

- Manual function(JOG ,INC_JOG ,MPG)
- HOME
- Open a file (Program / USB / Network)
- Tool setting (G40/G41/G42 ,G43/G44/G49)
- Tool Length measurement (G43/G44/G49)
- Setting the Workpiece origin offset value(G54…G59)
- Manual Data Input(MDI)
- Assigned an executing NC file (AUTO)
- Graphic Simulation
- How to check NC file in SYNTEC controller
- Network Setting
- Bit Function in PLC Parameters

3.1 Manual Function (JOG ,INC_JOG ,MPG)

When power on SYNTEC CNC, there are 3 mode to manual machine
1st CON_JOG:

1. Release emergency stop button ,CNC status "NOT READY" change to "READY "
2. Mode select switch rotate to JOG mode
3. Press axis direction key(X+,X-,Y+,Y-,Z+…) ,table would move
4. operator can use JOG% adjust JOG speed
5. operator can press axis direction key and rapid key "～ "at the same time ,machine will move by rapid speed
6. rapid JOG speed can be adjusted by G00%

Incremental JOG:

1. Release emergency stop button ,CNC status "NOT READY" change to "READY "
2. Mode select switch rotate to INC JOG mode
3. Press axis direction key(X+,X-,Y+,Y-,Z+…) ,table will move a fixed distance once
4. operator can select incremental distance by G0% (*1,*10,*100)

MPG incremental JOG (MPG):

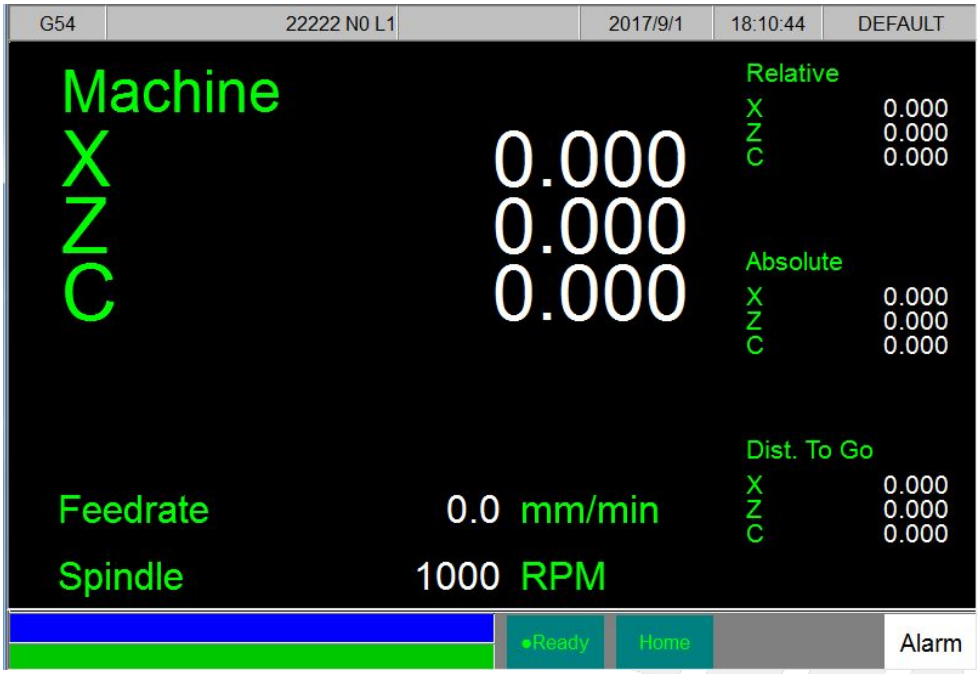
1. Release emergency stop button ,CNC status "NOT READY" change to "READY "
2. Mode select switch rotate to MPG INC JOG mode
3. Select movement axis
4. Select movement distance (*1,*10,*100)
5. Rotate MPG, table would move.

3.2 HOME

Because tool setting, workpiece coordinate setting is based on Machine zero point, it is necessary to make sure machine is homed. Machines with absolute encoders has a fast-retun function under Home mode.
Procedure:

1. Release emergency stop button ,CNC status "NOT READY" change to " READY "
2. Mode select switch rotate to HOME mode

3. Make sure Axis Fast Return Home Pr1001~ =1 to enable fast-retun for all axes.
4. Press axis direction key(X+,X-,Y+,Y-,Z+…) ,axis would start returning Machine home with G00 max speed (Pr461~).
5. Fast-return function can run 3 axis at the same time

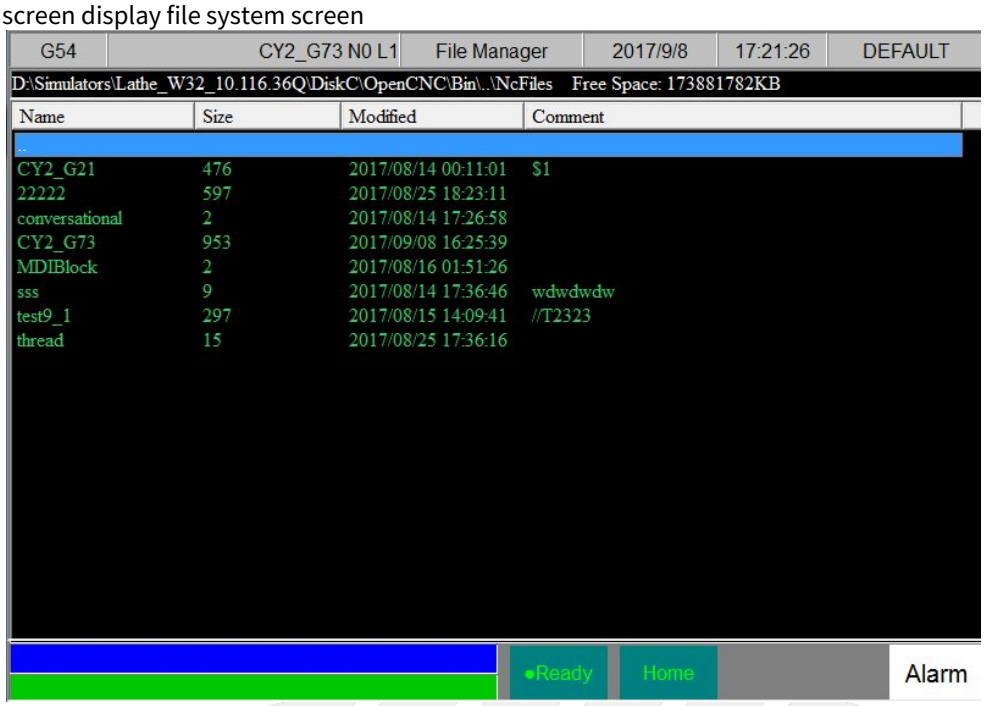


3.3 Open a File (Program / USB / Network)

Procedure:

1. Press F2 "Program" → F8 "File manage"

2. screen display file system screen



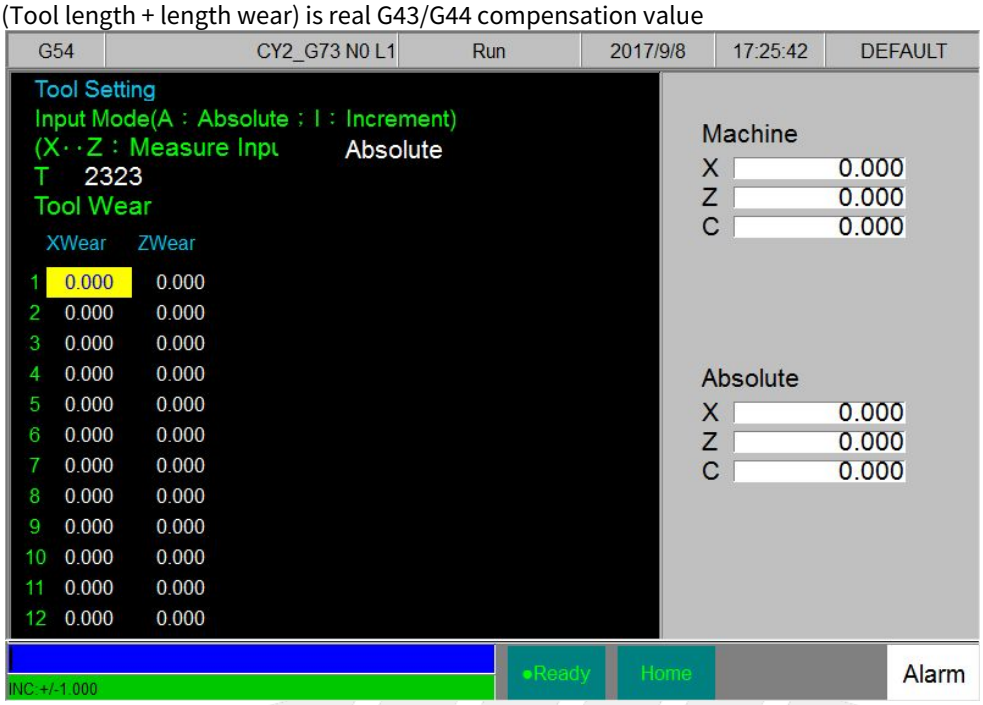
3. Press F1 "New file" to open a new file
4. Press F2 "copy file" to copy current focused file to a new file
5. Press F3 "delete file " and check files top delete.
6. Press F4 "File Transfer" to import/export files from/into a USB or network folder.
7. Press F5 "Execute" to load focused file.
8. Press F6 "Select DNC" to open a NC file from external storage
9. Press F7 "Folder Manager" to manage folders in controller. (10.116.36 or newer version)

3.4 Tool Setting

Procedure for setting Tool offset value :

1. Press Group function key "Monitor"
2. Press function key F5 "tool setting "
3. Move the cursor to the compensation value to be set or change using page keys and cursor keys
4. Type "A" or "I" key to selected input type is "Absolute" or "Incremental"
5. Generally use Absolute type to input Tool radius and Tool length
6. Use Incremental type to input radius wear and Length wear for small value adjust
7. (Tool radius + radius wear) is real G41/G42 compensation value

8. (Tool length + length wear) is real G43/G44 compensation value



3.5 Tool Length Measurement

After NC code select T code, then G43 is enables automatically, so, operator must set tool length data before cutting.

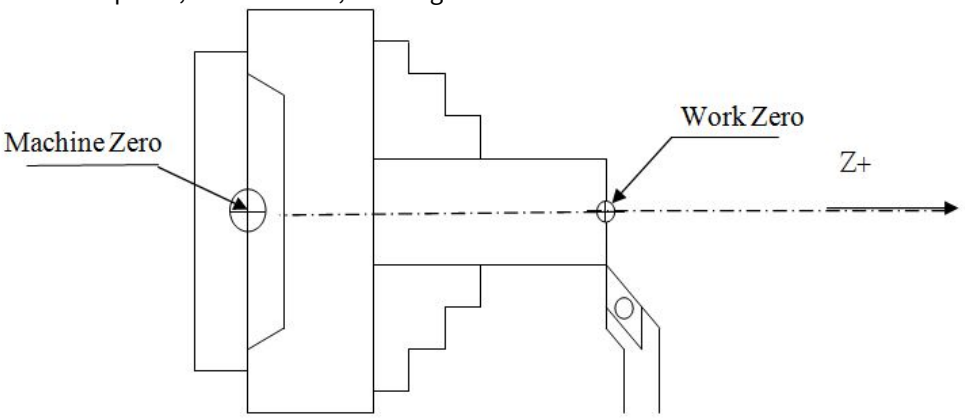
1. Press Group function key "Monitor"
2. Press F5 "Tool setting" to this screen
3. Press F2 "Tool len" to set tool length
4. Move the cursor to the compensation number for the target tool

3.5.1 Z Axis Tool Length Measurement Procedure

1. Use manual operation to move the tool to be measured until it touches the z axis work zero point.
2. There are 3 methods to input z tool length

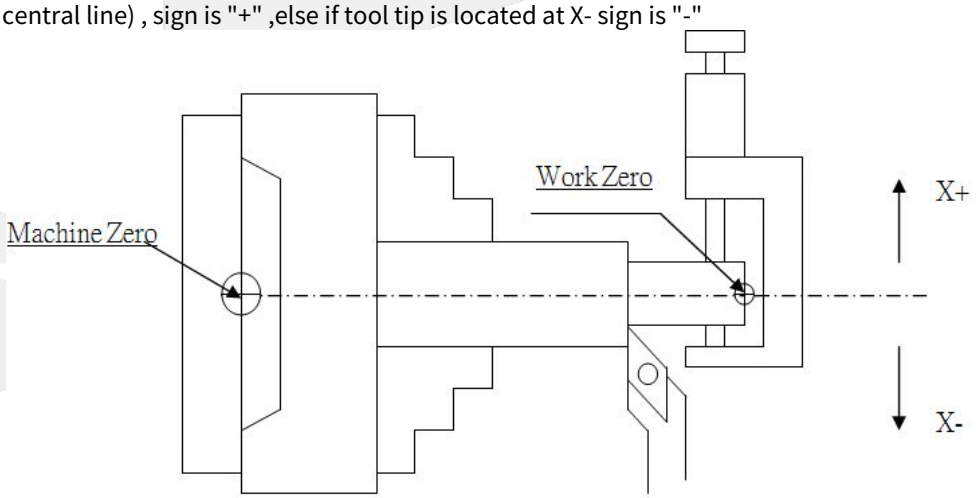
- Absolute : enter work zero point machine coordinate ,by "A*** " ,tool length would be replaced by this value
- Incremental : enter Incremental value ,by "I*** " , use this function to shift current tool length by this value

- Teach : enter current position relative to work zero point by "Z**" ,for example "Z0" means current position is work zero point ,and after this ,tool length would be the same with machine coordinate .



3.5.2 X Axis Tool Length Measurement Procedure

1. Use manual operation to move the tool to be measured until it touches the X axis work zero point.
2. There are 3 methods to input z tool length
    - Absolute : enter work zero point machine coordinate ,by "A*** " ,tool length would be replaced by this value
    - Incremental : enter Incremental value ,by "I*** " , use this function to shift current tool length by this value
    - Teach : enter current position relative to work zero point by "X**" ,for example "X0" means current position is work zero point ,and after this ,tool length would be the same with machine coordinate .
3. Teach by cutting : procedure as below
    - Cut a distance along Z axis
    - Retract tool along Z axis , don't move X axis
    - Measure cutting diameter "D "
    - Input this value by teaching input "X+/-D"
    - The plus/minus sign is decided by tool tip position ,if tool tip is located at X+(tip is located above central line) , sign is "+" ,else if tool tip is located at X- sign is "-"



3.5.3 Tool Wear Setting

Use tool wear setting to adjust cutting dimension, procedures are as below

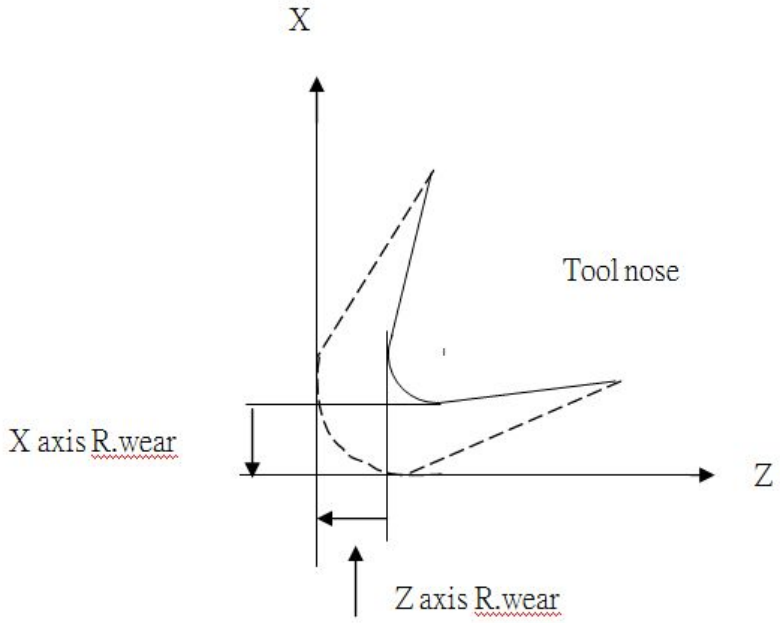
1. After tool length setting ,tool wear value is set to zero automatically
2. measure cutting result ,if there is an error ,use tool wear setting to compensate tool length ,actually tool length = tool length + tool wear
3. There are 2 methods to input tool wear
    - Absolute : enter tool wear absolute value by "A##" , then the tool wear is changed to the entered value
    - Incremental : enter tool wear incremental value ,by "I## " the the tool wear will add the entered value to the current one.
    - Modal setting: only enter "A" or "I" single character, then "absolute " or " incremental" setting would be kept ,after this user can directly input value only .
4. plus/minus sign +/- :
    - Input value can have "+/-" signs.
    - The plus/minus sign is decided by tool tip would be adjusted direction
    - if tool tip is modified to move "Plus" direction ,then key in tool wear value by plus sign"+"
    - if tool tip is modified to move "minus" direction ,then key in tool wear value by minus sign"-"

Example:
If cutting result is 10um oversize in diameter, and tool tip needs to compensate toward X-. Enter " I-0.01 " to tool wear value, then a negative 10um would be added to the next cutting path in X direction.

3.5.4 Tool Nose Setting:

Because tool nose is circular, tool length measurement only measure tool length when precision cutting, use tool nose setting to compensate tool nose dimension error, procedures are as below

1. Press F4 "Run"
2. Press F5 "Tool Setting"
3. Press F3 "Tool Nose Set" to select tool nose compensate
4. There are 3 kinds tool nose data to input Radius : tool nose radius R.wear : tool nose radius wear ,actually the real tool nose is (Radius+R.wear) Nose : tool nose direction ,there are 8 tool nose directions to be elected ,it depends on tool nose shape ,please see SYNTEC programming manual more detail(G41/G42)
5. Use G41/G42 in NC files, to enable tool nose compensation .
6. please see SYNTEC programming manual more detail(G41/G42)



3.6 Setting the Workpiece Origin Offset Value (G54…G59)

Procedure:

1. Press group function key "Position"
2. Press sub selection soft key "Workpiece coor."



3. The screen for displaying the workpiece origin offset values consists multiple pages. Display a desired page by Press PageUp/PageDn key
4. Move the cursor to the workpiece origin offset to be changed
5. "External shift" input the value, which can shift the whole coordinate (G54…G59.8) simultaneous
6. F1"coor. Latched": user can press this function key ,and CNC would latch current machine coordinate to the screen where cursor is located .

Note:

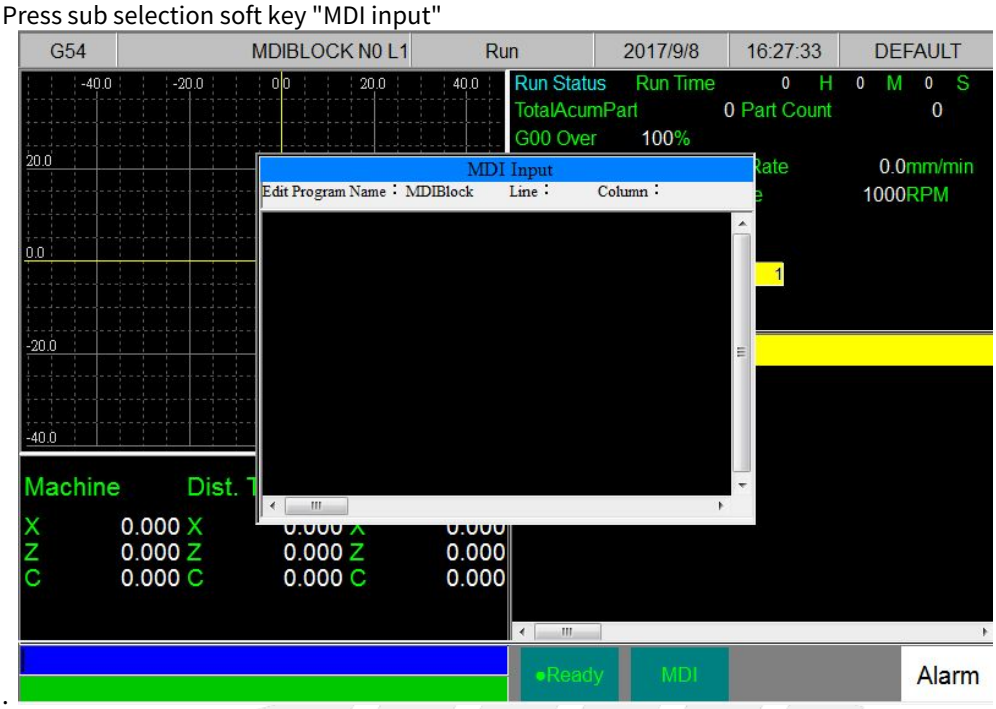
1. This function is enable by setting CNC parameter 3229 : disable workpiece coordinate by " 1 "
2. System default is G54 ,and values are zero every axis
3. After change G54...G59 ,please reset tool length again ,this is very important

3.7 Manual Data Input (MDI)

Procedure:

1. Mode select switch rotate to MDI mode
2. Press group function key "Monitor "

3. Press sub selection soft key "MDI input"



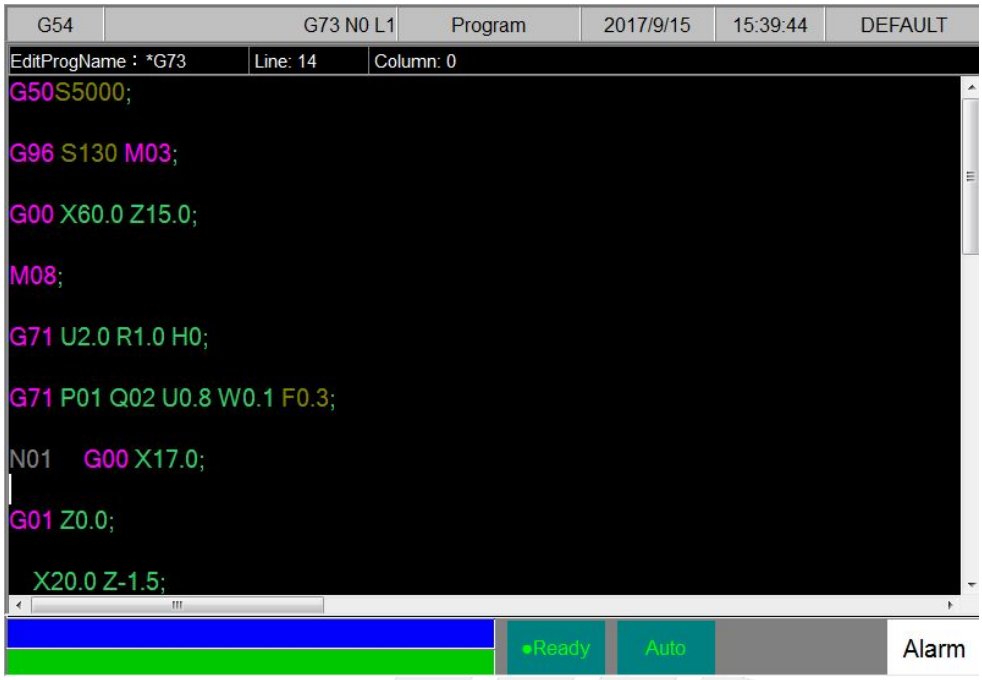
.

4. The screen display the MDI input window
5. Key in MDI data at input bar and PRESS "ENTER"
6. Press 2nd operation panel "START" to execute the current block
7. If current block SYNTAX is correct, the data in the window would be disappeared.

3.8 Assign an Executing NC File (AUTO)

Procedure:

1. Switch to AUTO mode
2. From Main screen press F2 "Program" to open NC file in editor. Or go to F8 "File Manager" to open another NC file.



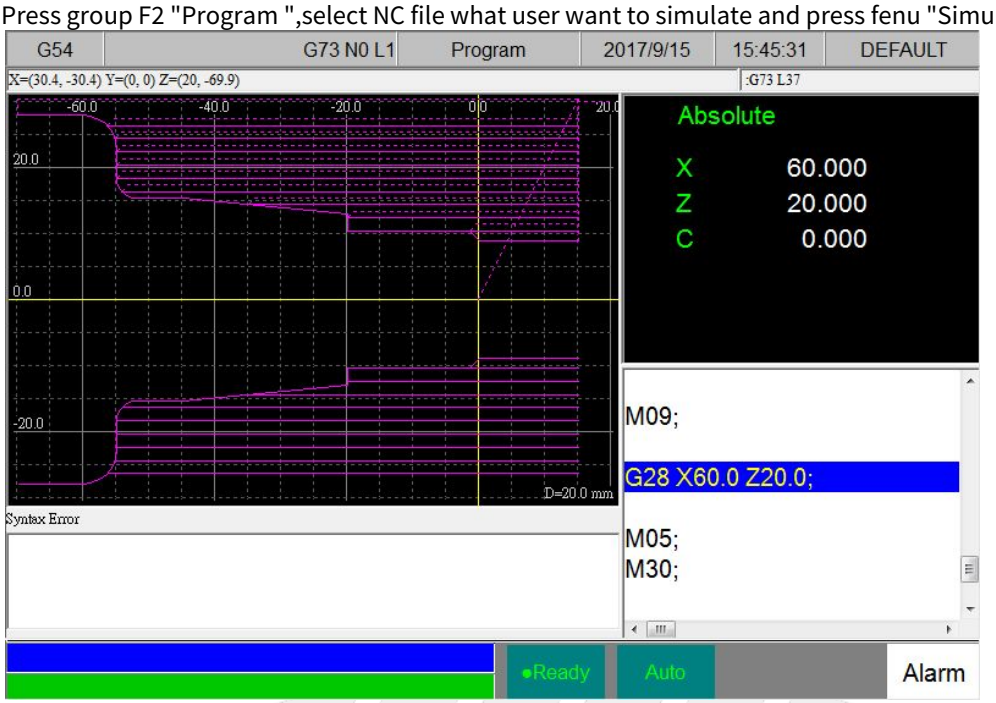
3. Then press F1 "Execute", the NC file is now assigned automatically.



3.9 Graphic Simulation

Procedure:

1. Press group F2 "Program ",select NC file what user want to simulate and press fenu "Simulation"



2. User can use "Step" to check NC file STEP by STEP
3. use "Continue" to simulation whole picture
4. Use "Zoom" check more detail
5. Use "simulation setting" set simulation parameter
6. CNC status is "READY", Press group function key "Monitor" that is the available status to assigned executing NC file.

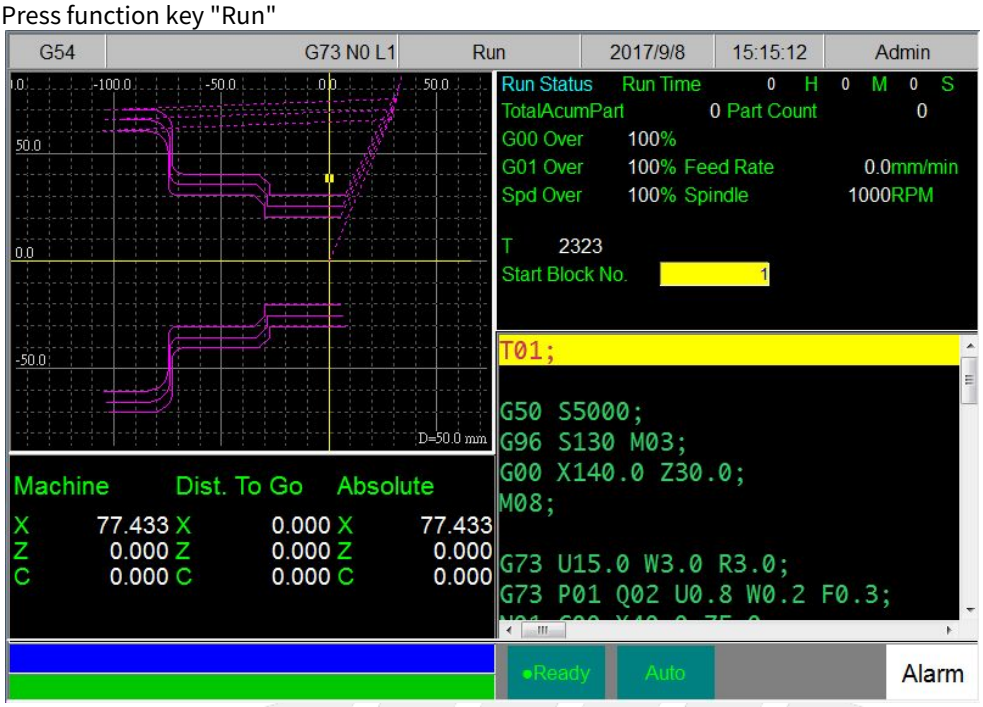
3.10 To Check NC File on Controller

MPG simulation Procedure:

1. Switch to AUTO mode
2. MAke sure status is READY

SYNTEC

3. Press function key "Run"



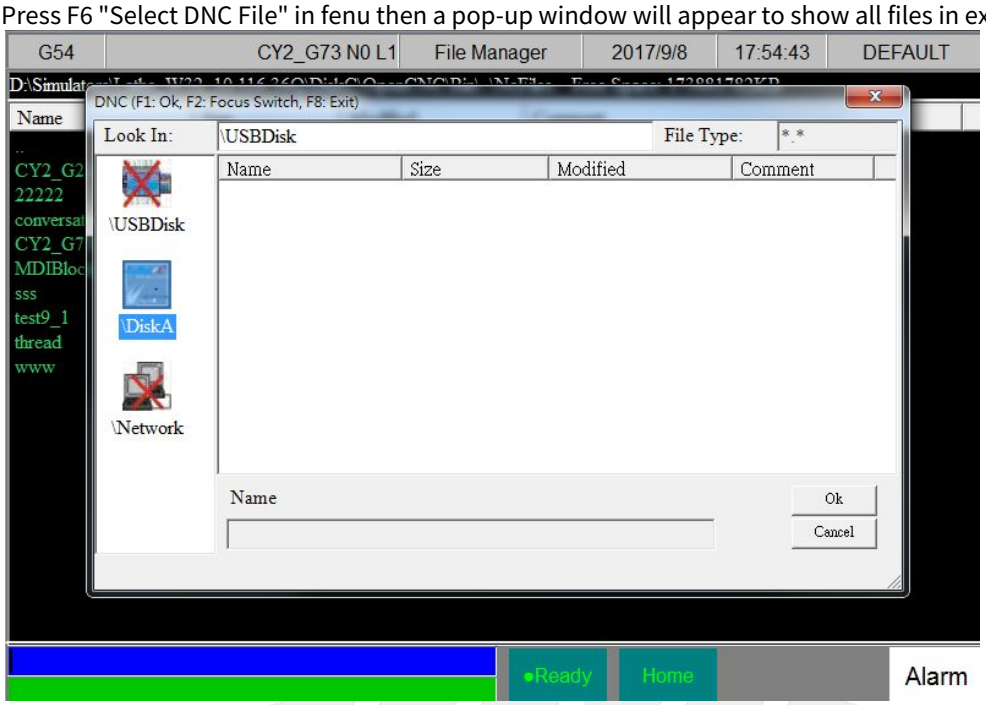
4. Press "MPG simulation "key to enable this function (LED on)
5. Press "Cycle Start "on control panel, CNC status from "READY" to "BUSY"
6. Machine remains static.
7. Rotate MPG handwheel, then the machine starts moving along cutting path
8. The faster MPG wheel is rotated, the fater the machine moves. Machine stops moving if MPG wheel is not rotating. From monitor screen ,operator can see the cutting cursor move along simulation path
9. Operator also can Press "Single Block" key to enable single block function when "MPG simulation " function is ON ,then user can use test two function simultaneous ,check NC file STEP by STEP ,with" MPG simulation " function .
10. "MPG simulation" can control table forward ,also can control backward too ,but NC file would stop at M,S,T code when backward .

3.10.1 DNC Function

DESCRIPTION:
SYNTEC controller can run a program that is saved on an external storage such as USB or network folder. This allows user to save controller memory or save time to transfer large files.
Operation:

1. Insert USB disk or setup network folder on PC
2. Go to main menu, and press F2 "Program", and then select F8 "File Manager".

3. Press F6 "Select DNC File" in fenu then a pop-up window will appear to show all files in external storage.

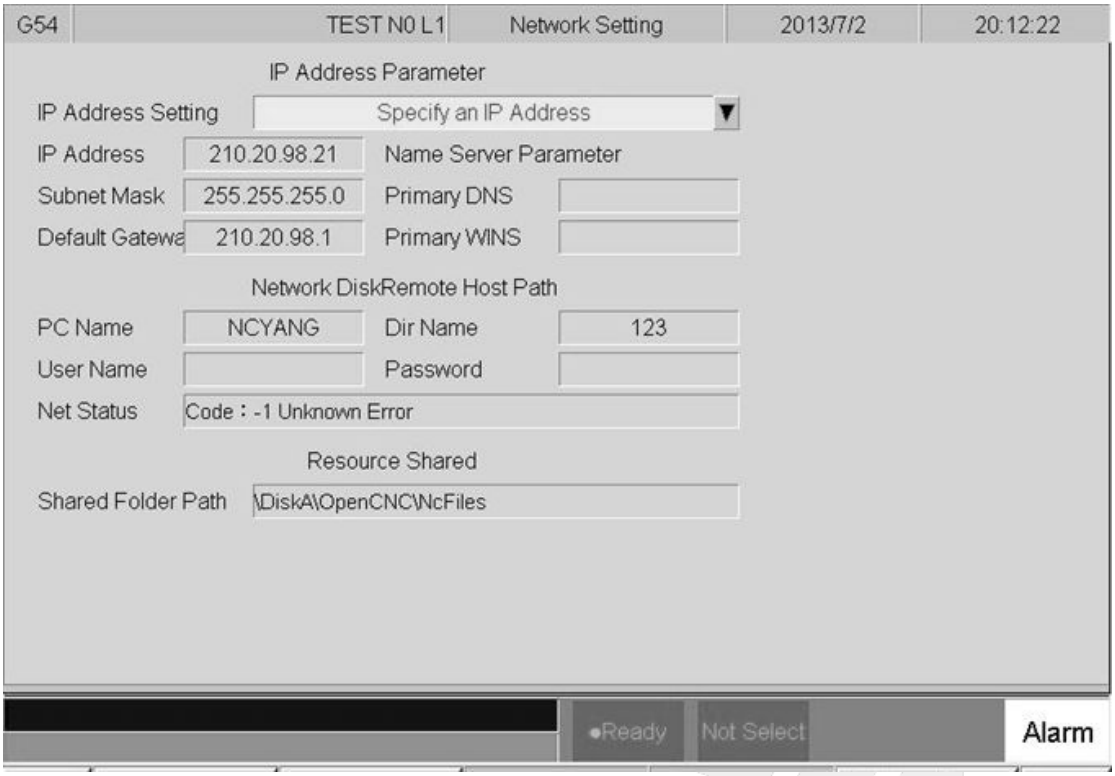


4. Once selected, press F1 to load this file and jump to Run screen.

3.11 Network Setting

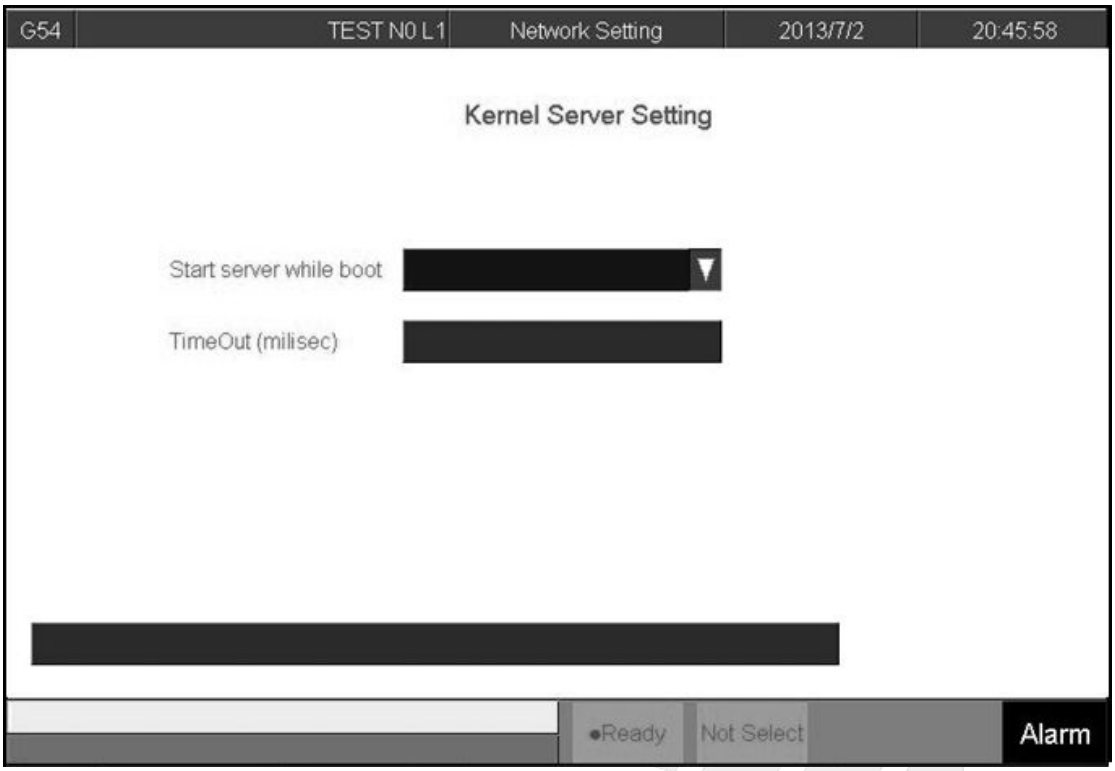
Description: This procedure allows user to setup a file sharing connection between a PC abd SYNTEC Controller. The devices can be connected via direct cable or a switch hub.

3.11.1 Controller Setup



- Path:
  - F5 Maintain->F2 Network setting
- Description
  - Set system network setting
- Related infor.
  - IP address setting
  - Network cable(with HUB), select "Obtain an IP address automatically"
  - Jumper cable(without HUB), select "use the following IP address" and enter IP address(the forth number is different from PC setting) and Subnet mask(same with PC setting)
- IP address
  - Only applied for "use the following IP address" option.
- Sunet mask
  - Enter the IP address for subnet mask (the same with PC subnet mask).
- PC name
  - Enter the full computer name
  - Need to be the same with PC
- Dir name
  - Enter the sharing folder name (the same name with PC sharing folder)
  - User name and password
  - If the shared folder is not set the password, user do not need to enter user name, if yes, please enter the same password.

Set Kernel Server



- Path:
  - F5 Maintain->F2 Network Setting-> F5 Set Kernel Server
- Description
  - Setting function related to kernel server
  - Related infor.
  - Start server while boot
  - Setting whether server is started when booting
  - Timeout(Milisec)
  - Set the acceptable time out when connecting to the Kernel server unsuccessfully

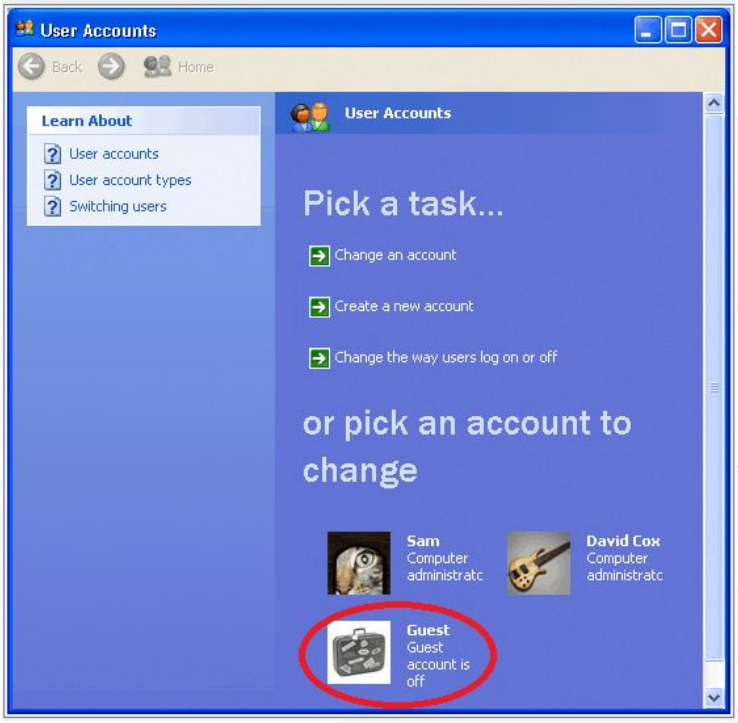
Start Server

- Path:
  - F5 Maintain->F2 Network setting-> F5 Set Kernel Server-> F1 Start Server
- Description
  - Start server immediately

3.11.2 PC Setup

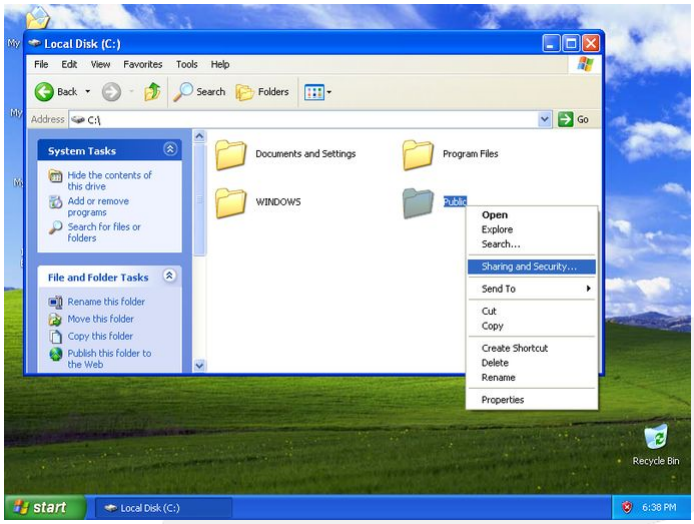
XP OS

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"

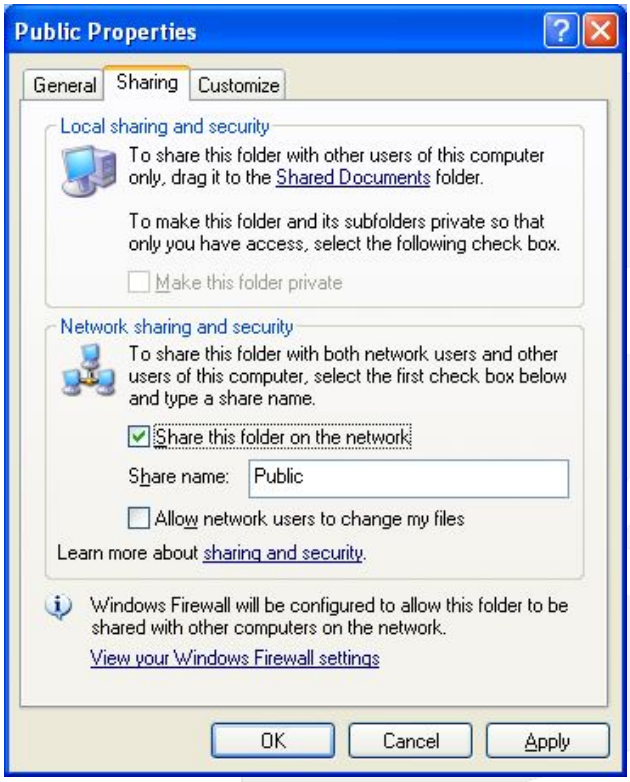
SYNTEC



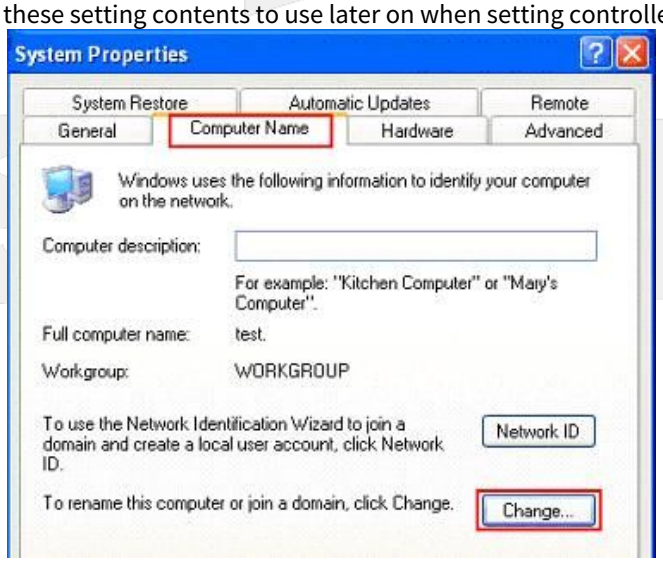


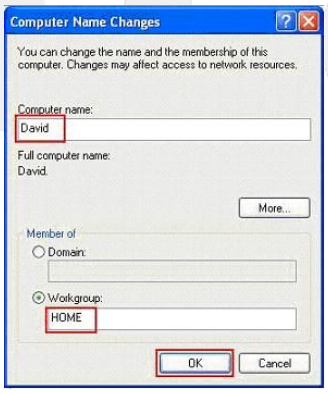
3. Click "OK" to confirm sharing setting; Select "Share this folder on the network", and "Allow network users to change my files".



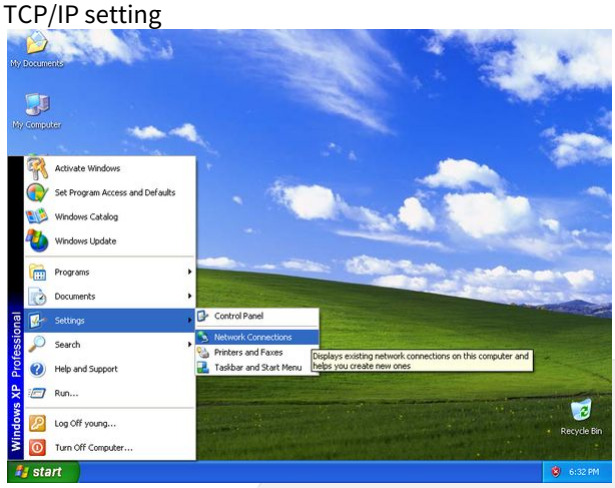


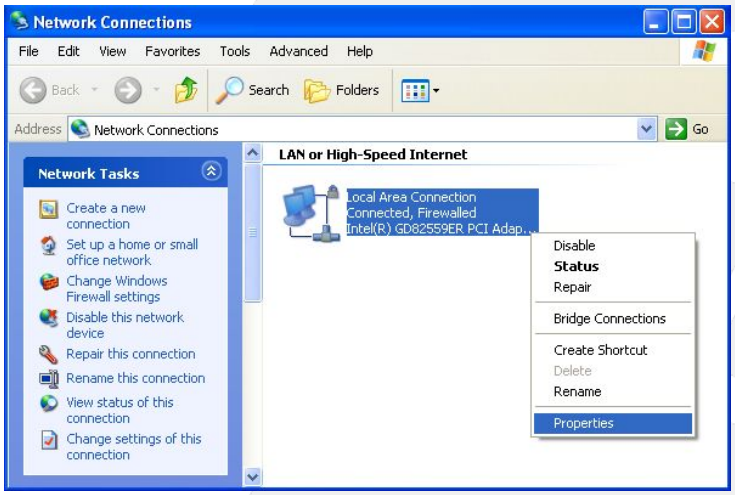
4. Setting PC name and workgroup
"Start"-> "control panel"->"System"-> "change" to set "Computer Name" and "Workgroup", and remember these setting contents to use later on when setting controller.





5. TCP/IP setting

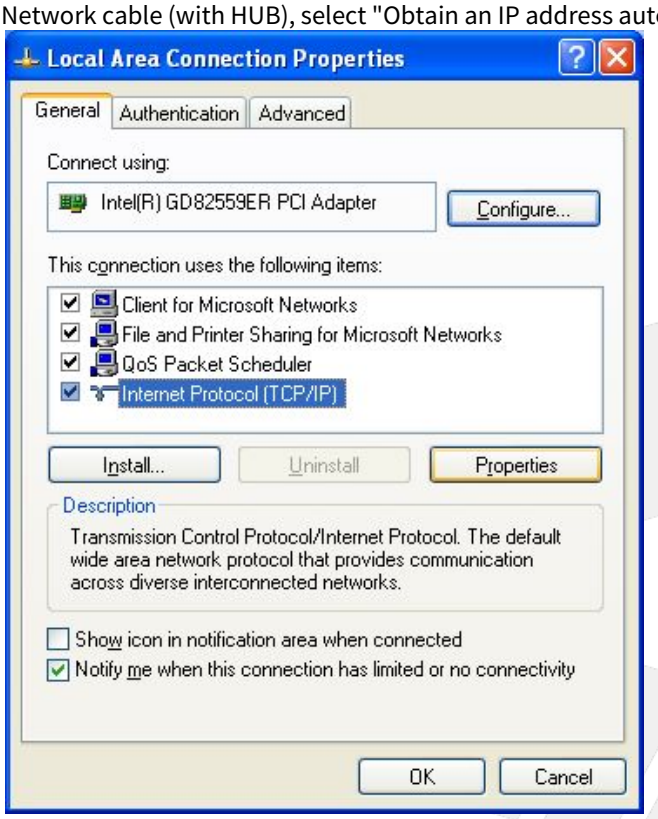


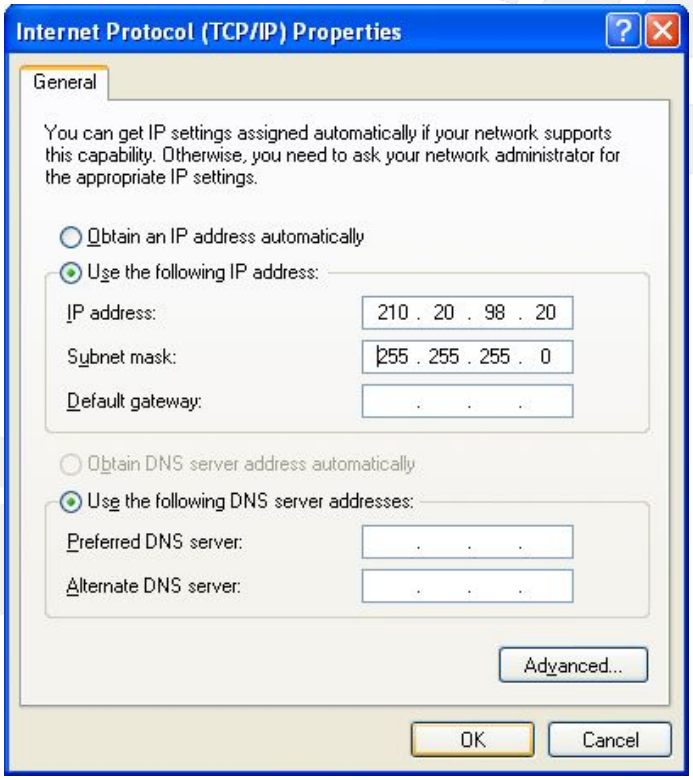


"Start" => "Setting" => "Network connections" and right click on "Properties", and select "Internet Protocol [TCP/IP]"

- 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)

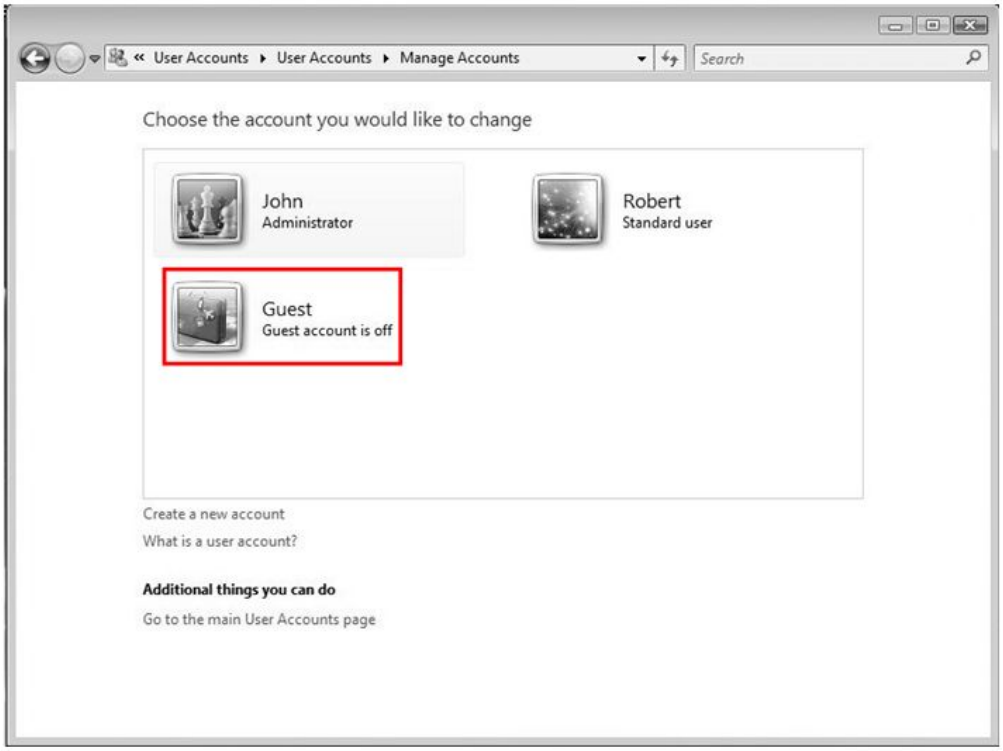
- Network cable (with HUB), select "Obtain an IP address automatically"





VISTA OS

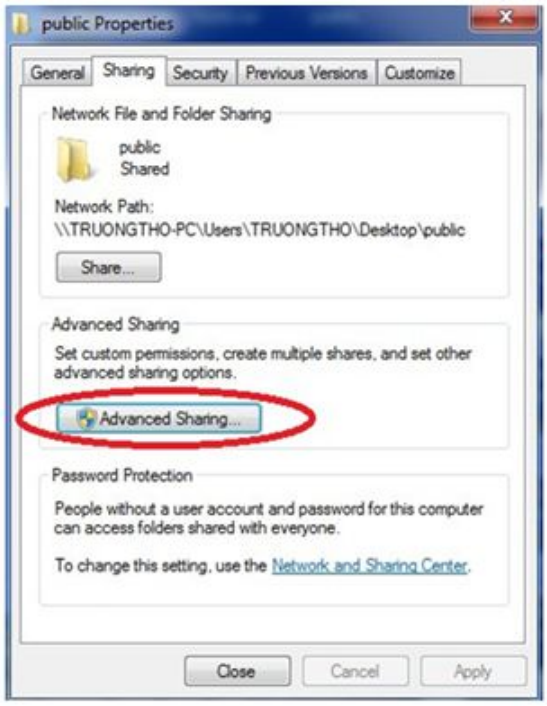
1. Guest account setting
   Log in as Administrator and select "Start" "Control Panel" "User Account" Guest

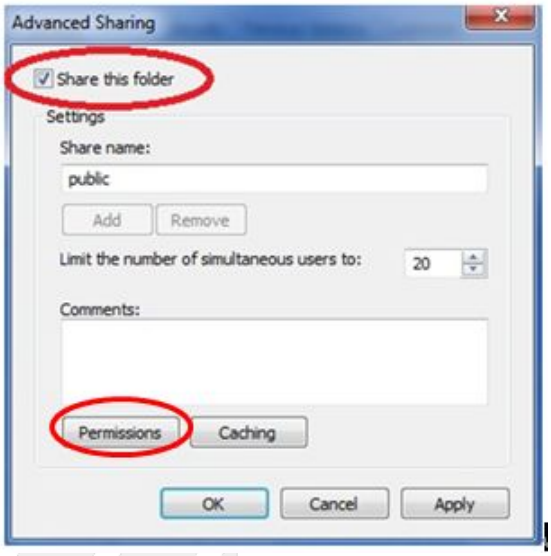


2. Sharing Resource Setting
   Create a sharing folder, and change this folder's setting to offer controller to use, as the below figure.
   a. Click on "advanced sharing"

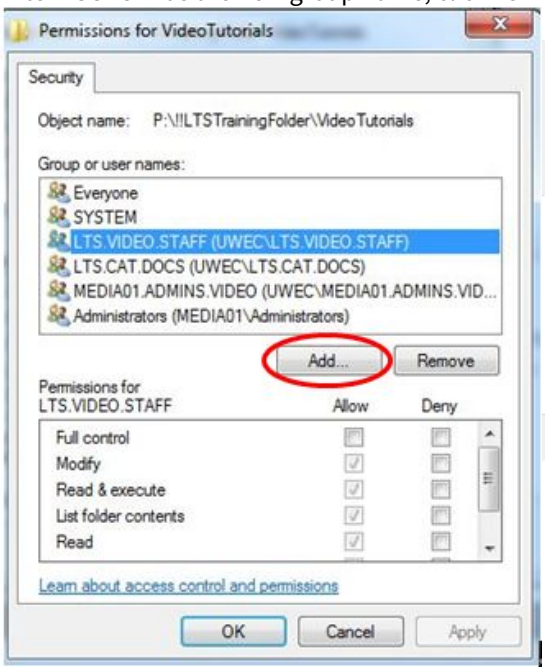
SYNTEC

b. Click on "share this folder"



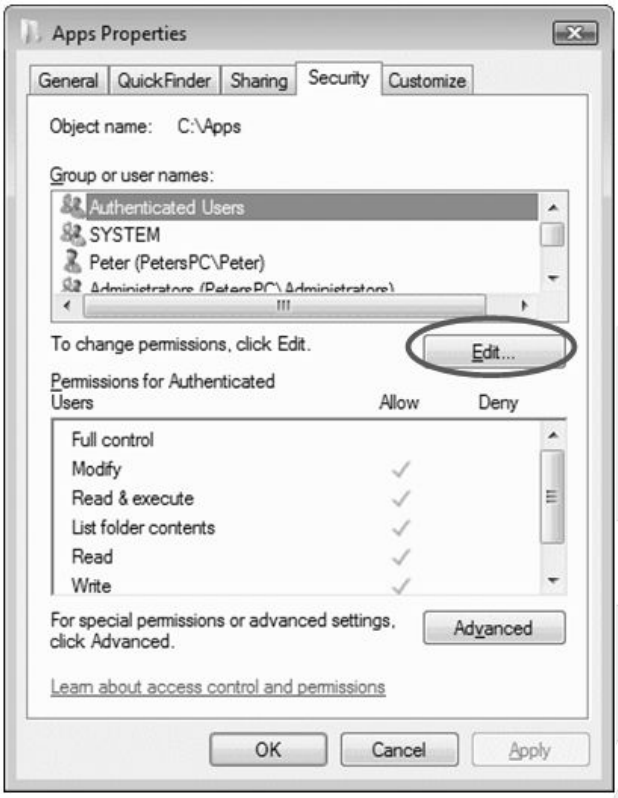


c. Click on "permission"
d. Click on "add"
e. Enter "GUEST" as the new group name, click "OK" to complete setting

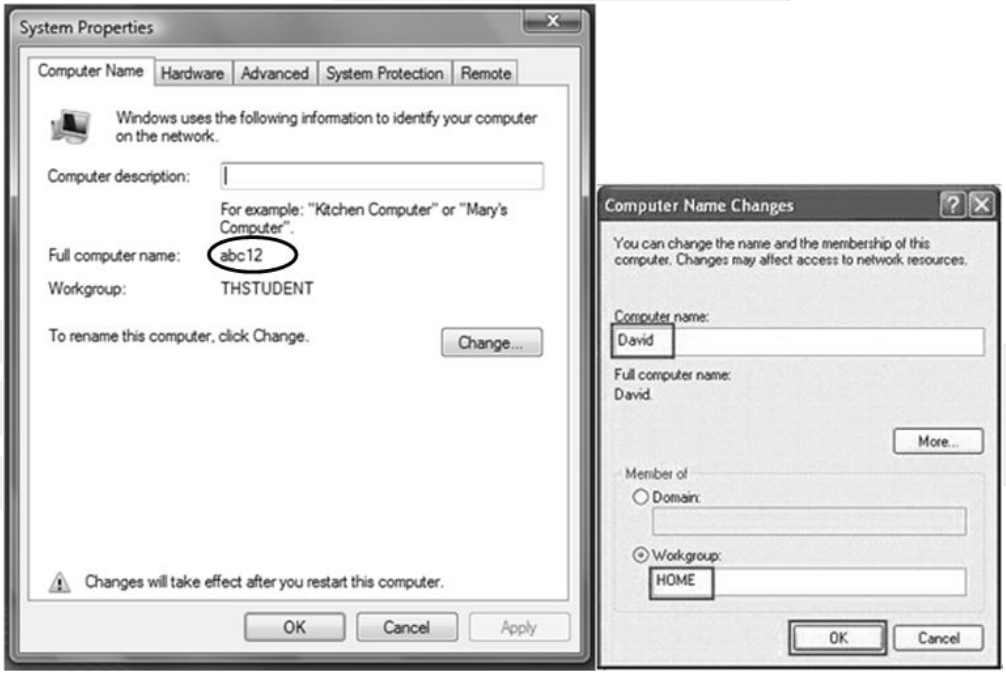


3. Security setting
   Right click on folder to share-> properties-> security-> Edit-> add "Guest" as a new group, then open group

permissions to maximum.

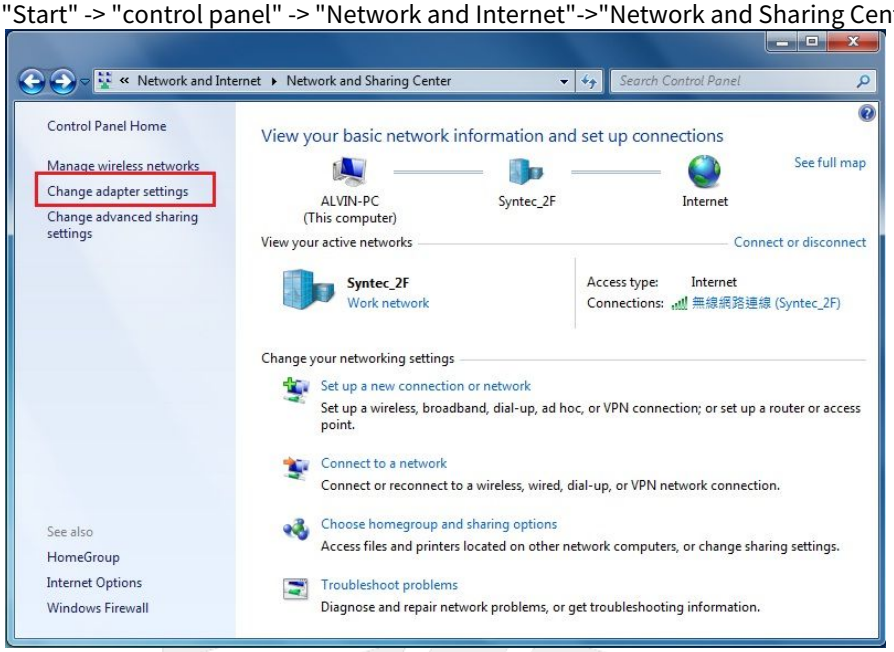


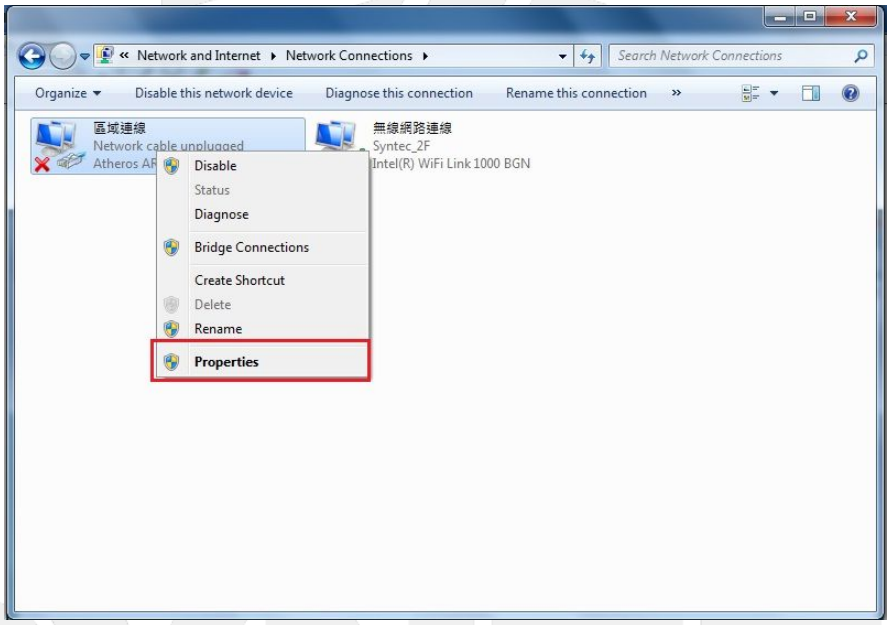
4. Setting PC name and workgroup
"Start"-> "control panel"->"System"-> "change" to set "Computer Name" and "Workgroup", and remember these setting contents to use later on when setting controller.



5. TCP/IP Setting

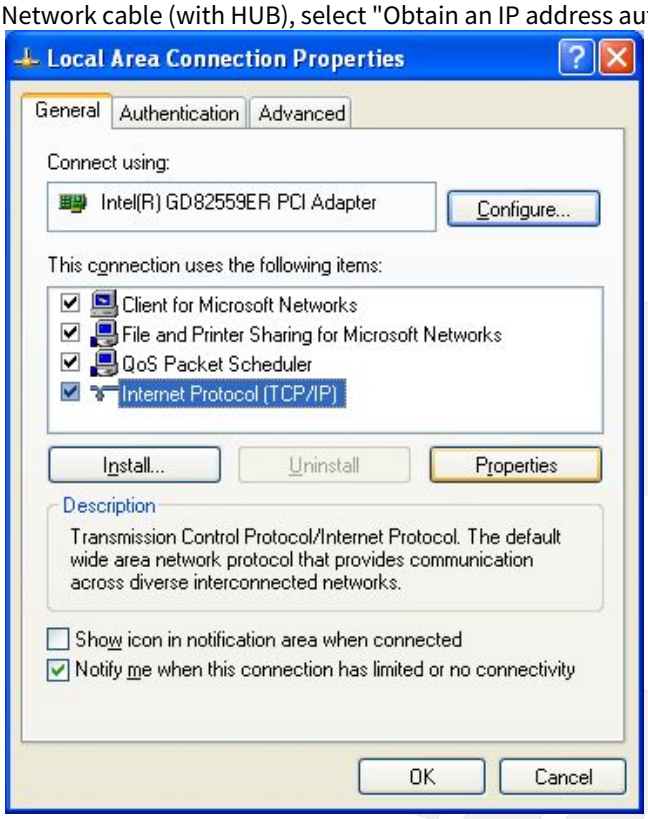
a. "Start" -> "control panel" -> "Network and Internet"->"Network and Sharing Center" -> "Properties"

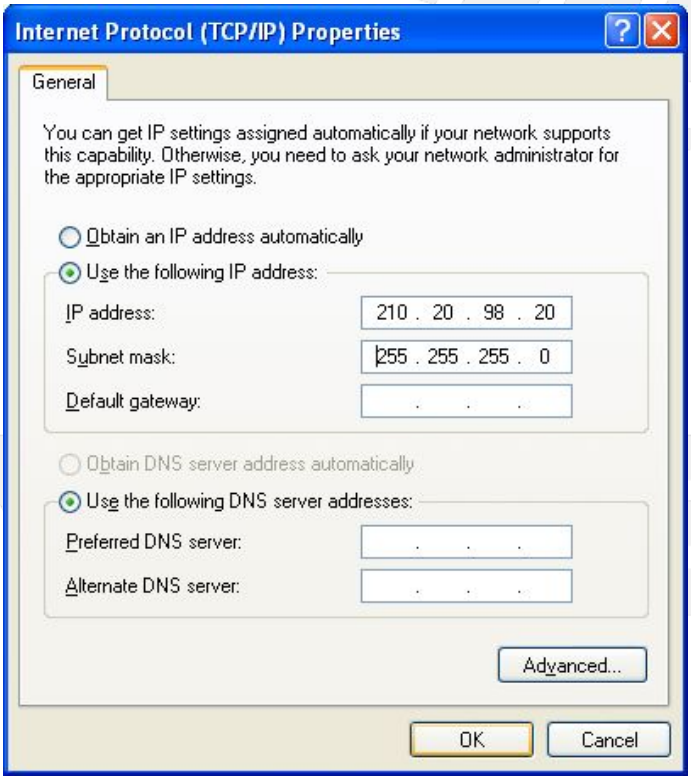




b. Select "internet protocol(TCP/IP)" as shown below:
   - Jumper cable (without HUB), select "use the following IP address" and enter IP address (the forth number is different from controller setting) and Subnet mask (same with controller setting)

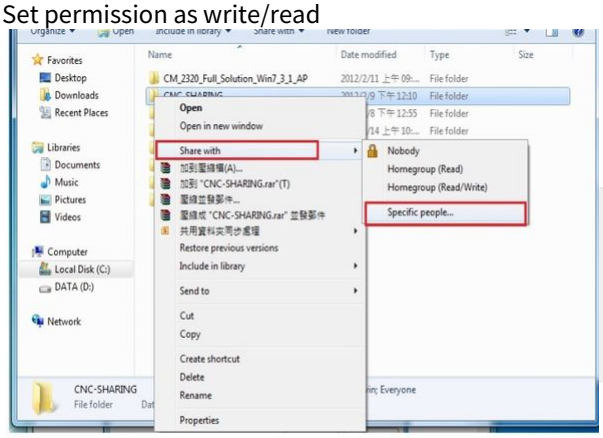
- Network cable (with HUB), select "Obtain an IP address automatically"

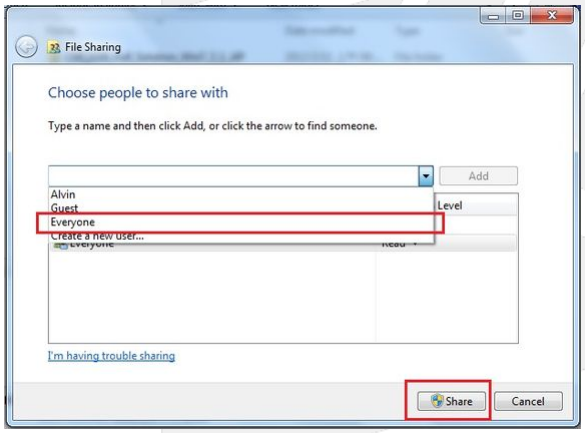




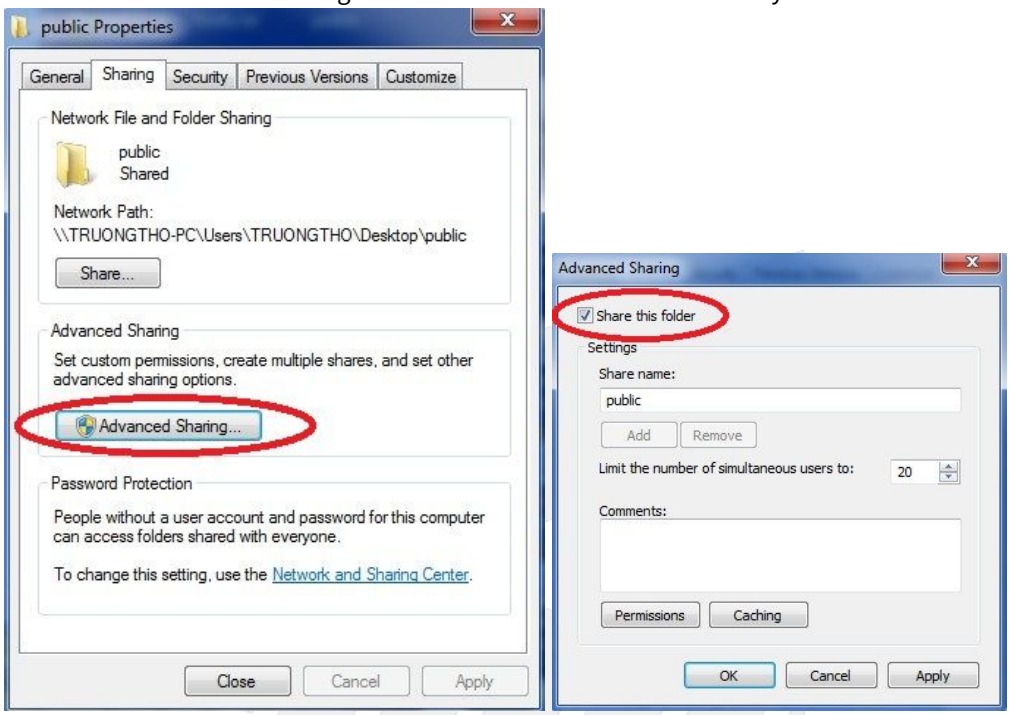
Win 7 OS

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

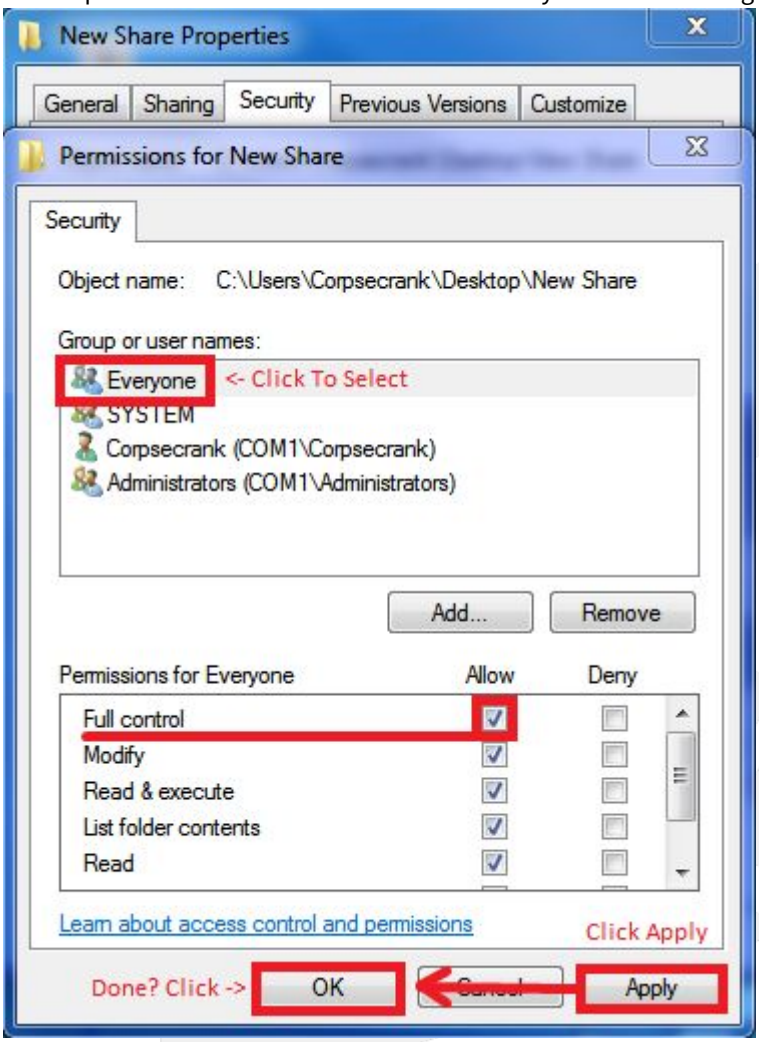




- Left-click on "advanced sharing" and select "share this folder" to everyone.



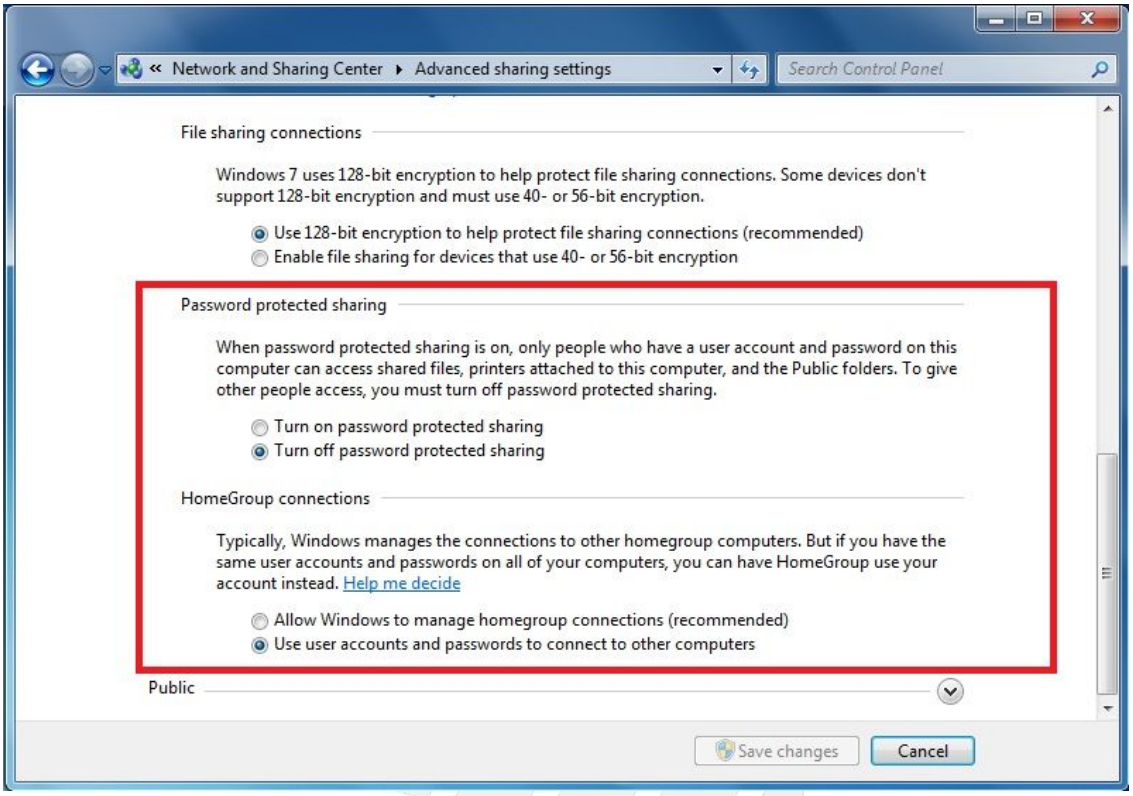
- Select "permission" and select "full control" "only read" and "change"



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


New Share Properties
General
Sharing
Security
Previous Versions
Customize
Network File and Folder Sharing
New Share
Shared
Network Path:
\\COM1\Users\Corpsecrank\Desktop\New Share
Share...
Advanced Sharing
Set custom permissions, create multiple shares, and set other advanced sharing options.
Advanced Sharing...
Password Protection
People without a user account and password for this computer can access folders shared with everyone.
To change this setting, use the Network and Sharing Center.
OK
Cancel
Apply

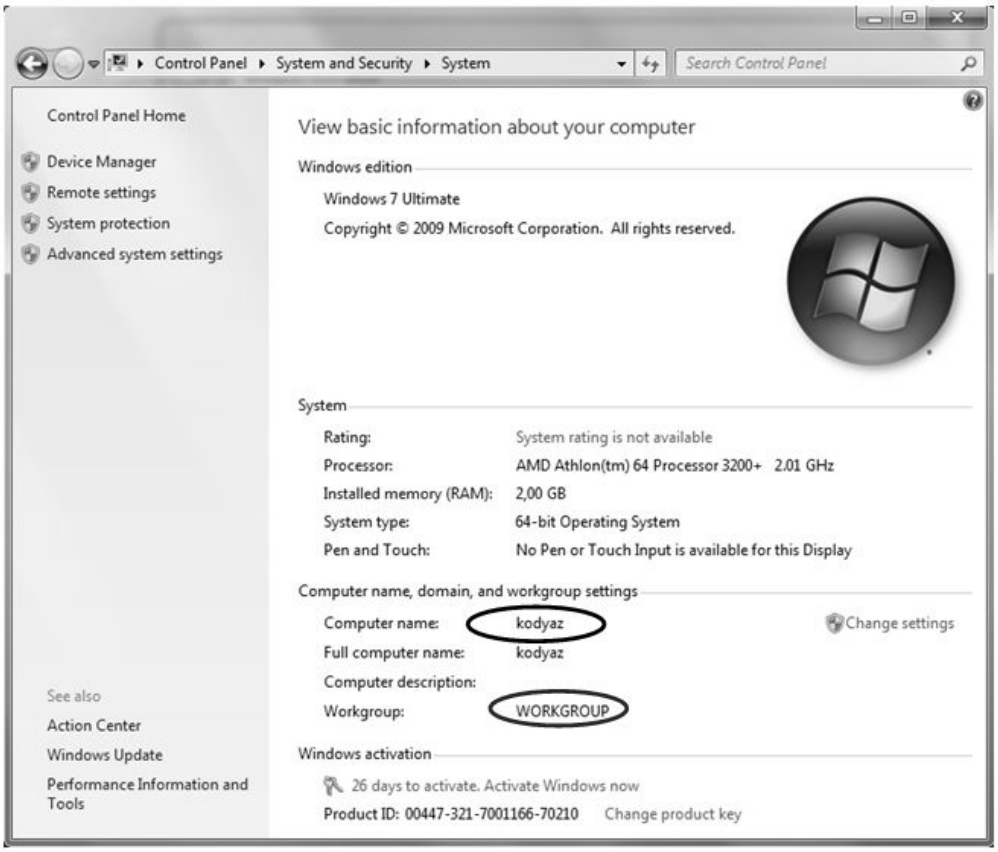
SYNTEC



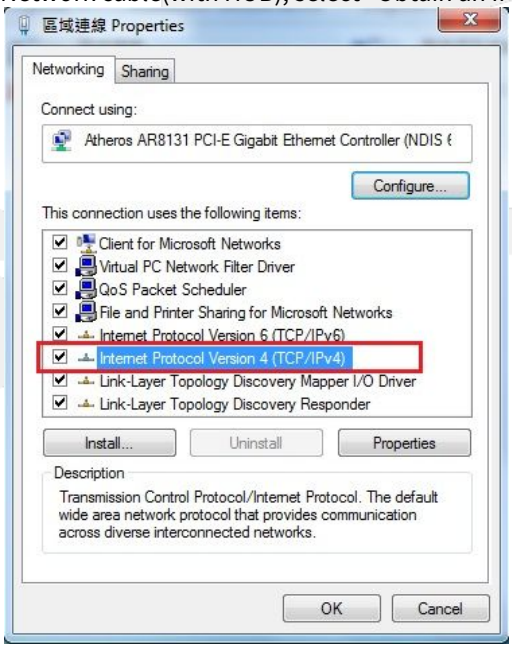
2. Setting PC name and workgroup
   "Start"-> "control panel"->"system and security"-> "System"-> "change " to set "Computer Name" and

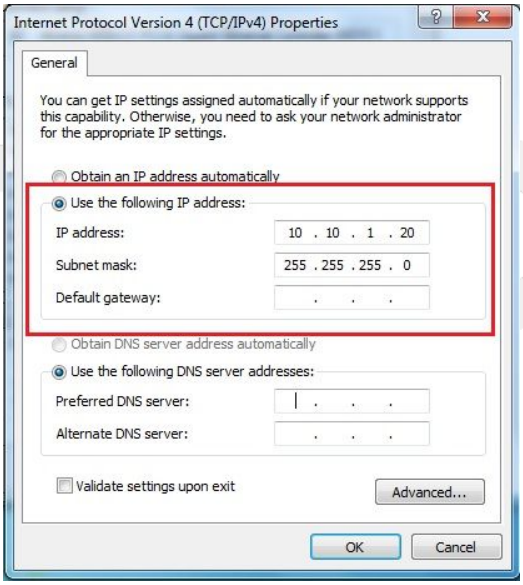
SYNTEC

"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"





3.11.3 Controller FTP

Controller Setup

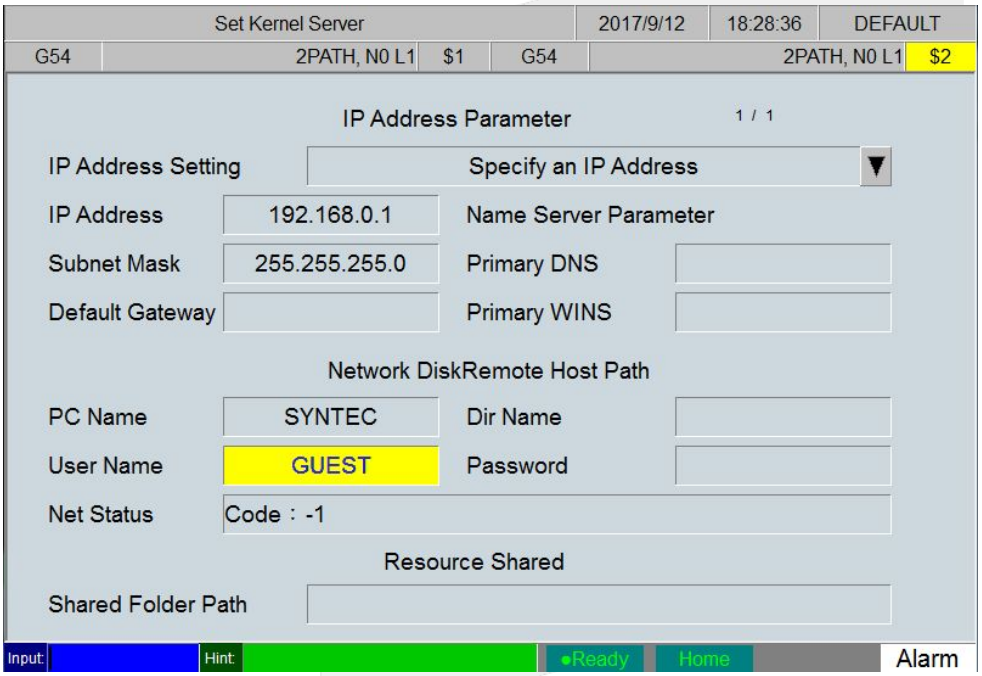
ARM System: F5 Maintain → F2 Set Kernel Server

X86 System: F6 Parameters → F10 Next → F3 Set Kernel Server

Follow the instruction of section above to setup controller's network.

NOTICE:

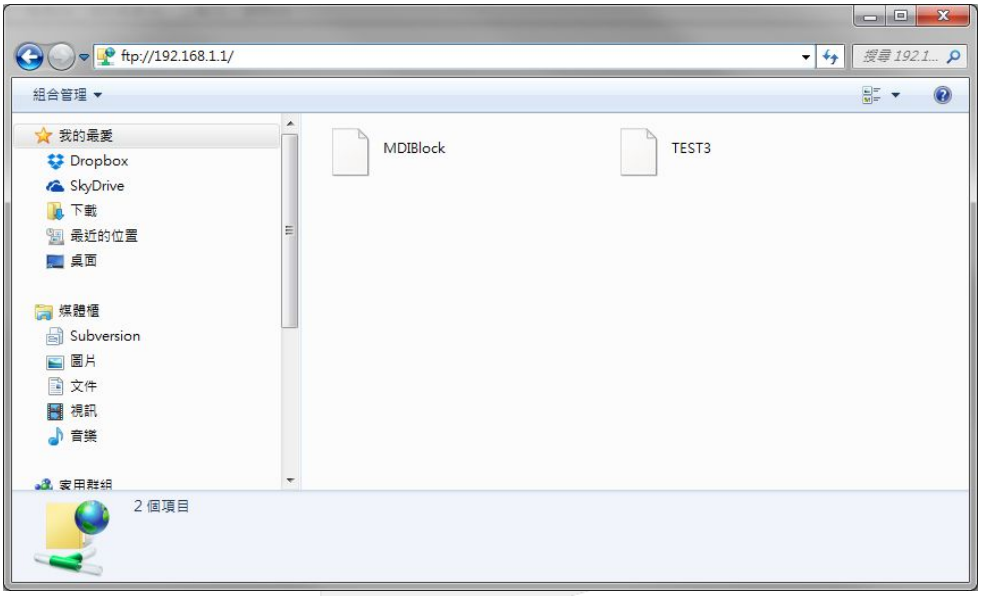
1. make sure to select "Specify an IP Address".
2. Net Status does not have to be "0 No Error/Success" to setup a FTP server.



PC Setup

Open a folder and enter "ftp://controller IP", and then windows will automatically connect to controller's NcFiles folder. All programs can be found here.

SYNTEC


ftp://192.168.1.1/
組合管理
我的最愛
Dropbox
SkyDrive
下載
最近的位置
桌面
媒體櫃
Subversion
圖片
文件
視訊
音樂
MDIBlock
TEST3
2 個項目

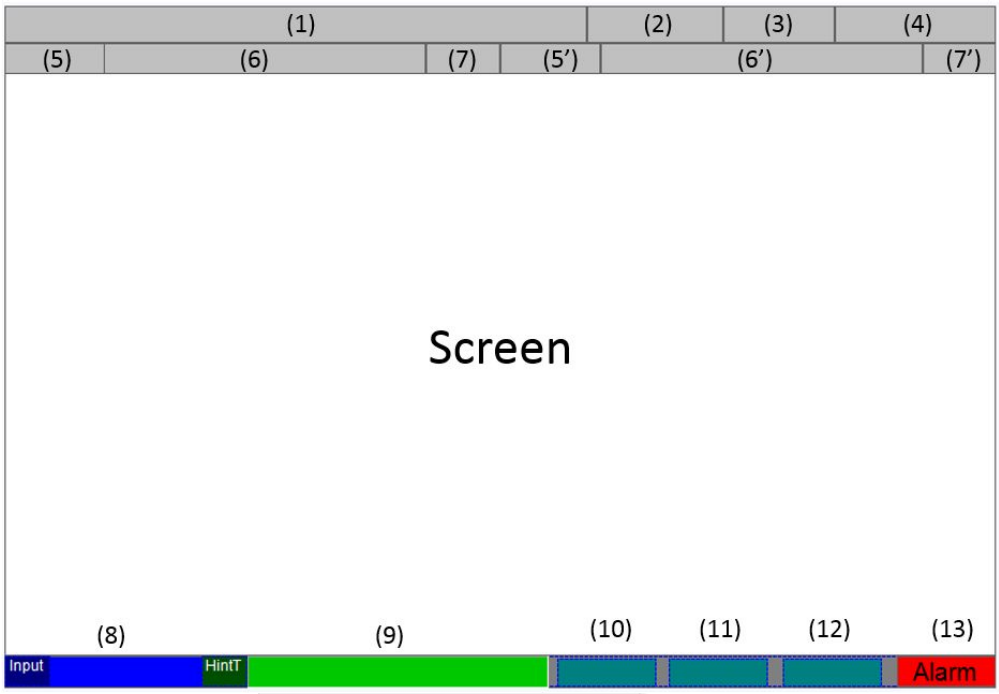
SYNTEC

4 Use of Multi-Path Interface

4.1 Summary

Controller version 10.116.35, 10.117.35 and newer version provide multi-path user interface that display separate path coordinate and machining information. The following is the display

4.2 Screen Layout



e

Description:

- (1) Screen Title
- (2) Time
- (3) Date
- (4) User
- (5) Work Piece Coordinate
- (6) Program Name / Line No.
- (7) Path Label
- (8) Input
- (9) Hint
- (10) Status
- (11) Mode
- (12) Dipole Status (Only visible under Dipole)
- (13) Alarm

4.3 Main Screen

The main screen of different path numbers are show below, press the fenu keys to enter corresponding function screen.

Dual Path



Screen layouts: Left area display 1st path and right area displays 2nd path information.

- X: X axis coordinate
- Y: Y axis coordinate
- Z: Z axis coordinate
- C: C axis coordinate
- Feedrate (F): Speed of machine motion, unit mm/min.
- Spindle(S): Spindle RPM.
- Loading Percentage: Spindle real time load info
- Note: Axes that are NOT within Main System Paths (axis groups) will be displayed in $1 coordinate screen. (ver 10.116.36R, 10.116.47, 10.117.47)
  Axes that are NOT within Main System Paths (axis groups) will be displayed in coordinate screen.ver

Triple Path

SYNTEC



Description:

- Area: 1st, 2nd, and 3rd paths are displayed in areas from left to right.
- Machine Coordinate: Fixed coordinate system referring to machine structure.
- Feedrate (F): Speed of machine motion, unit mm/min.
- Speed(S): Spindle RPM.
- Loading Percentage: Spindle real time load info
- Note: Axes that are NOT within Main System Paths (axis groups) will be displayed in $1 coordinate screen. (ver 10.116.36R, 10.116.47, 10.117.47)

Quad Path

SYNTEC



Description:

- Area: 1st, 2nd, 3rd, and 4th paths are displayed in areas from left to right.Area:1st, 2nd, 3rd, and 4th paths are displayed in areas from left to right.
- Machine Coordinate: Fixed coordinate system referring to machine structure.
- Feedrate (F): Speed of machine motion, unit mm/min.
- Speed(S): Spindle RPM.
- Loading Percentage: Spindle real time load info
- Note: Axes that are NOT within Main System Paths (axis groups) will be displayed in $1 coordinate screen. (ver 10.116.36R, 10.116.47, 10.117.47)

4.3.1 F1: Position

Display all coordinates information of current position. This section explains the difference between multi-path and single-path interfaces.Display all coordinates information of current position. This section explains the difference between multi-path and single-path interfaces.

Dual Path

SYNTEC



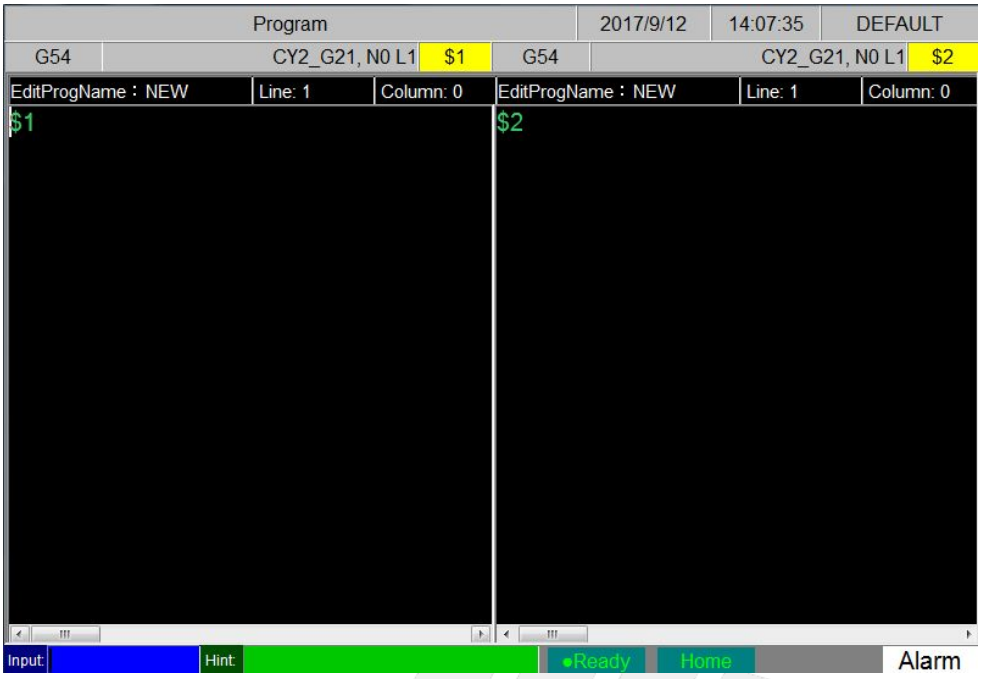
Function Keys are identical to single-path interface:

- **F1： Swithc Coordinate**
  Function: Switch displayed coordinate systemOperation: Under Position screen, press F1 to swtich coordinate display between Machine, Absolute, Relative, and Dist. to Go.
- **F2： Half Coordinate**
  Function: Set the center point of work piece as coordinate origin.
  Operation: Under the Position submenu, when the message line shows "X Input"(or "Y Input" or "Z Input"), press "F2 Half Coordinate" and the relative coordinate value will become half in order to easily calculate center of a workpiece.
- **F3 Clear Coordinates**
  Function: Reset the value of X(or Y or Z) axis relative coordinate to zero.(No effect on other axis)Operation: Under the Position submenu, when "X Input" is entered in the Data Input row (or "Y Input" or "Z Input"), press F3 to reset the value of X(or Y or Z) axis relative coordinate to zero.
- **F4： Clear All Coordinate**
  Function: Set all value of Relative Coord to zero.Function: Set all value of Relative Coord to zero.Operation: Under Coordinate(Position) screen, press F4 to clear all Relative Coord.

4.3.2 F2: Program

The main program editig interface of controller, user can edit all paths in this interface.The main program editig interface of controller, user can edit all paths in this interface.

Dual Path

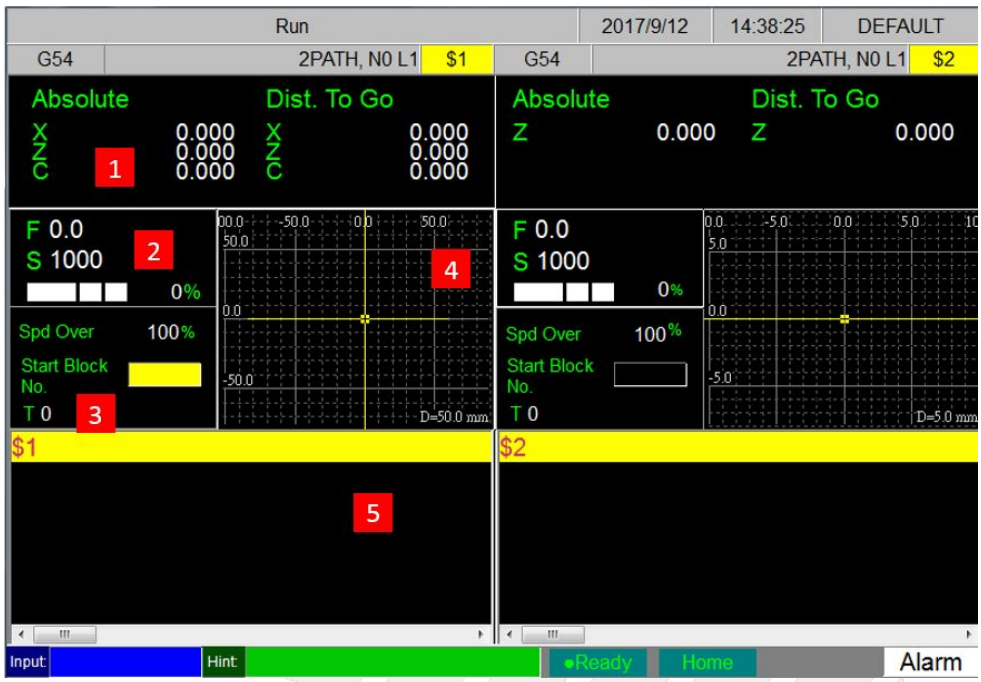


- **F1: Execute**
  Function: Load the current program and jump to Run Menu
  Operation: Under Program submenu, press F1 to load current program and jump to Run menu.
- **F2: Delete Line**
  Function: Delete a line at cursor position.Operation: Under Program submenu, press F2 to delete a line where the cursor is located.
- **F3: Can Cycle**
  Function: Enter Can Cycle fenu to add or edit a cycle.
  Operation: Under Program submenu, press F3 to enter Can Cycle fenu and select Insert Cycle or Edit Cycle.
- **F4: Select Program**
  Function: Switch cursor between paths to edit programs.
  Operation: In Program submenu, press F4 to switch cursor between programs.
- **F5: Half/Full Screen Switch**
  Function: To switch single path display or dual path display.
- **F6： Sub Edit Menu**
  Function: To perform edit functions such as edit, copy, paste, search, and so on.
- **F7: Simulation**
  Function: Simulation program can verify the accuracy of the editing program.
  Operation: Under Program submenu, press F7.
- **F8: File Manager**
  Function: To manage all programs including copy, paste, import/export, and so on.Description: To manage all programs including copy, paste, import/export, and so on.

4.3.3 F4: Run

Displays Coordinates, Program, Simulation, Feedrate, Run Time, MDI, and all useful information during machining.Displays Coordinates, Program, Simulation, Feedrate, Run Time, , and all useful information during machining.

Dual Path



Screen Description

1. Coordinates
    - Note: Axes that are NOT within Main System Paths (axis groups) will be displayed in $1 coordinate screen. (ver 10.116.36R, 10.116.47, 10.117.47)
2. Spindle RPM & Spindle Load, Feedrate.
3. Spindle Override %, Start Block Number, Tool No.
4. Simulation Display
5. Program Display

Auxiliary Functions

1. Cursor at Start Block will switch to active path automatically.
    - Version: 10.116.36J、10.116.42、10.117.42
    - Description:
        - When only one path is activated, cursor will switch to the Start Block of the actived path.
        - When multiple paths are activated:
            - Cursor does not move position if current path is also activated. (E.g. Cursor is currently at $1, if $1 & $2 are activated, cursor is still at $1.)
            - If activated path does not include the path that cursor is currently at, then the cursor will jump to the first path among newly activated paths. (E.g. Cursor at $1, if $2 & $3 are activated and $1 is not, then cursor moves to $2)
        - Cursor remain its position if screens are switched.
2. Jump to Line
    - Description:
        - Enter the content to search in Start Line box, and it will display content while searching.
        - If N# (sequence no.) is entered, the N numer in all activated paths will be found.
        - If number, L#, string, G code, M code are entered, system will only search the path which the cursor locates.
3. Additional Jump to Line Function
    - Version: 10.116.36P、10.116.45、10.117.45

- Description:
  - Use slash "/" to separate the contents to search in different paths.
  - Example: Entering "N1/L2" means seach N1 in $1 and L2 in $2.
  - Example: Entering "/L2" means seach L2 in $2 only.
  - To seach contents in n pathes at the same time, n-1 "/" must be used. It is not irrelevant to number of activated paths.
  - If no slash "/" is entered, then suystem regard it as regular jump to line operation.
- Compatibility:
  - Can no longer seach strings with "/" symbol.
  - If some content is not found, the message will appear after all paths finish searching.
  - The not found message will specify the path number.

4. Refer CNC and PLC Interface section of Application Manual for the definition of R21.

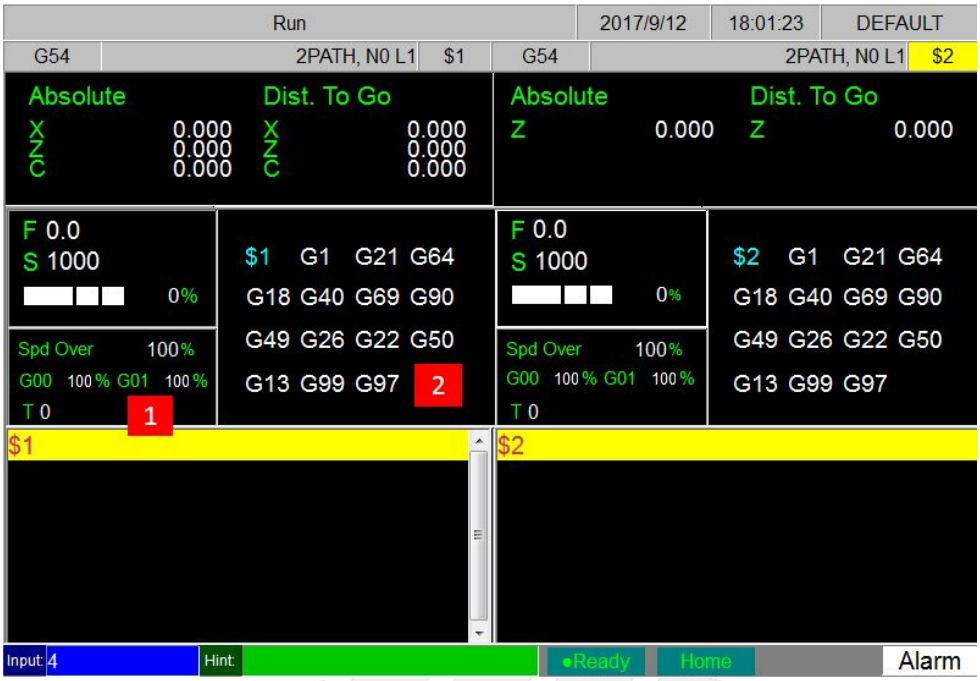
Function Keys

- **F1: Edit**
  Function: Open the currently loaded program to edit.
  Operation: Under Run submenu, press F1 to open this program and jump to Edit screen.
- **F3: MDI Input**
  Function: Manual Data Input. MDI window can display multiple paths in split window, the regulation is:
  a. When R21.11 is OFF:
     - R21=0: displays two paths
     - R21=1: displays 1st path only
     - R21=2: displays 2nd path only
     - R21=3: displays 3rd path only
     - R21=4: displays 4th path only

  b. When R21.11 is ON:
     - R21.1 on: displays 1st path only
     - R21.2 on: displays 2nd path only
     - R21.3 on: displays 3rd path only
     - R21.4 on: displays 4th path only
     - The cases above are mutual exclusive, MDI split window **will display all paths** if more than one bit is ON.
- **F4: Parameter Set**
  Function: Switch screens to display various machining information.

  - Press F4 to switch to screen below

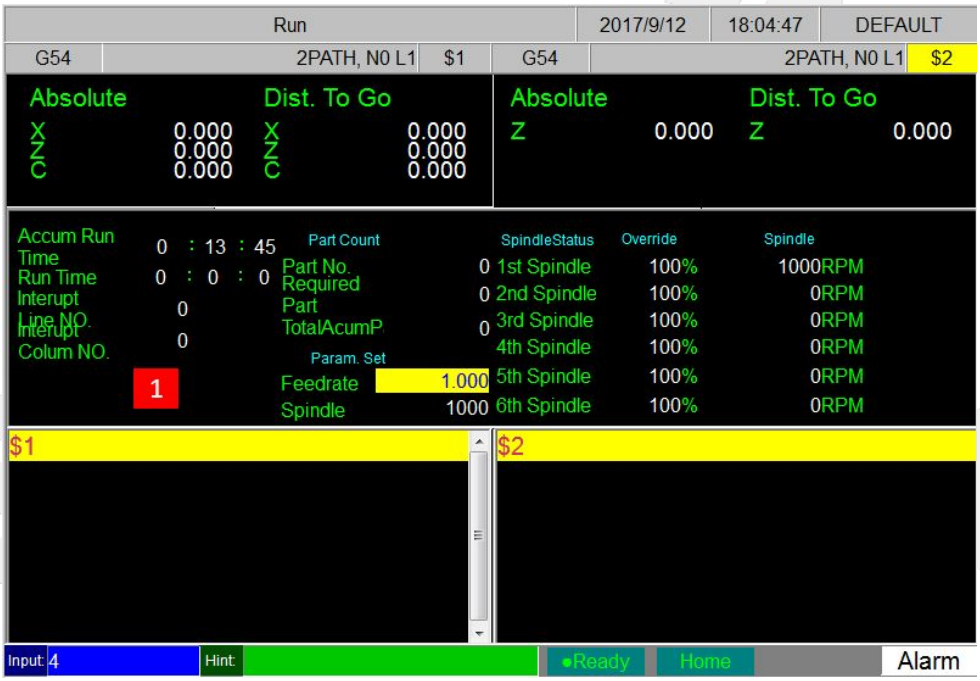
SYNTEC



Display Devices:

a. Spindle Override %, G00 & G01 Override %, Tool No.
b. Modal G Codes

- Press F4 again to switch to ext screen



Display Devices:

a. Displays mutual information of a paths. Including total run time, interrupted line no., part count, and etc. It also shows spindle load status.

- Press F4 again will return to the initial Run screen.

- **F5: Tool Set**
  Function: Display tool dimension setup including Tool Length, Tool Wear, Tool Nose.
  Operation: Identical to Single Path interface
- **F7: Work Record**
  Function: Shows the table of work record, including program name, run time, total part count, and comments.
  Operation: Under Run submenu, press F7 to open Work Record. NOTICE: This is different from Single Path interface.
- **F8: Clear Accumulated Cycle Time**
  Function: Clear all accumulated run time.

SYNTEC

5 Appendix

5.1 Contact Window

Hsinchu SYNTEC TECHNOLOGY CO., LTD. (Headquarters) No. 21 Industry E. Rd. 4, Hsinchu Science Park, Hsinchu City 30077, Taiwan, ROC TEL：+886-3-6663553 FAX：+886-3-6663505 E-Mail： sales@syntecclub.com.twcustomerservice@syntecclub.com.tw	Los Angeles, California, USA Syntec Technology Inc 1370 Valley Vista Dr Suite 200, Diamond Bar, CA 91765 TEL：+1-909-979-9723