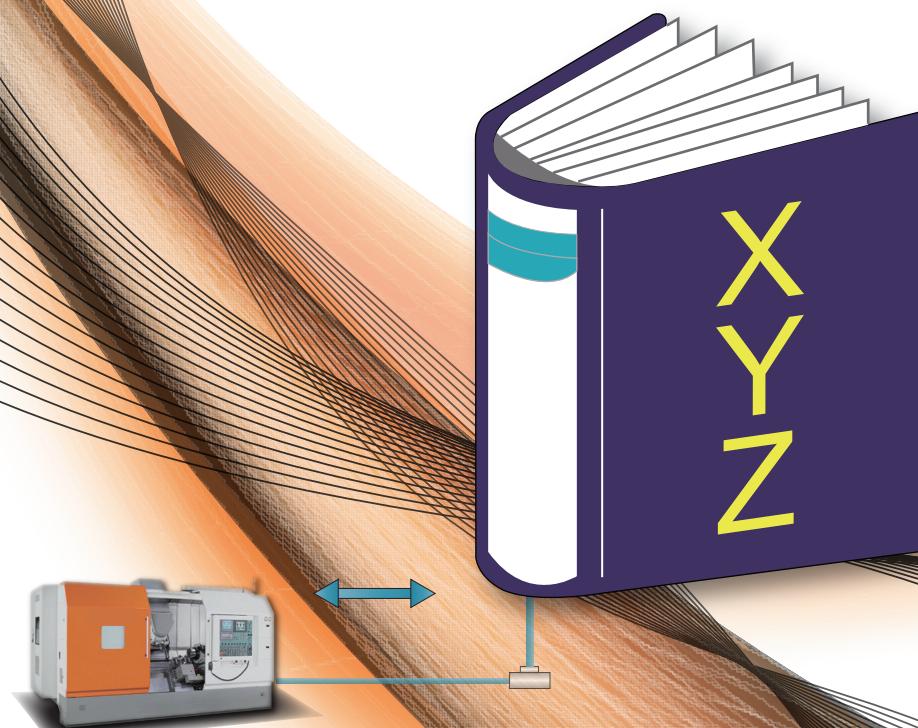


CNC Program Editing and Management System

NcEditor

Operation Manual



RenAn

Content

| | |
|--|------------|
| Content | I |
| 1. Software Installation Introduction | 1-1 |
| 1.1 Terms Definitions | 1-1 |
| 1.2 Symbols Explanations | 1-1 |
| 1.3 System Requirement | 1-1 |
| 1.4 Software Installation | 1-2 |
| 1.5 Software Uninstallation | 1-7 |
| 2. Software Interface | 2-1 |
| 2.1 Interface Introduction | 2-1 |
| 3. File Management | 3-1 |
| 3.1 Open File | 3-1 |
| 3.2 Open Recent File | 3-3 |
| 3.3 Close File | 3-5 |
| 3.4 Rename and comment | 3-7 |
| 3.5 Search File | 3-11 |
| 3.6 New File | 3-17 |
| 3.7 New Folder | 3-18 |
| 3.8 Delete File | 3-19 |
| 3.9 Delete Folder | 3-21 |
| 3.10 Copy and Move | 3-23 |
| 3.11 Merge File | 3-27 |
| 3.12 Sort Files and Save Columns Position | 3-30 |
| 4. Program Edit | 4-1 |
| 4.1 Command Set | 4-1 |
| 4.2 Save File, Save As and Refresh | 4-9 |
| 4.3 Copy, Move and Undo | 4-14 |
| 4.4 Syntax Debug | 4-21 |
| 4.5 Search | 4-24 |
| 4.6 Process Description, Fixture and Product | 4-29 |
| 4.7 Design | 4-31 |
| 4.8 Calculator | 4-33 |

| | |
|--|-------------|
| 5. Part Contour Calculation | 5-1 |
| 5.1 Counter Turn | 5-1 |
| 5.2 Counter Mill | 5-6 |
| 6. NC Code Comparison | 6-1 |
| 6.1 Files comparison | 6-1 |
| 6.2 Comparison Settings | 6-5 |
| 7. Tools Management | 7-1 |
| 7.1 Use Tool Magazines | 7-1 |
| 7.2 Tool Magazines Management | 7-4 |
| 7.3 Tool Data Management | 7-11 |
| 8. Machine Type File | 8-1 |
| 8.1 Select Machine Type File | 8-1 |
| 8.2 Common Machine Setting File | 8-2 |
| 8.3 Machine Type Parameters Settings | 8-4 |
| 9. CNC Program Simulation (Turning and Milling) | 9-1 |
| 9.1 Material Setting for Turning | 9-1 |
| 9.2 Simulation Operation and Settings for Turning | 9-2 |
| 9.3 Material Setting for Milling | 9-10 |
| 9.4 Simulation Operation and Settings for Milling | 9-11 |
| 10. NC Files Transmissions | 10-1 |
| 10.1 Send Program – From PC to CNC | 10-1 |
| 10.2 Receive Program – From CNC to PC | 10-10 |
| 10.3 DNC Mode | 10-16 |
| 10.4 DNC Mode – Restart | 10-19 |
| 10.5 Common Transmission Term | 10-21 |
| 10.6 Protocol Settings – Basic | 10-26 |
| 10.7 Protocol Settings – Advance | 10-29 |
| 10.8 Protocol Settings – File Type | 10-30 |
| 10.9 Protocol Settings – DNC Mode | 10-31 |
| 10.10 Protocol Settings – Remote Mode | 10-32 |
| 11. Print Programs/Graphics | 11-1 |

| | |
|---|-------------|
| 11.1 Print Programs | 11-1 |
| 11.2 Print Programs/Graphics | 11-2 |
| 11.3 Double Columns for One Page | 11-4 |
| 12. Remote Mode for Transmission | 12-1 |
| 12.1 Remote Transmission | 12-1 |
| 12.2 Remote Transmission Command List | 12-2 |
| 12.3 Remote Transmission Message Explain List | 12-3 |
| 12.4 Programs from CNC to PC | 12-4 |
| 12.5 Transmit NC Code from PC to CNC | 12-13 |
| 12.6 Remote Advanced Functions | 12-18 |
| 12.7 Remote Deletion and Assigned File Name Functions | 12-23 |
| 13. Software System Settings | 13-1 |
| 13.1 Debug Mode Settings | 13-1 |
| 13.2 Alignment Settings | 13-3 |
| 13.3 Extend Files Settings | 13-5 |
| 13.4 Image Search Path Settings | 13-6 |
| 13.5 Setting Font | 13-8 |
| 13.6 User Manage | 13-13 |
| 13.7 Language Setting | 13-17 |



1. Software Installation Introduction

1.1 Terms Definitions

PC : Personal Computer

1.2 Symbols Explanations

Button : 【 】 e.g. Click 【New】 button

List : 【 】 e.g. Click 【Setting】 in the toolbar

Key : [] e.g. Click [Enter] to change column

Column : _____ e.g. Click the field Account in the user login window

Term : 「 」 e.g. 「CNC」、「PC」

1.3 System Requirement

CPU : Dual Core 2.8G/above

RAM : 2GB/above

HD : 5GB/above Available

VGA : 128MB/above and compatible with OpenGL

Monitor: 1600 x 900/above

Input : Keyboard, Mouse

OS : Win 7, Win 8

1.4 Software Installation

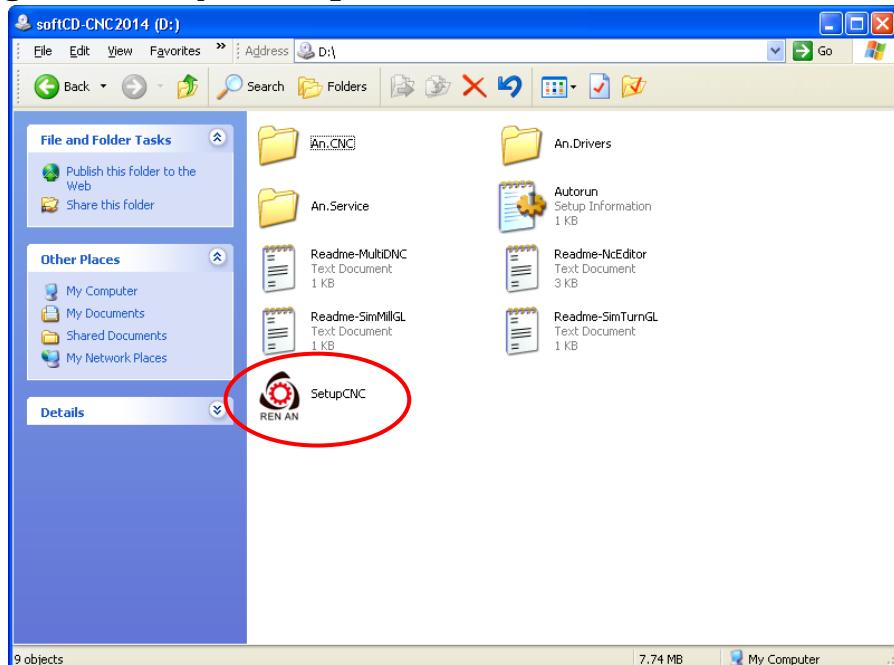
Before installation, please make sure the computer can meet the system requirement.

1.4.1 Steps

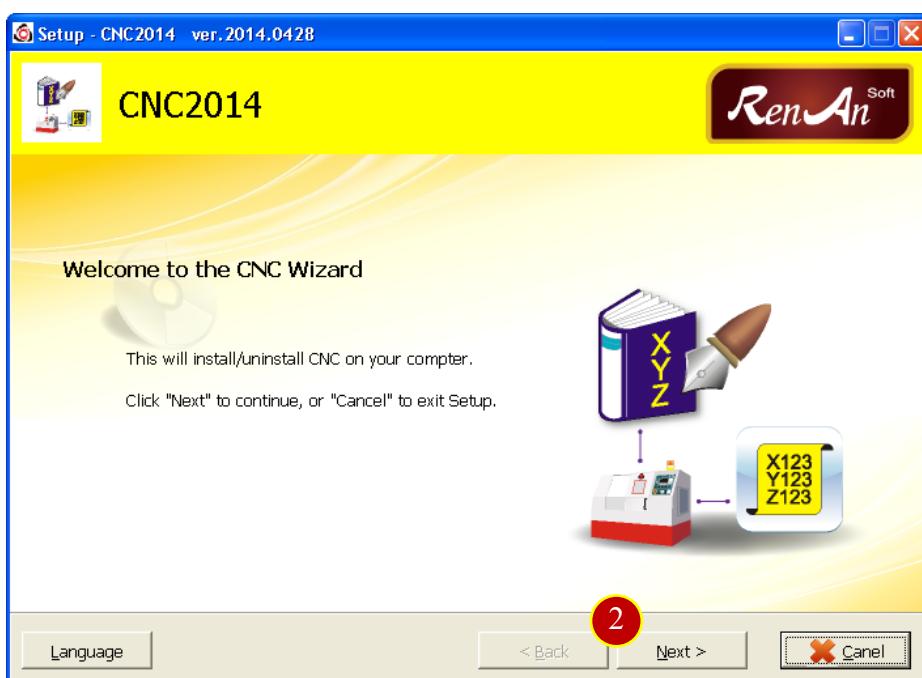
(1) Put RenAn software disc into DVD-ROM, the program will automatically run.

Please execute the exe file if the program does not open itself.

e.g. 「D:\SetupCNC.exe」

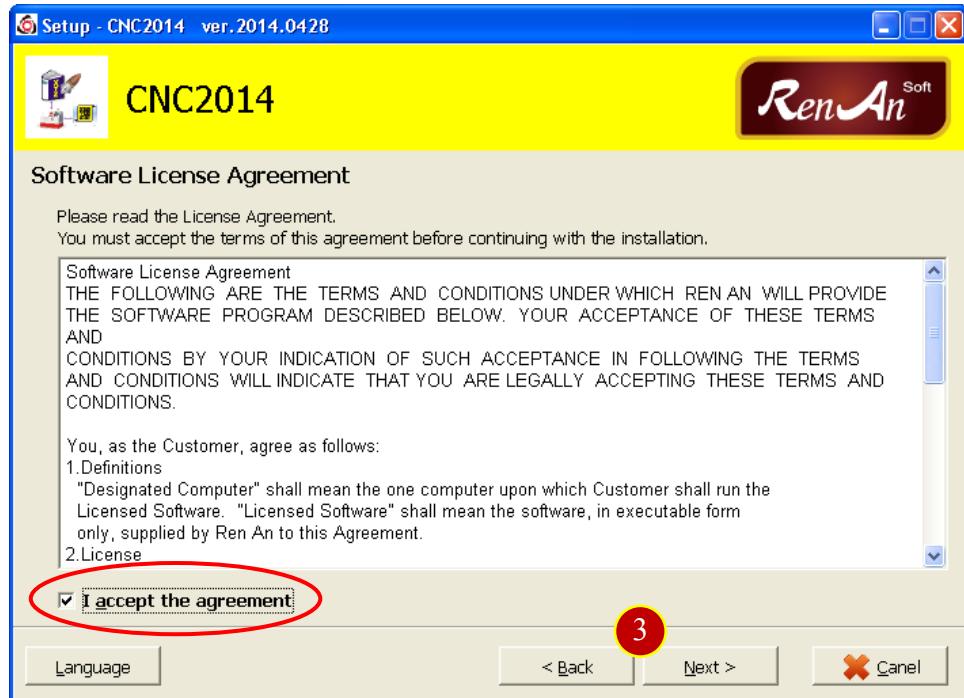


(2) Click 【Next】



Software Installation Introduction

(3) Read the agreement contract, check 【I agree the agreement】 box, then click 【Next】



(4) Click 【Install】



(5) Select

- NcEditor
- Demo files
- MultiDNC
- Lathe Counter
- Mill Counter
- Key Service
- KeyPro driver
- BDE (Borland Database Engine) driver

Default location will be C:\An.CNC

Or click 【Browse】 to set the location

(6) Click 【Install】



(7) Installing, please wait for a while.



(8) Installed successfully, click 【Finish】and restart the computer to execute the software functionally.



(9) After Installation, there will be a shortcut for NcEditor 2014 on the desktop



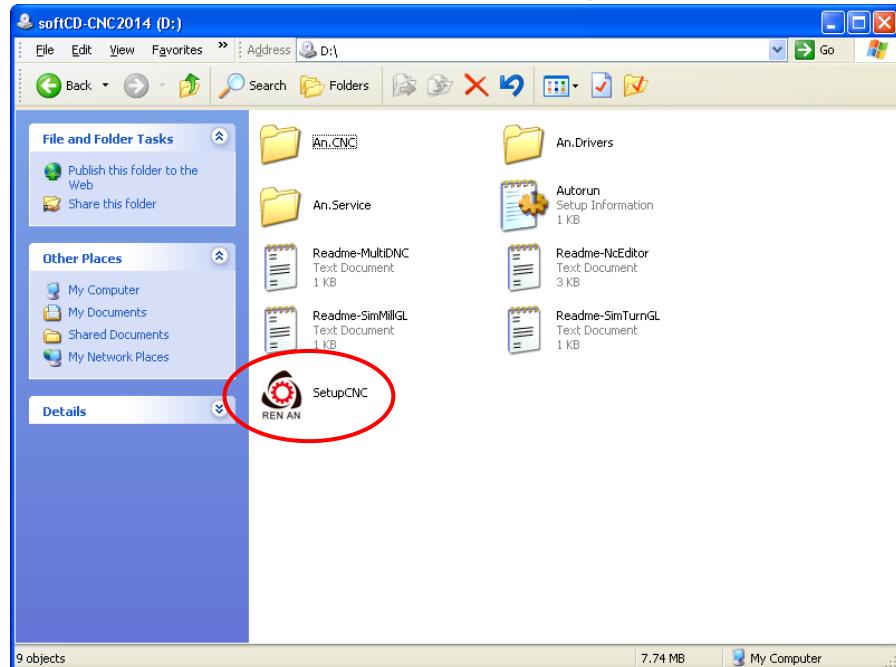
1.5 Software Uninstallation

Uninstall and remove CNC 2014 system

1.5.1 Steps

(1) Put RenAn software disc into DVD-ROM, the program will automatically run.

Please execute the exe file if the program does not open itself.



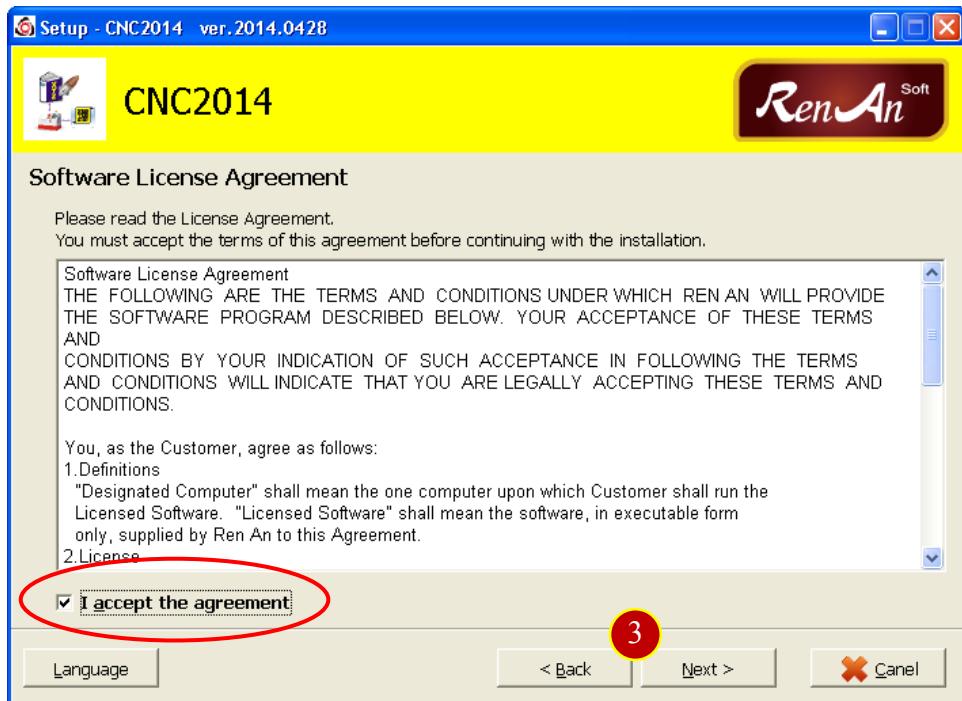
(2) Click 【Next】



NcEditor MANUAL

Software Installation Introduction

(3) Read the agreement contract, check 【I agree】 box, then click 【Next】



(4) Select 【Uninstall】



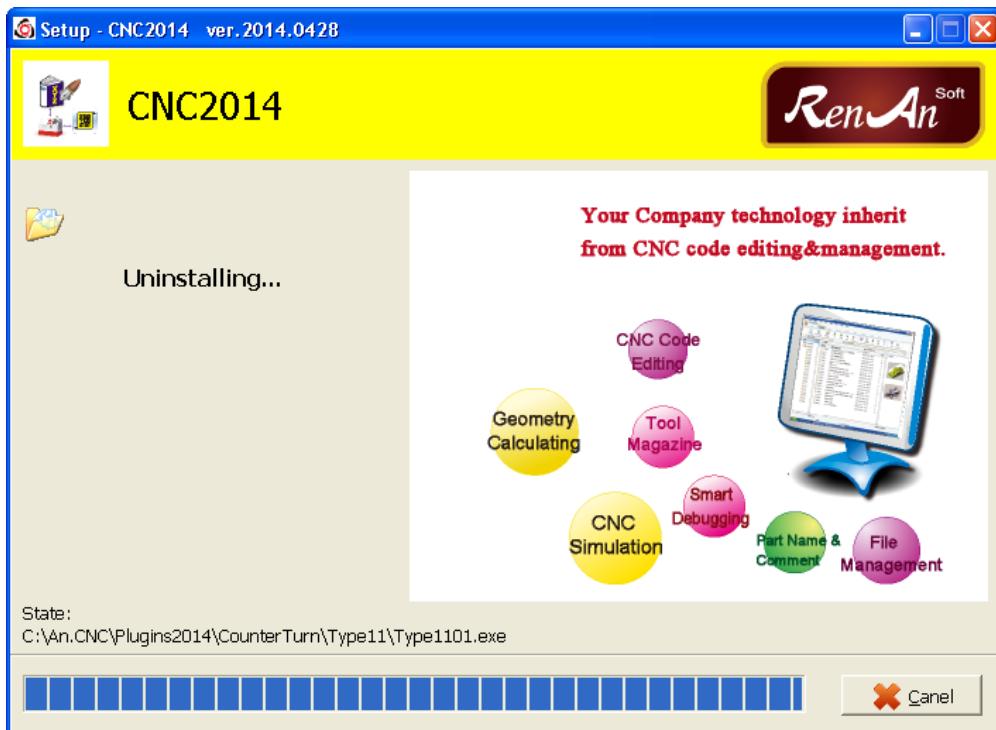
(5) Select

- NcEditor
- User & Setting data
- MultiDNC
- Lathe Counter
- Mill Counter
- Key Service
- KeyPro driver
- BDE (Borland Database Engine) driver

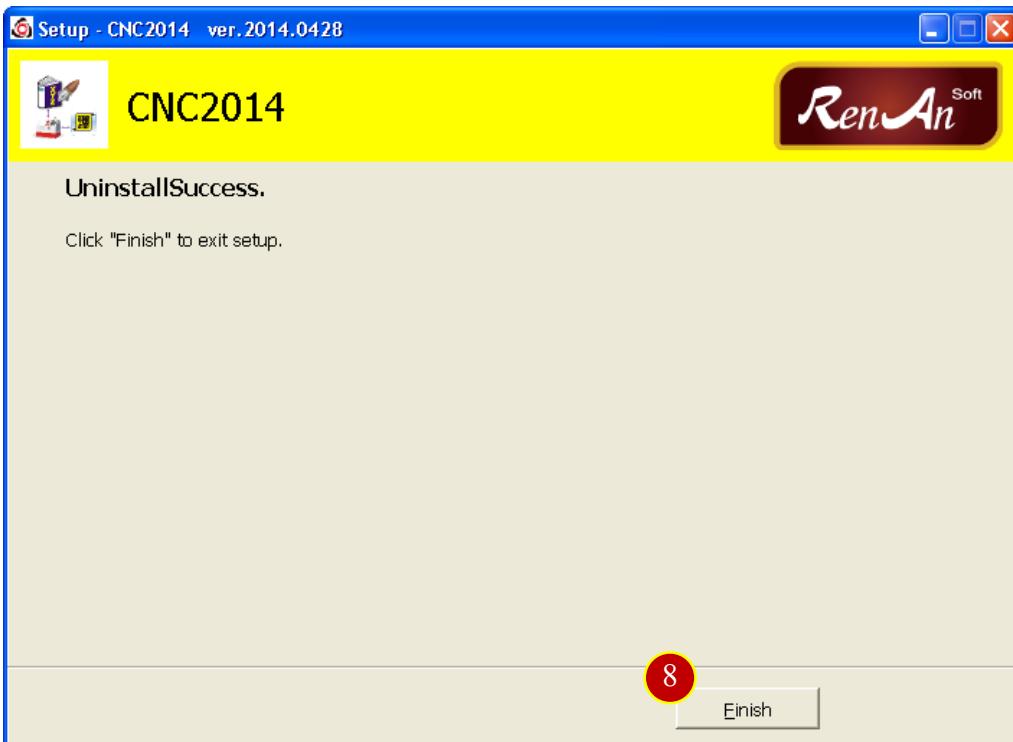
(6) Click 【Uninstall】



(7) Uninstalling, please wait for a while.



(8) Uninstalled successfully, click 【Finish】and restart the computer.

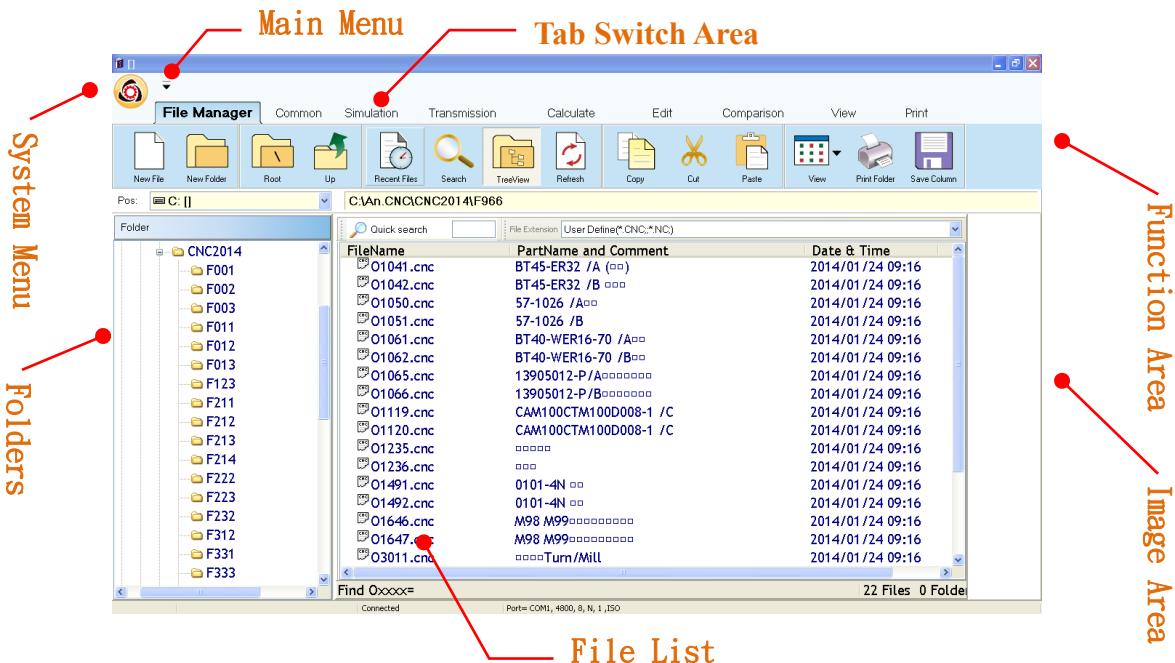


2. Software Interface

2.1 Interface Introduction

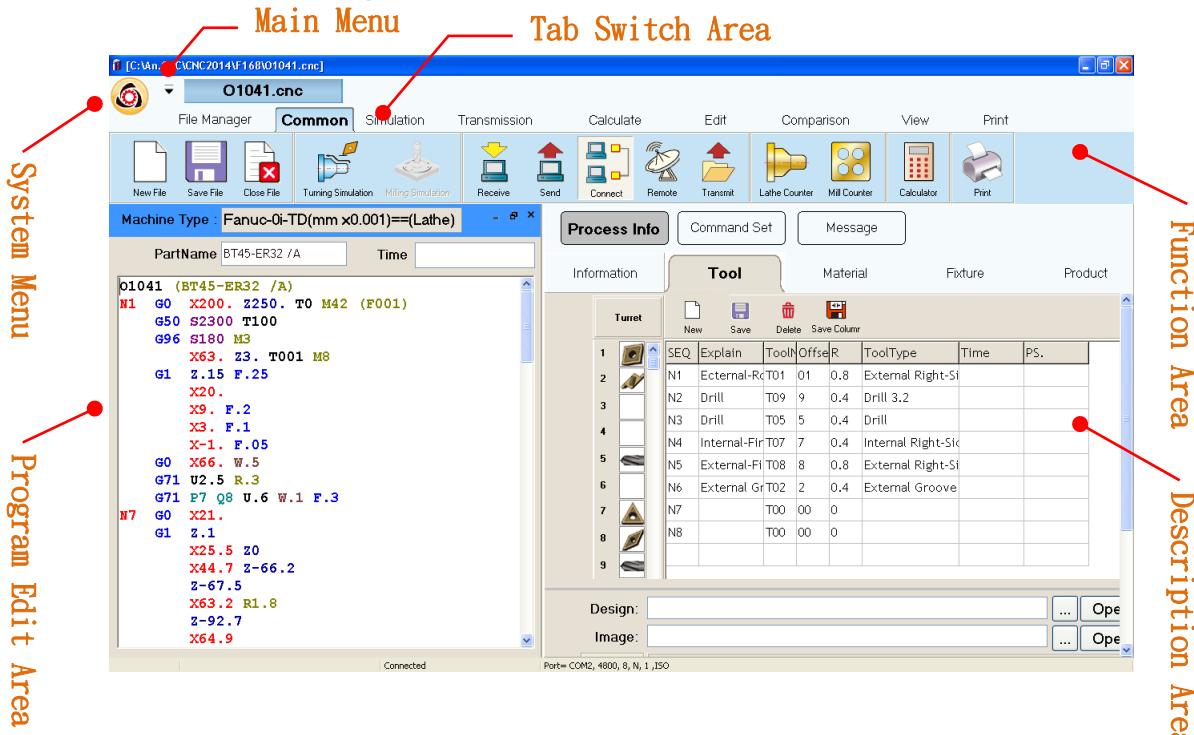
2.1.1 File Manager Mode

On File Manager Mode, there are Main Menu, System Menu, Tab Switch Area, Function Area, Folders, File List, Image Area.



2.1.2 Program Edit Mode

On Program Edit Mode, there are Main Menu, System Menu, Tab Switch Area, Function Area, Program Edit Area, Description Area.



3. File Management

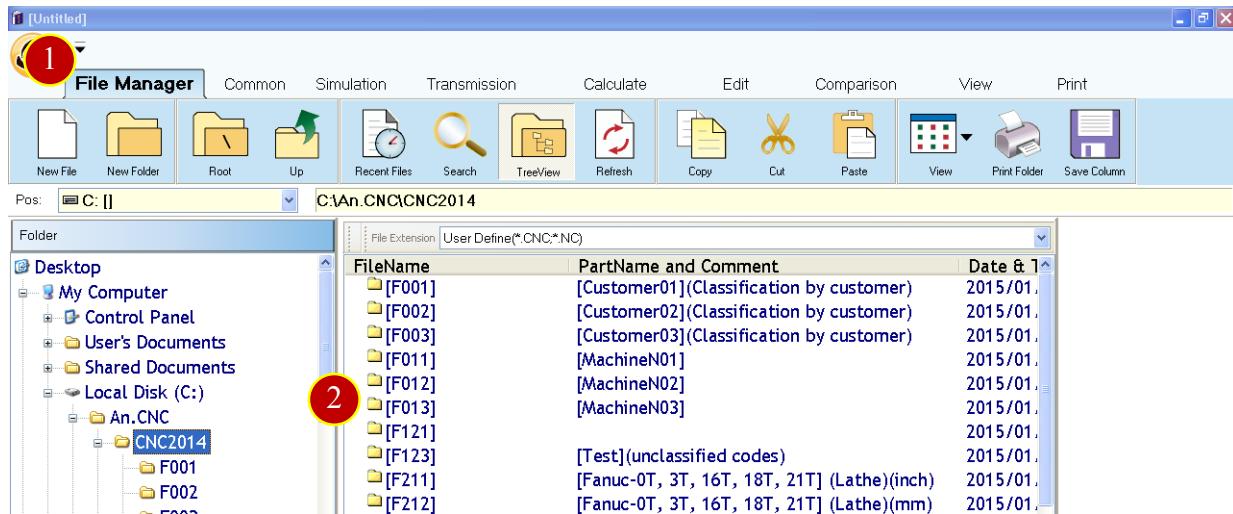
Operation of New File, Delete, Modify, Search NC files and classify folders.

3.1 Open File

3.1.1 Open File

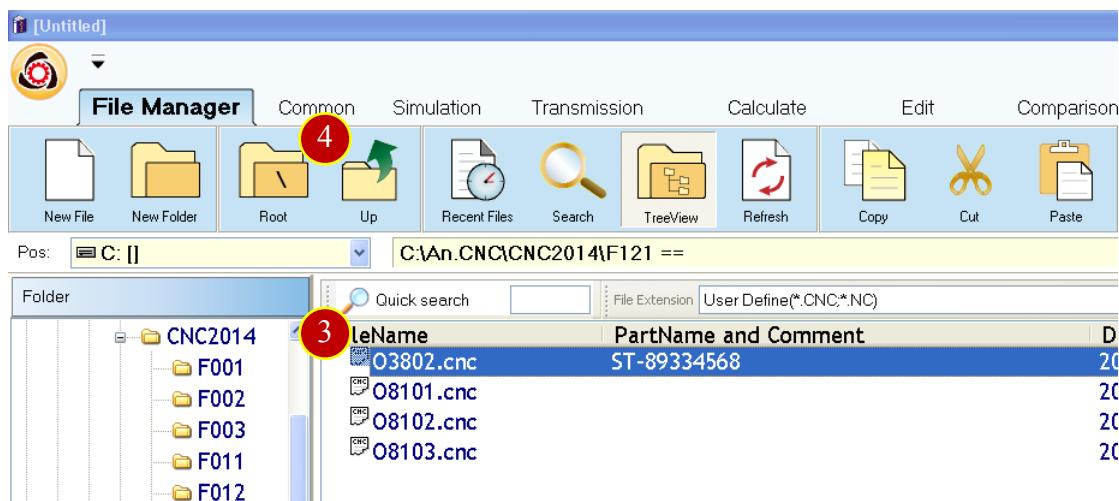
e.g. Open F121\03802.CNC

- (1) Click 【File Manager】 tab
- (2) Double click 【F121】 folder



(3) Double click file 【003802.CNC】

(4) Click 【Up】 , can be return



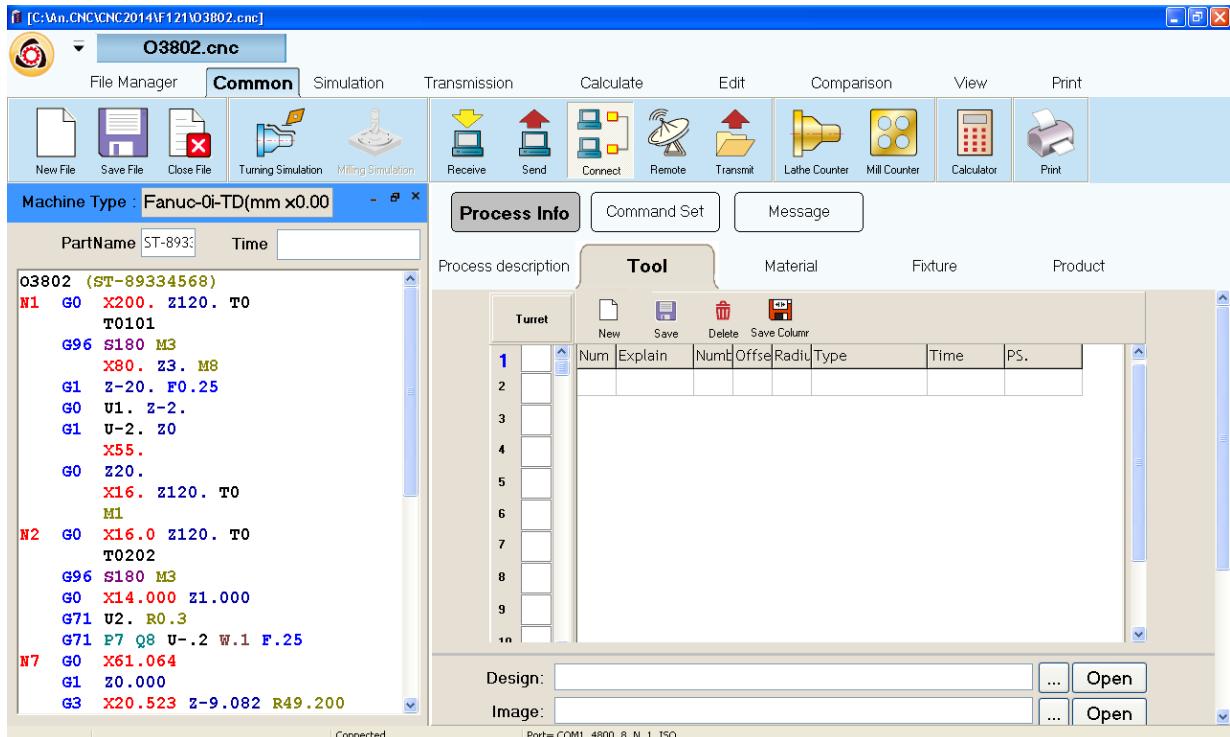
NcEditor MANUAL



RenAn Information Technology Co., Ltd.
www.RenAn.com.tw

File Management

(5) Program Edit Interface



3.2 Open Recent File

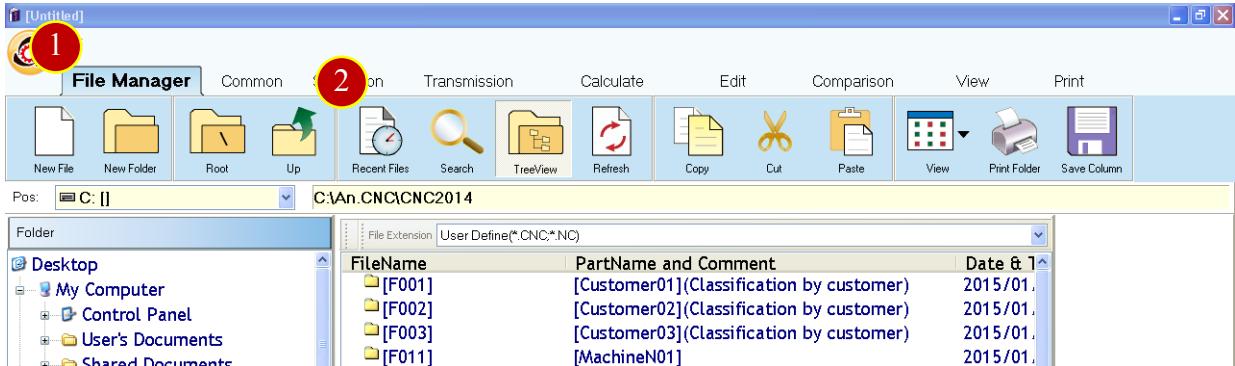
There are two methods to open recent files.

3.2.1 Open recent file(Method 1)

e.g. Open last file

(1) Click 【File Manager】 tab

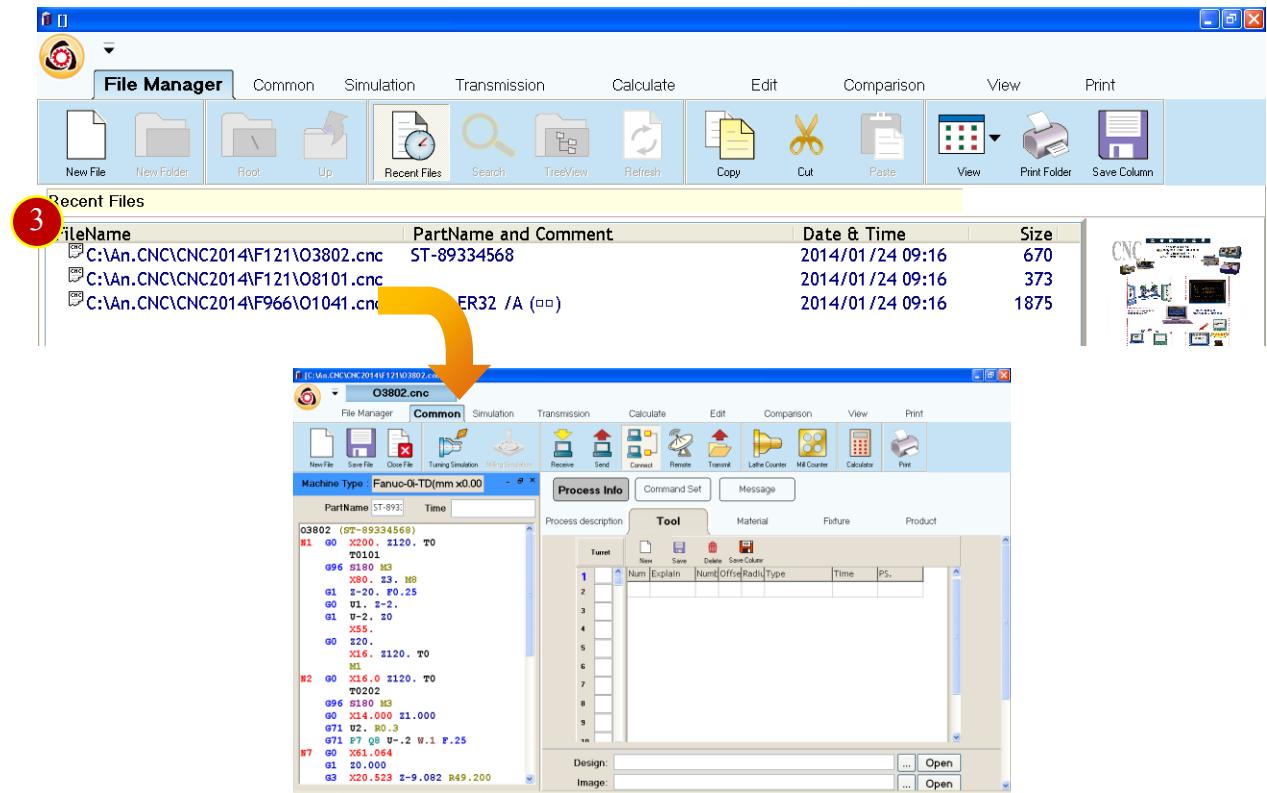
(2) Click 【Recent Files】



(3) Double click previous file name to open the program for edit

* Only display last 30 effective files

e.g. 03802.CNC



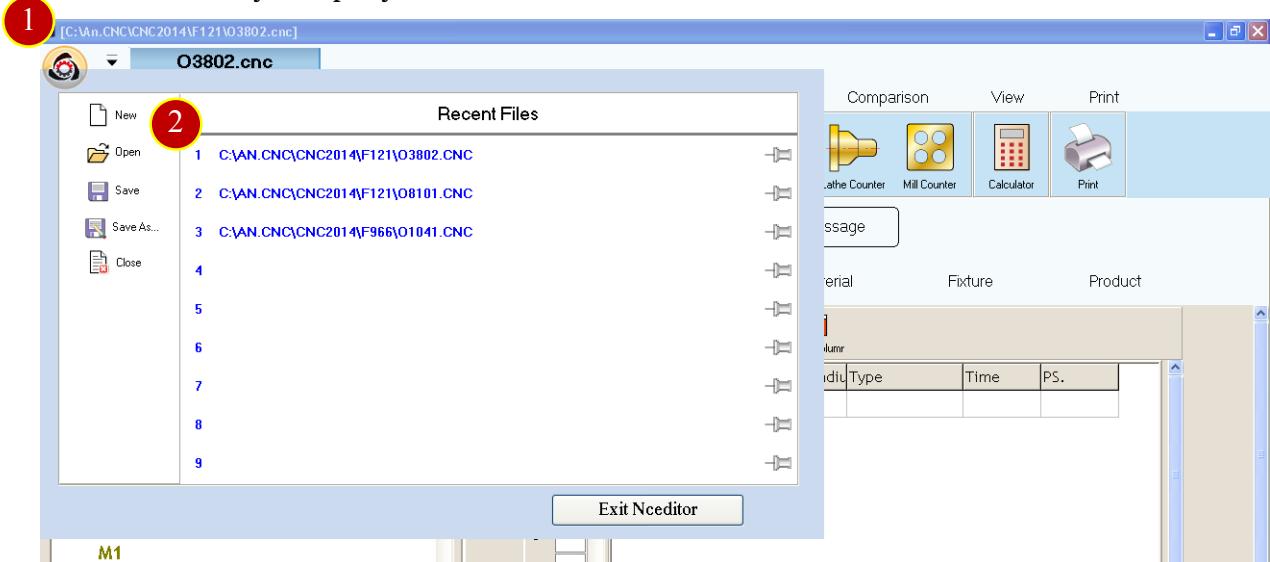
3.2.2 Open recent file(Method 2)

e.g. Open last file

(1) Click RenAn logo (System menu)

(2) Click the file name to open the program for edit

* Only display last 9 effective files



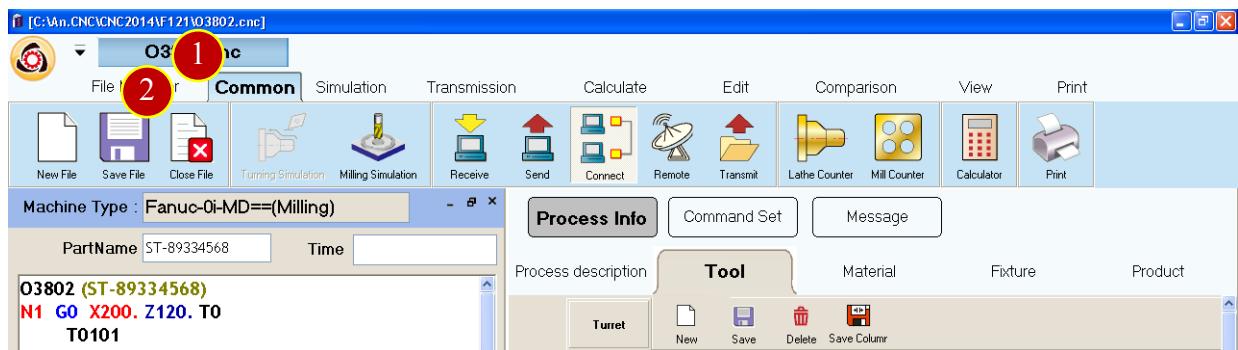
3.3 Close File

There are two methods to close a single file.

3.3.1 Close file(Method 1)

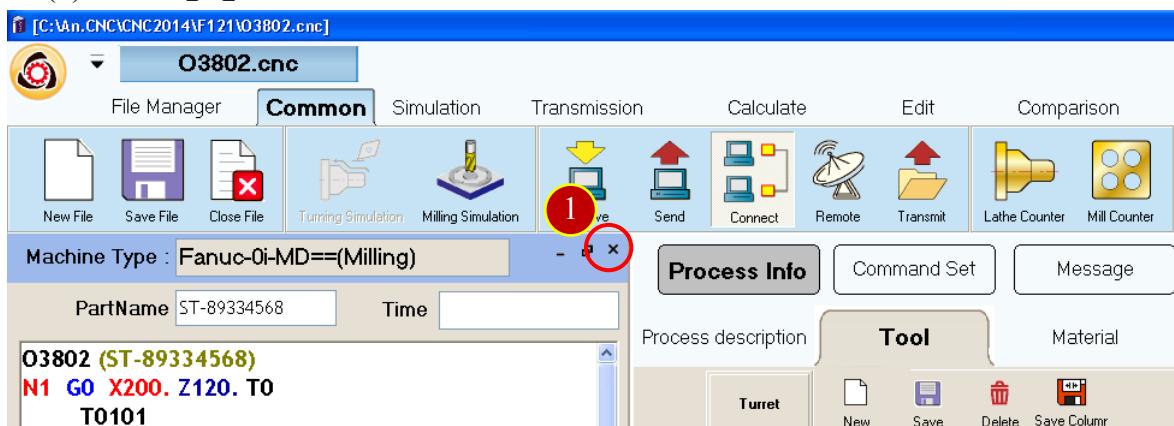
(1) Click 【Common】 tab

(2) Close the file by click 【Close File】 on the tool bar



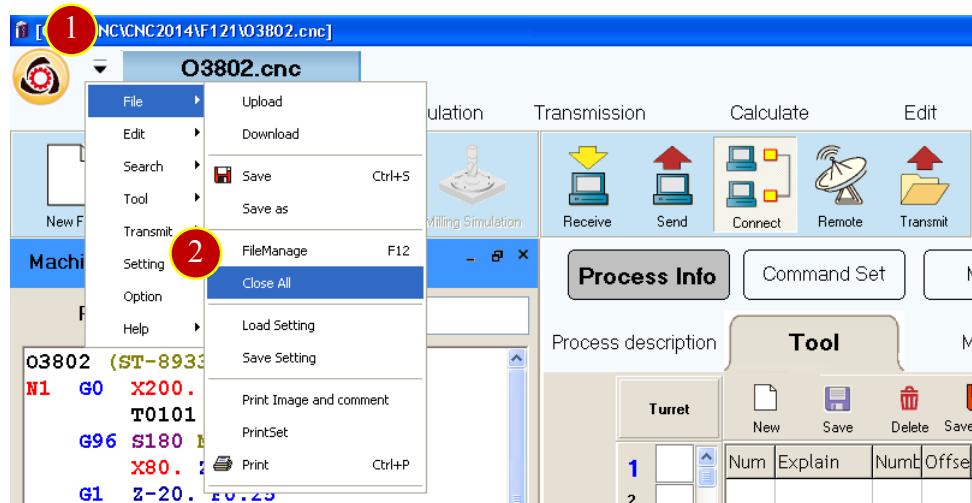
3.3.2 Close file (Method 2)

(1) Click 【X】 on the window to close the file



3.3.3 Close all the files

- (1) Click pull-down system list ▼
- (2) Click 【File】 > 【Close All】 to close all files
* The interface will back to File Manager



3.4 Rename and comment

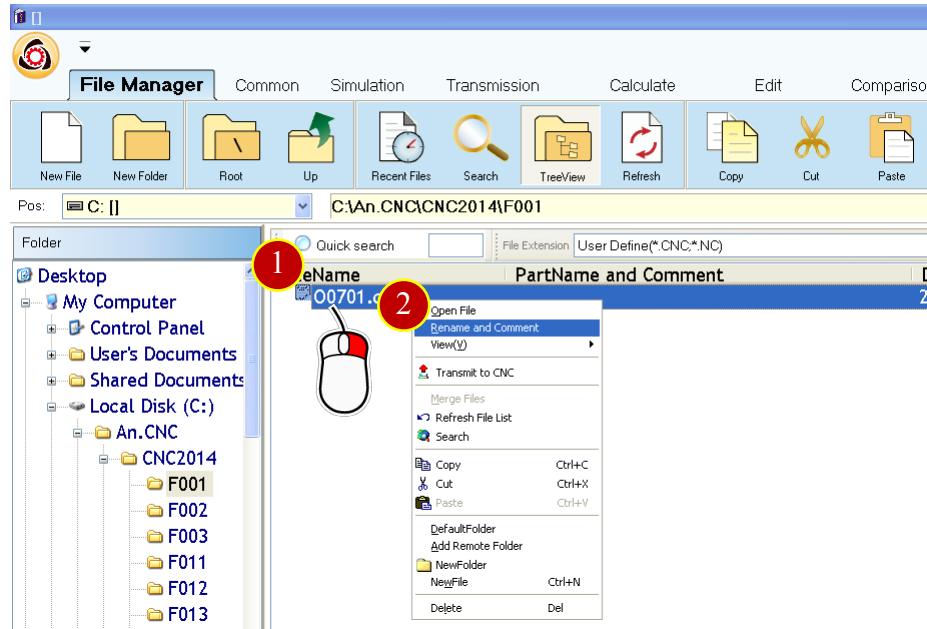
Move cursor at the files or folders and right click to rename or comment

3.4.1 Rename file

e.g. Change 00701.CNC to 00707.CNC

(1) Right click on file 【00701.CNC】

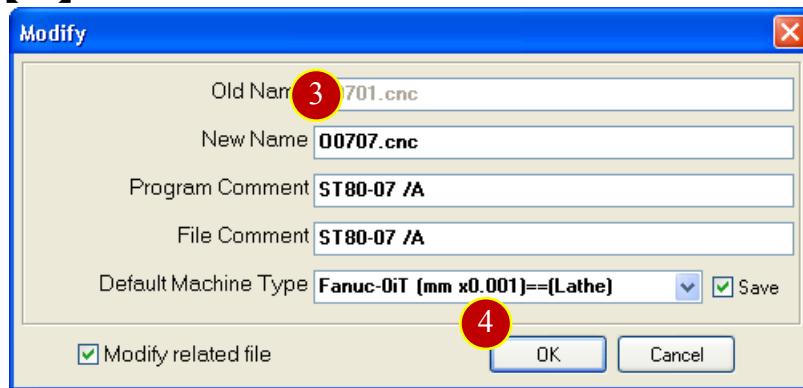
(2) Click 【 Rename and Comment】



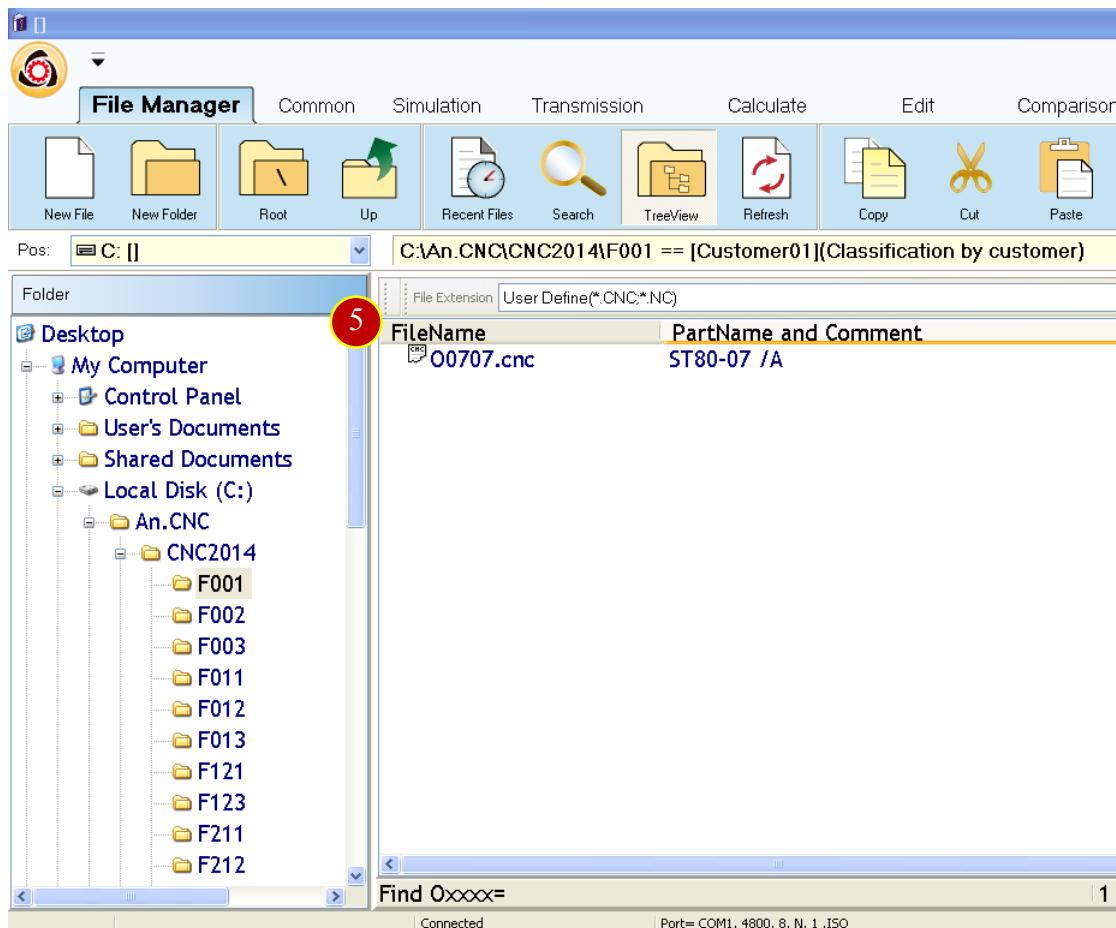
(3) Key in New Name, Comment and Default Machine Type

e.g. 【00707.CNC】 , 【ST80-07/A】 , 【Fanuc-0iT(mm x0.001)==(Turning)】

(4) Click 【OK】



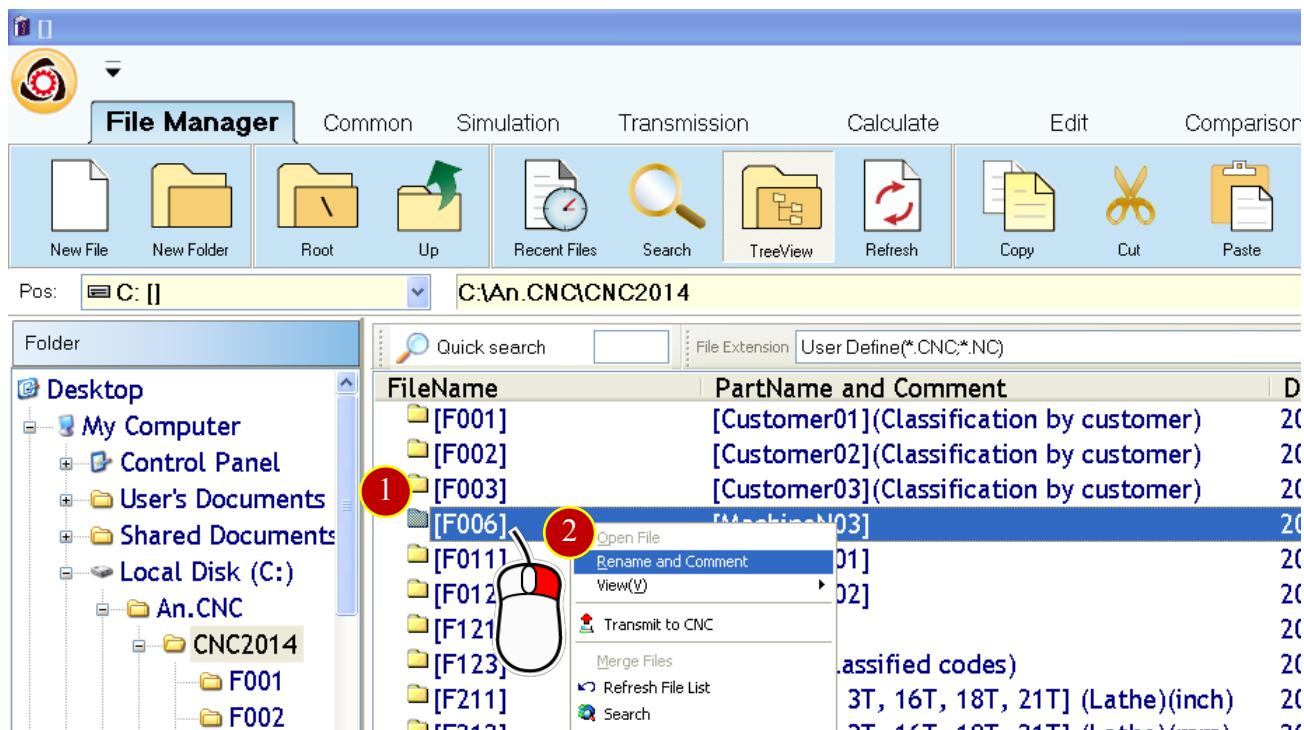
(5) Complete and display new file name



3.4.2 Rename folder

e.g. Change folder name from F006 to F013

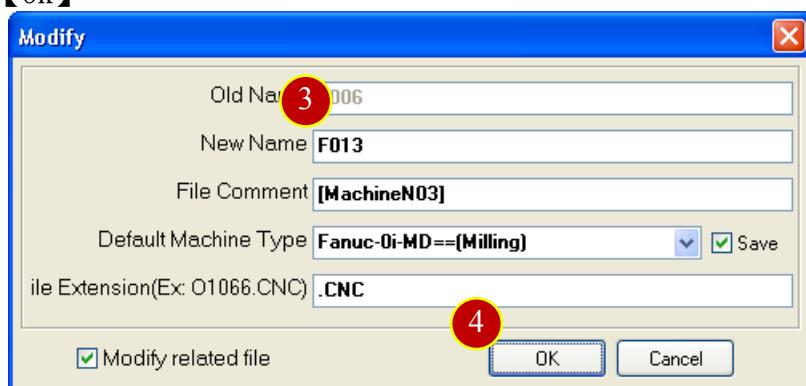
- (1) Right click on file **[F006]**
- (2) Click **[Rename and Comment]**



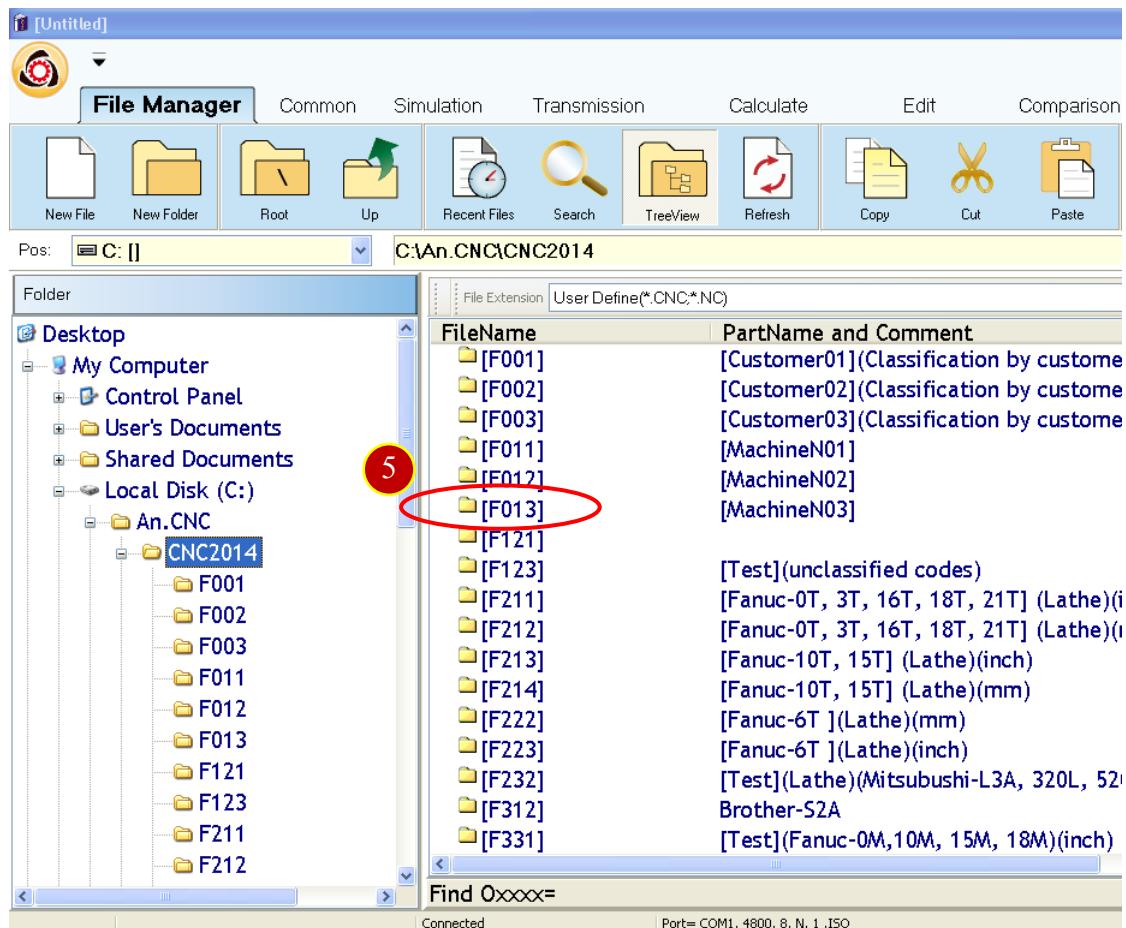
- (3) Key in new File Name, Comment, Default Machine Type and File Extension

e.g. **[F013]**, **[[Machine: C13]]**, **[Fanuc-0i-MD==(Milling)]**, **[.CNC]**

- (4) Click **[OK]**



(5) Complete and display new name



3.5 Search File

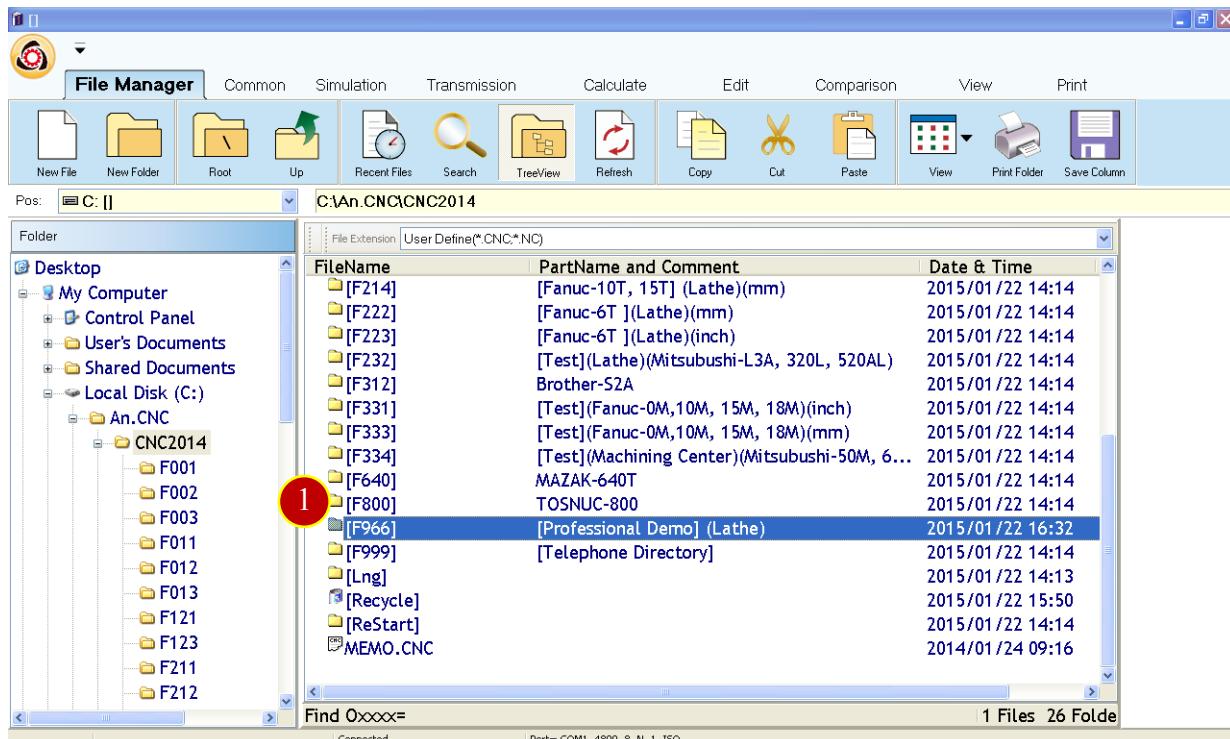
When the amount of files is enormous, there are methods to find the wanted files quickly.

3.5.1 Quick Search

e.g. Quick Search to find “KPU” related file in F966

(1) Double click 【F966】 folder to enter

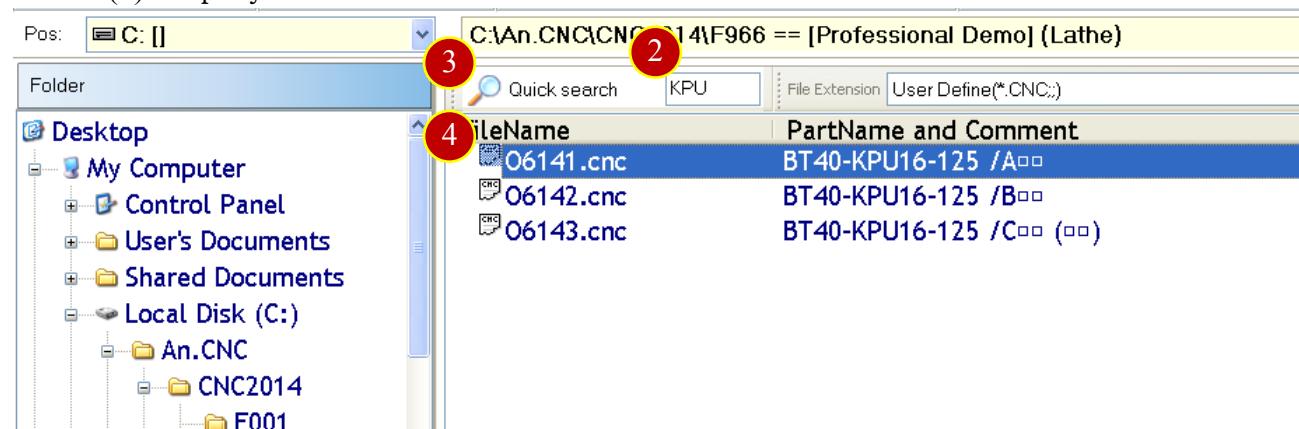
Display all the files



(2) Key in [KPU] in Quick Search

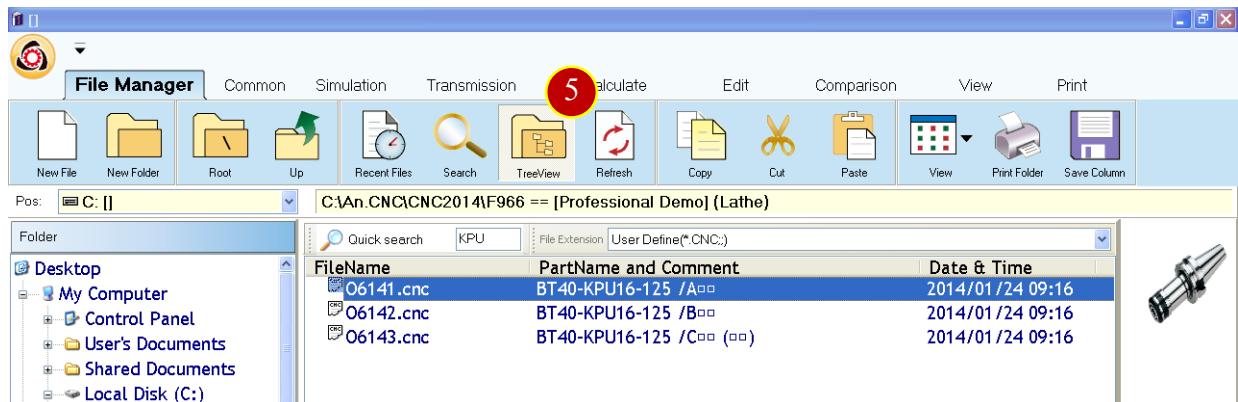
(3) Click 【Quick Search】

(4) Display search result



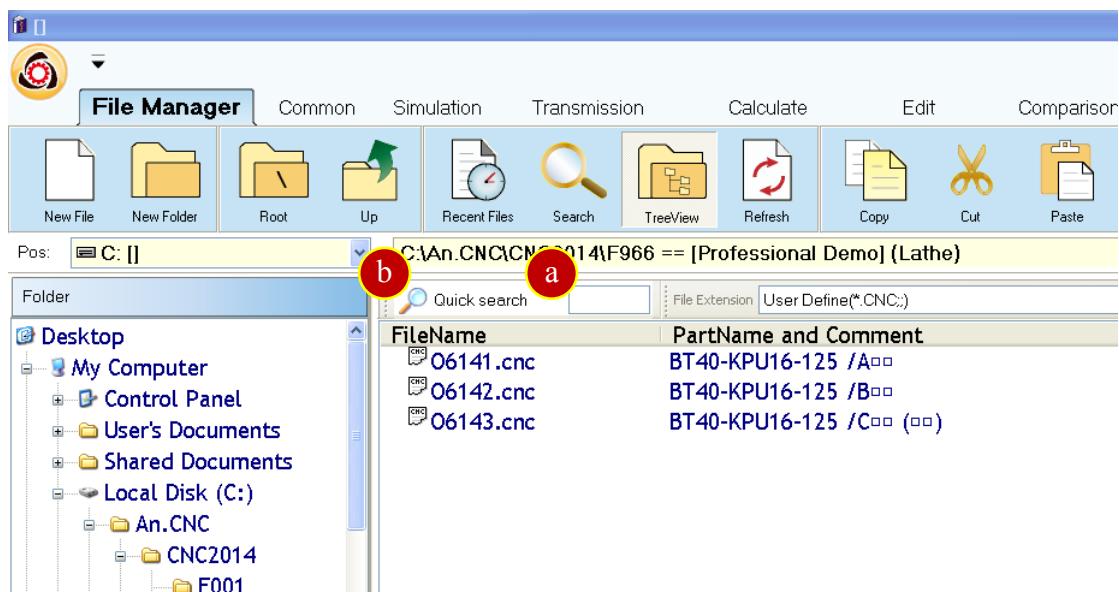
(5) If wish to display all the file data in the folder before quick search

Method 1: Click 【Refresh】



Method 2:

- Clear the content in Quick Search
- Click 【Quick Search】



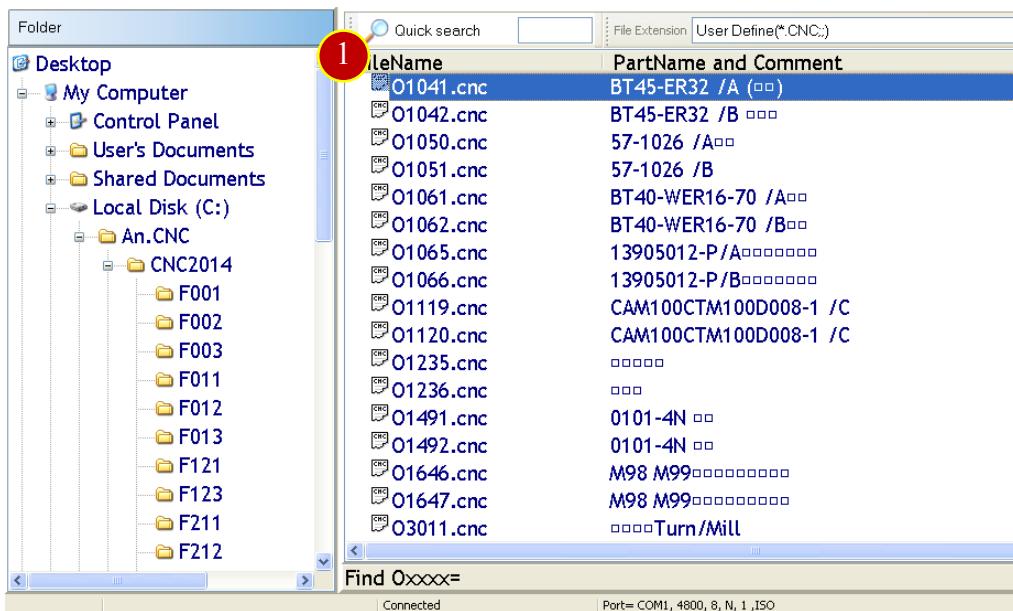
3.5.2 Key in 0 code to search directly

e.g. Search file "06143" in F966

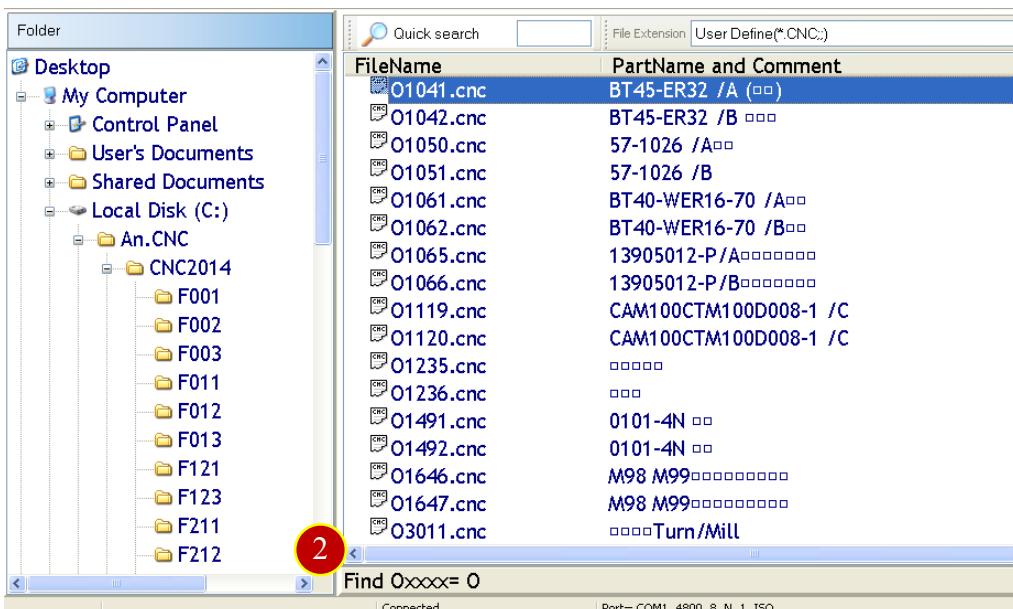
(1) Double click 【F966】 folder to enter

Click one file to move the cursor at Filename column

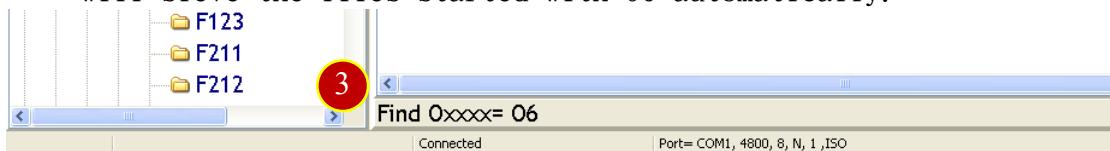
e.g. Click 01041.CNC



(2) Key in [0] English letter 0; status bar underneath will become 【Search 0xxxx= 0】 from 【Search 0xxxx=】



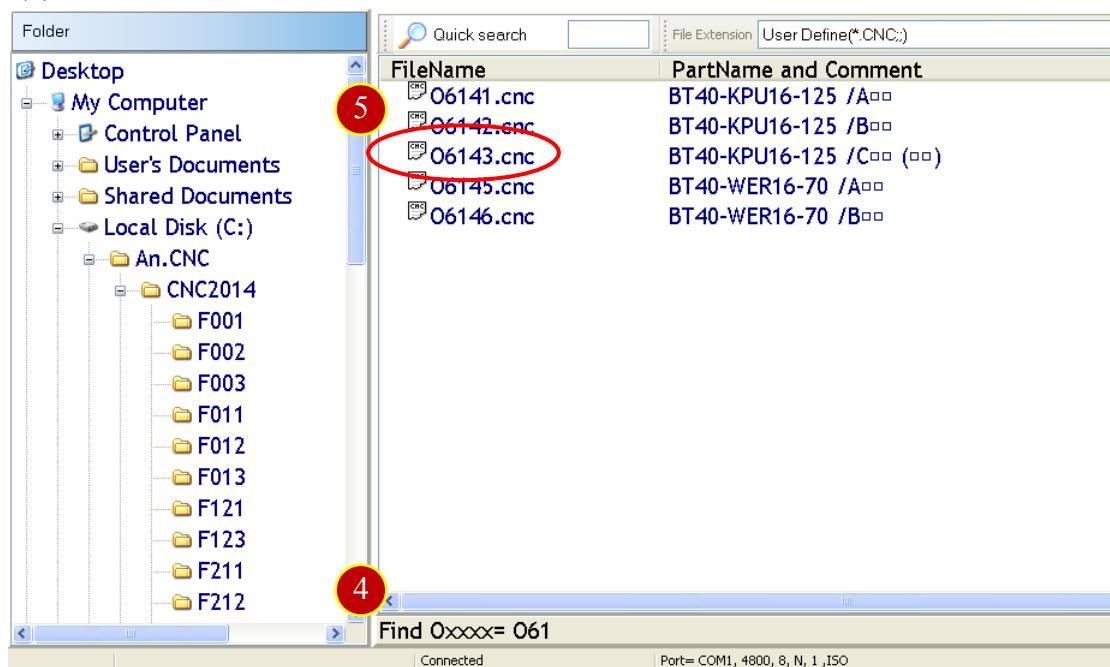
- (3) Key in [6]; status bar underneath will become 【Find 0xxxx=06】 , File Manager will sieve the files started with 06 automatically.



- (4) Key in [1]; status bar will become 【Find 0xxxx= 061】 , file list display the files started with 061.

* Key in English letter [0] again; status bar underneath will return to 【Search 0xxxx= 0】 and it can search for new ones

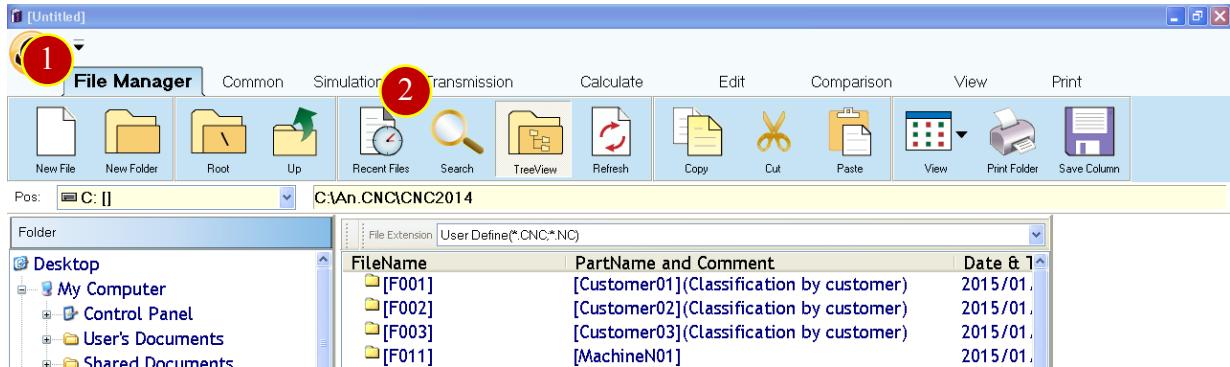
- (5) Move the cursor to 06143, double click to open the file



3.5.3 File Search in details

e.g. Search the files with comments BT40

- (1) Click 【File Manager】 tab
- (2) Click 【Search】



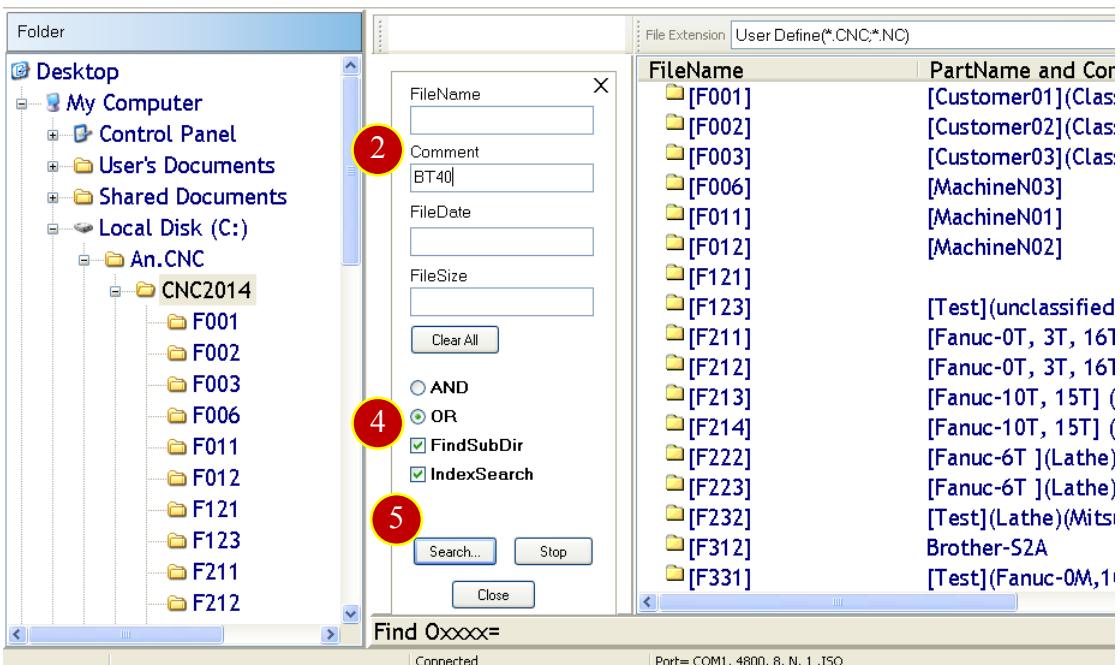
- (3) Key in search details

e.g. Key in BT40 in comment

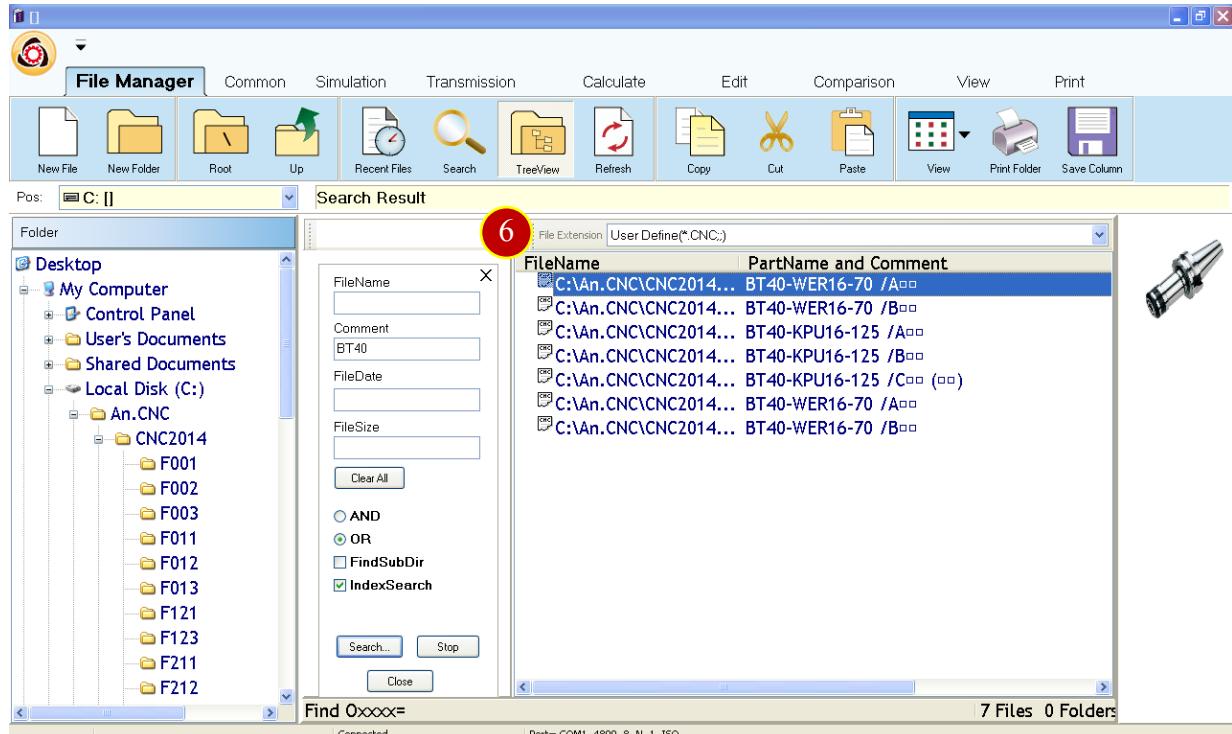
- (4) Search settings, check 【FindSubDir】 and 【IndexSearch】

- (a) AND: Meet all the details
- (b) OR: Meet one of the details
- (c) FindSubDir: Search the files in it
- (d) IndexSearch: Search the index

- (5) Click 【Search...】



(6) File list display the files meet the details



3.6 New File

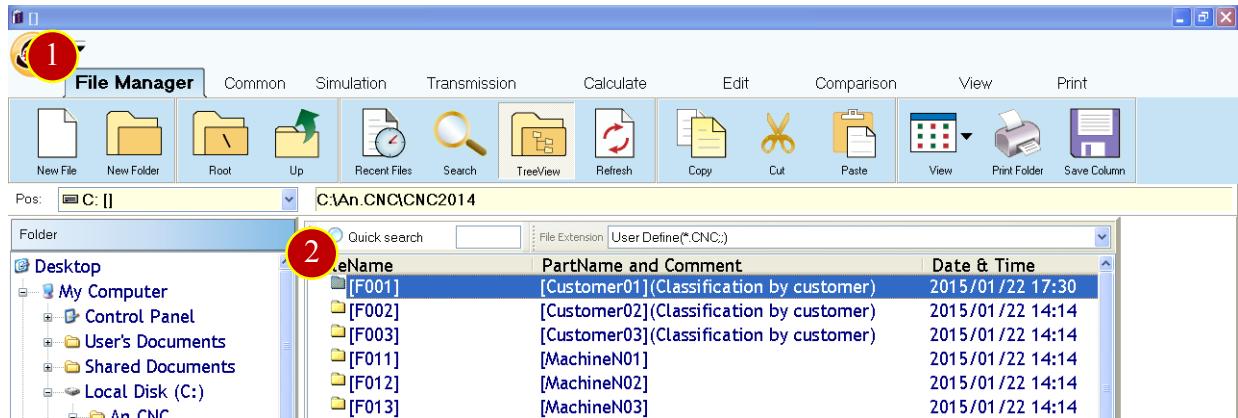
Add and create a blank CNC file

3.6.1 New File

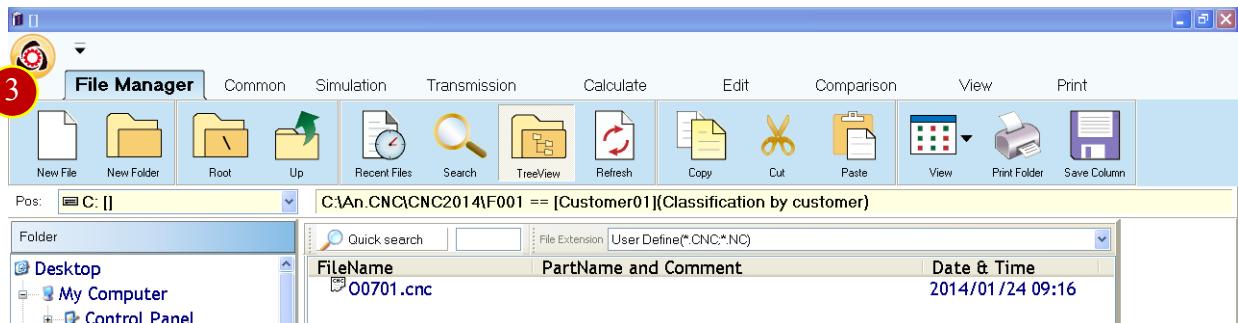
e.g. Add a new file in F001

(1) Click 【File Manager】

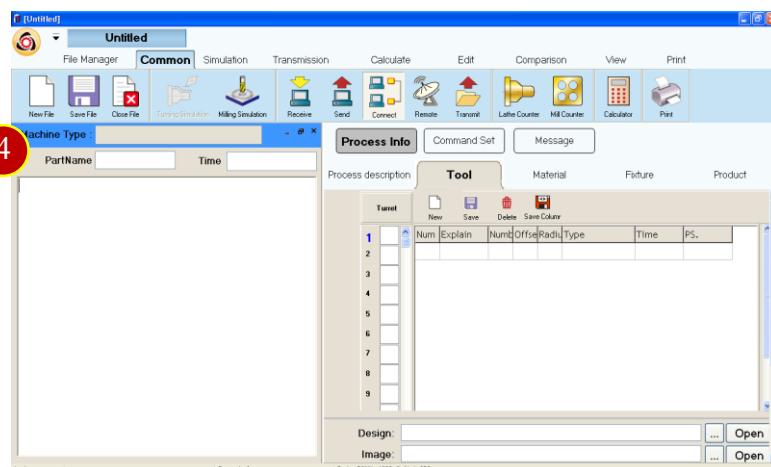
(2) Double click 【F001】 to enter the folder



(3) Click 【New File】



(4) Display blank CNC file



3.7 New Folder

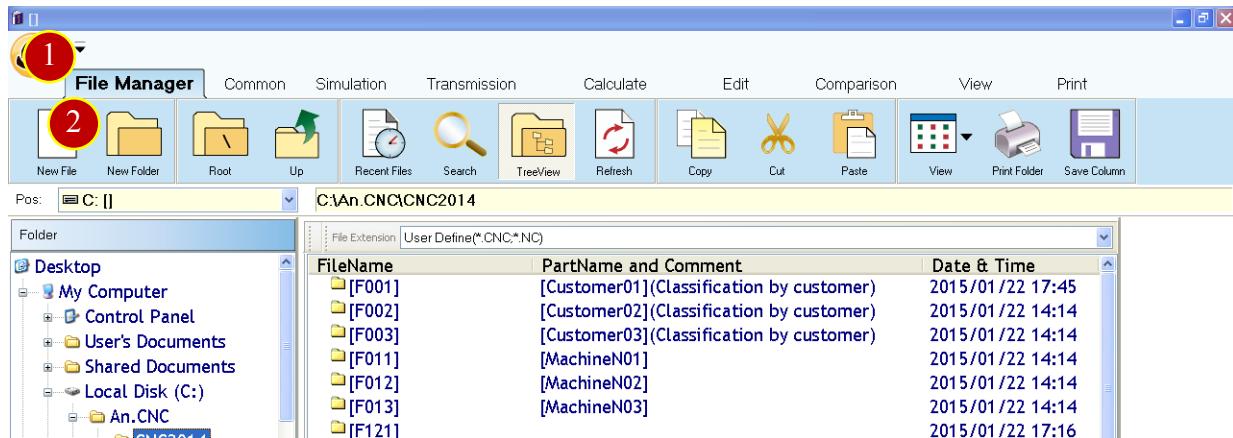
Add a new folder in order to sort the CNC files in it

3.7.1 Add a blank folder

e.g. Add F111 folder

(1) Click 【File Manager】 tab

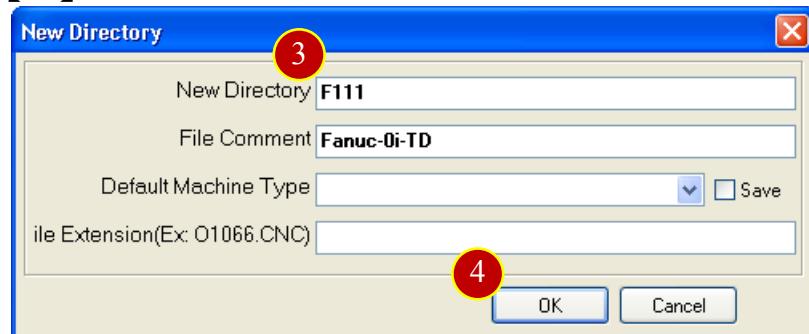
(2) Click 【New Folder】



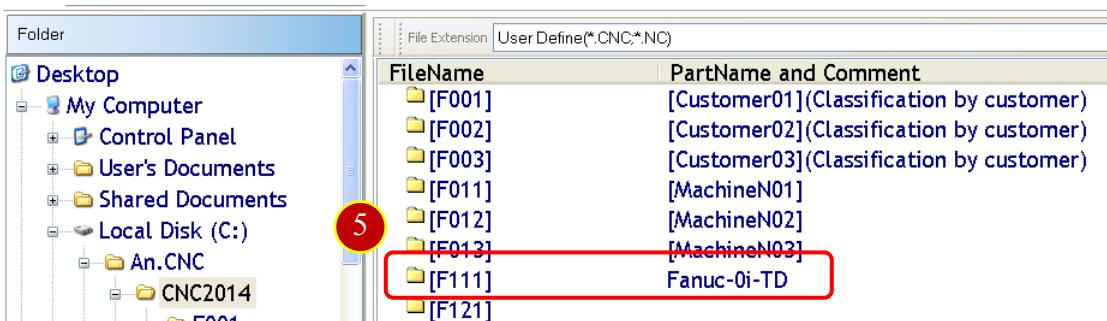
(3) Key in new folder name [F111] and Comment

e.g. Fanuc-0i-TD

(4) Click 【OK】



(5) File list displays new folder

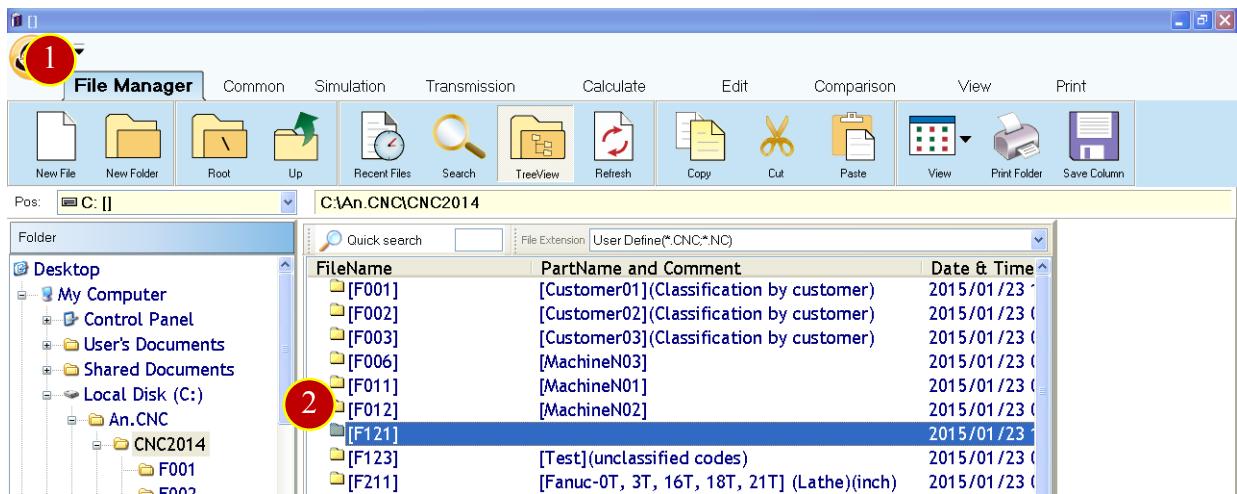


3.8 Delete File

3.8.1 Delete File

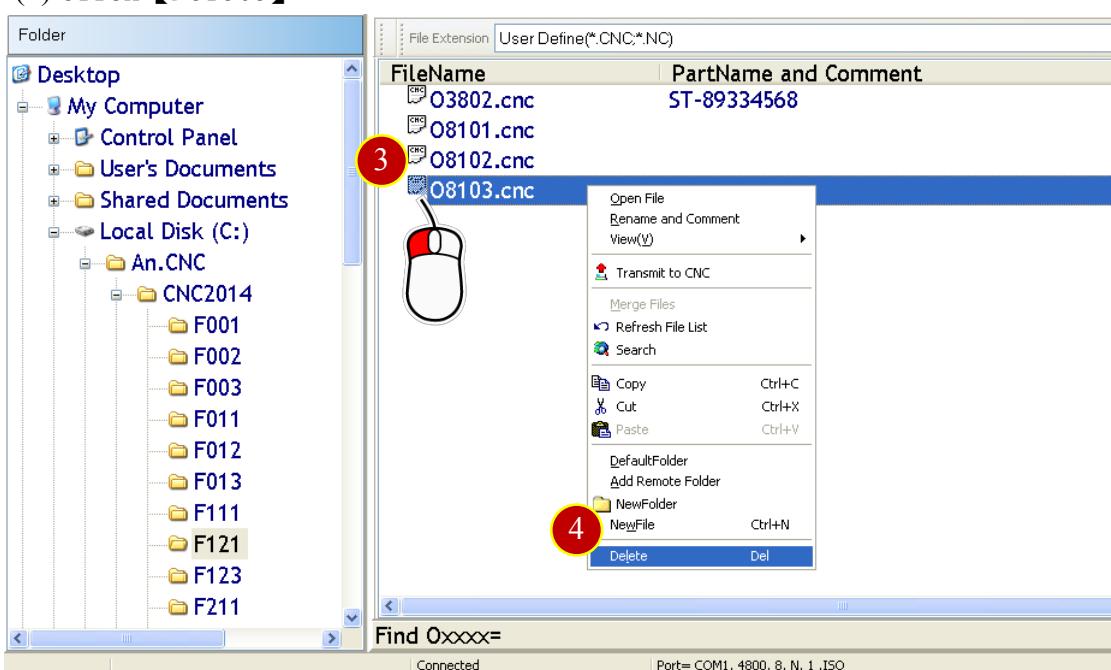
e.g. Delete 08103.CNC file in F121

- (1) Click File Manager tab
- (2) Double click 【F121】 to enter the folder

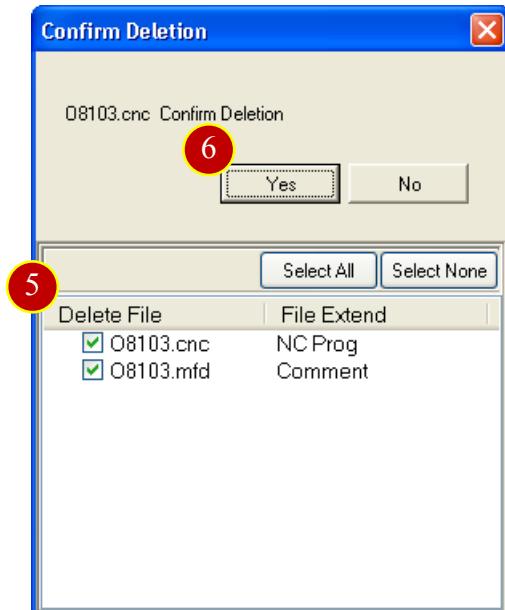


- (3) Right click at 08103.CNC file

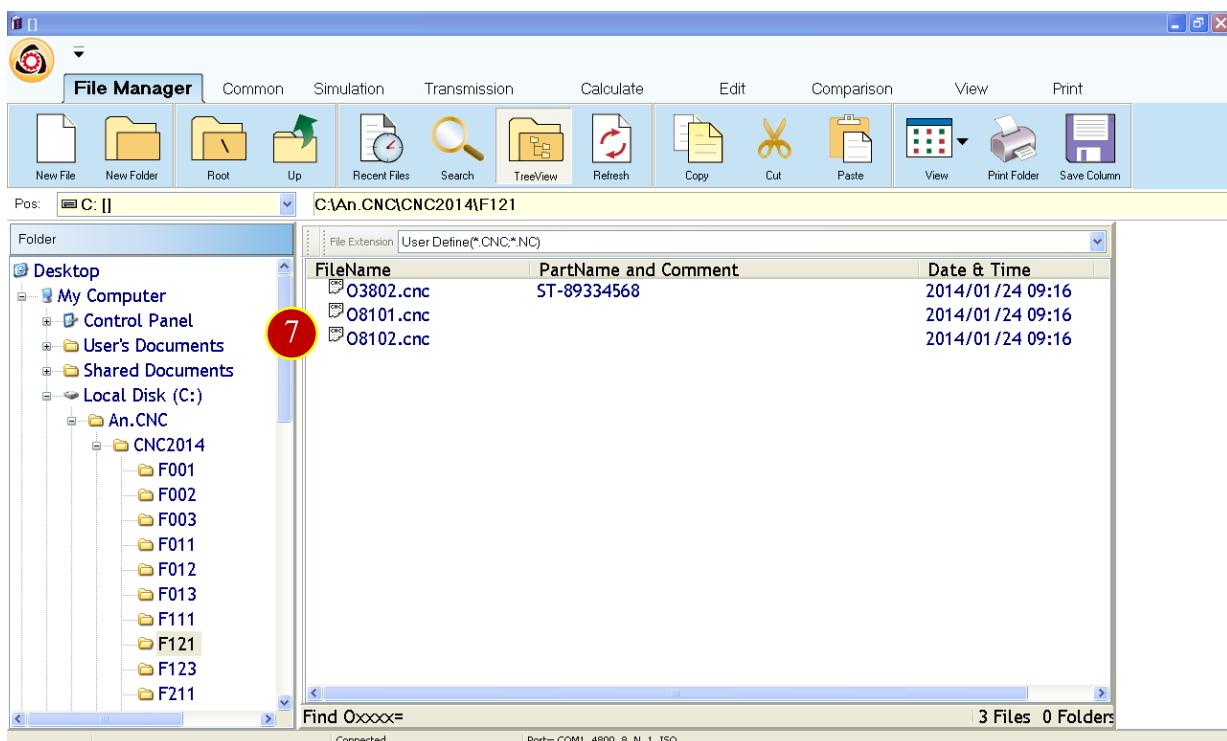
- (4) Click 【Delete】



- (5) Pop-out window with message “Confirm Deletion” to confirm deletion and whether the related files are to be deleted simultaneously
(6) Click 【Yes】



- (7) File list displays without those deleted files



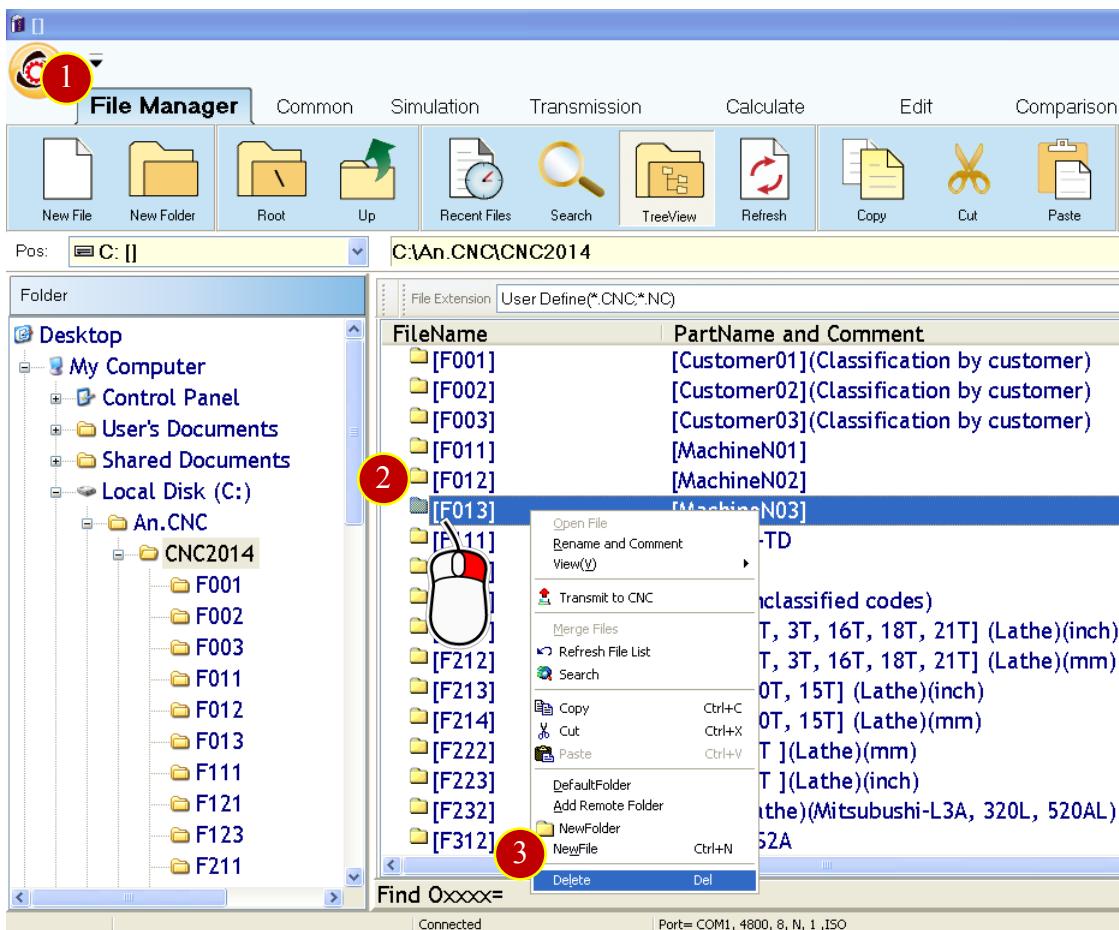
3.9 Delete Folder

3.9.1 Delete Folder

e.g. Delete F013 Folder

- (1) Click 【File Manager】 tab
- (2) Right click at folder 【F013】
- (3) Click 【Delete】

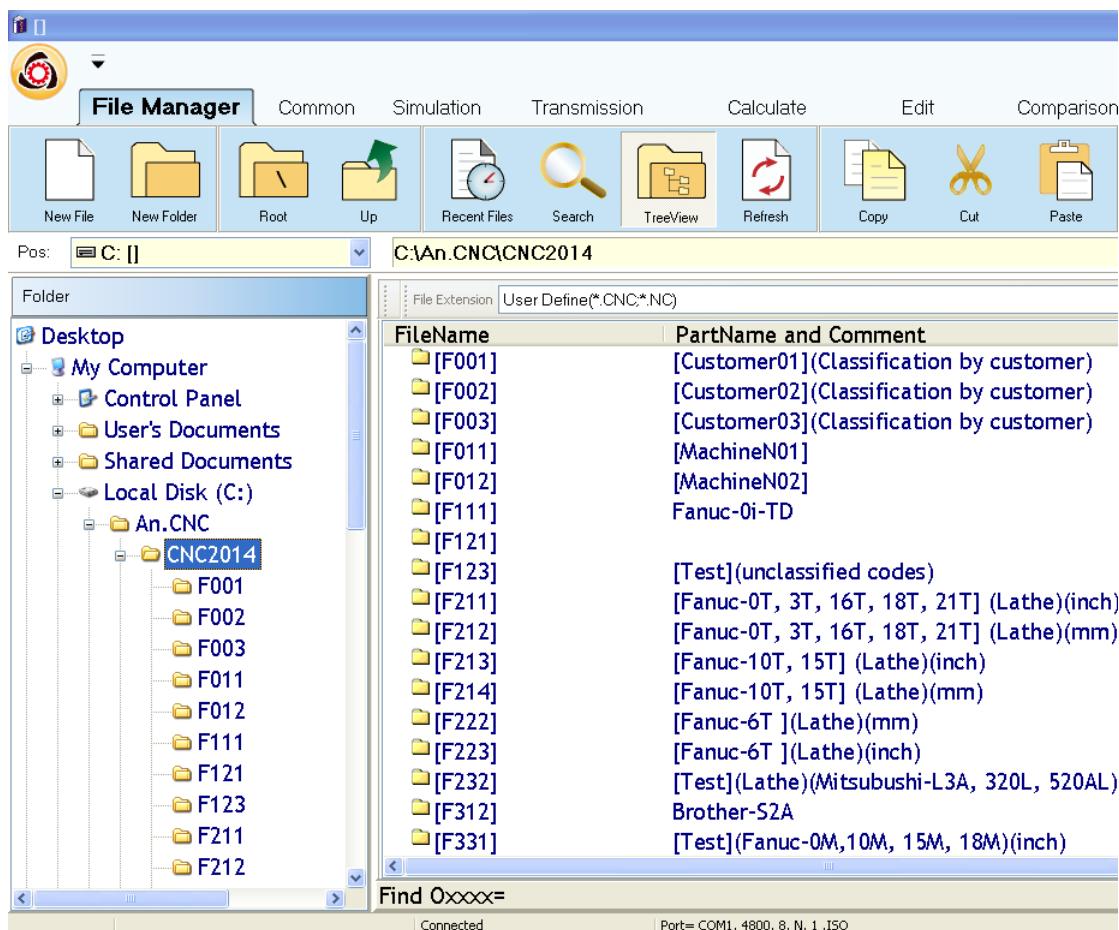
* Cannot delete the folder with CNC file in it



- (4) Click 【Yes】



(5) File list displays without those deleted folders

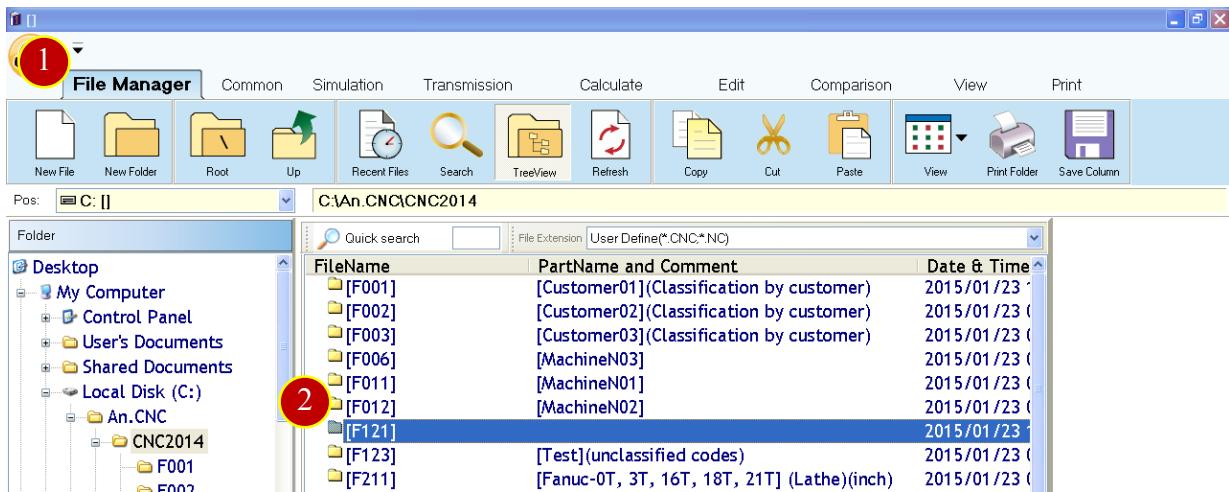


3.10 Copy and Move

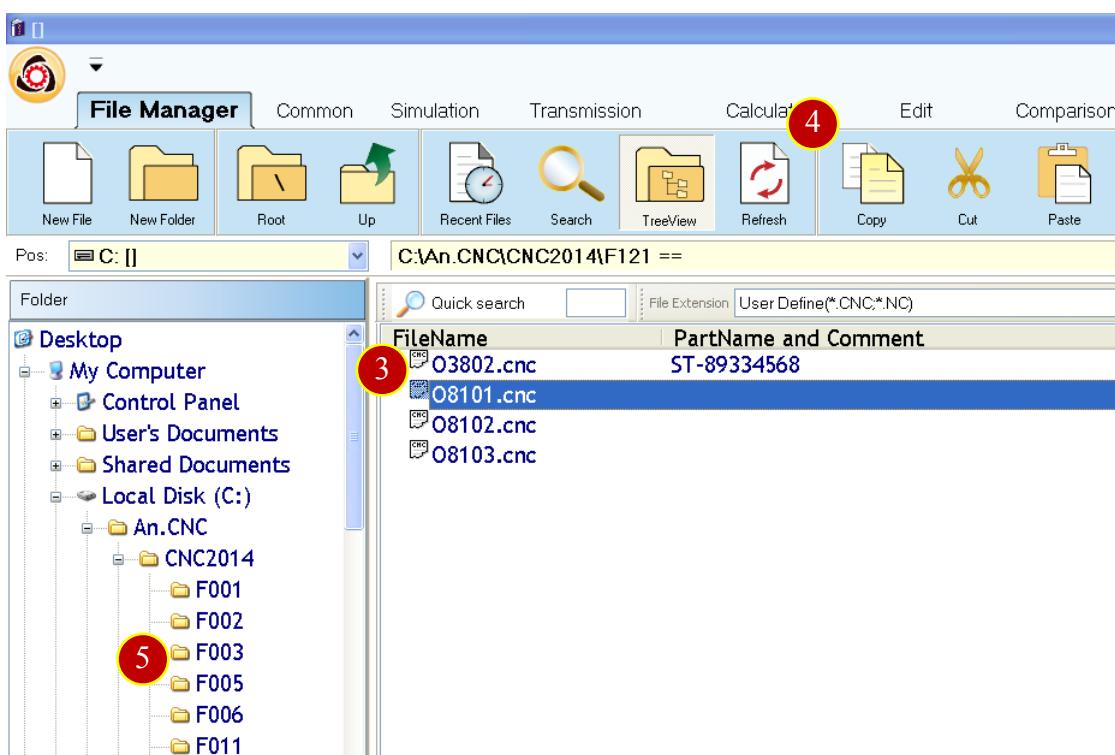
3.10.1 Copy

e.g. Copy F121\08101.CNC to folder F005

- (1) Click 【File Manager】 tab
- (2) Double click 【F121】 to enter



- (3) Select 08101.CNC
- (4) Click 【Copy】
- (5) Select 【F005】 folder to enter



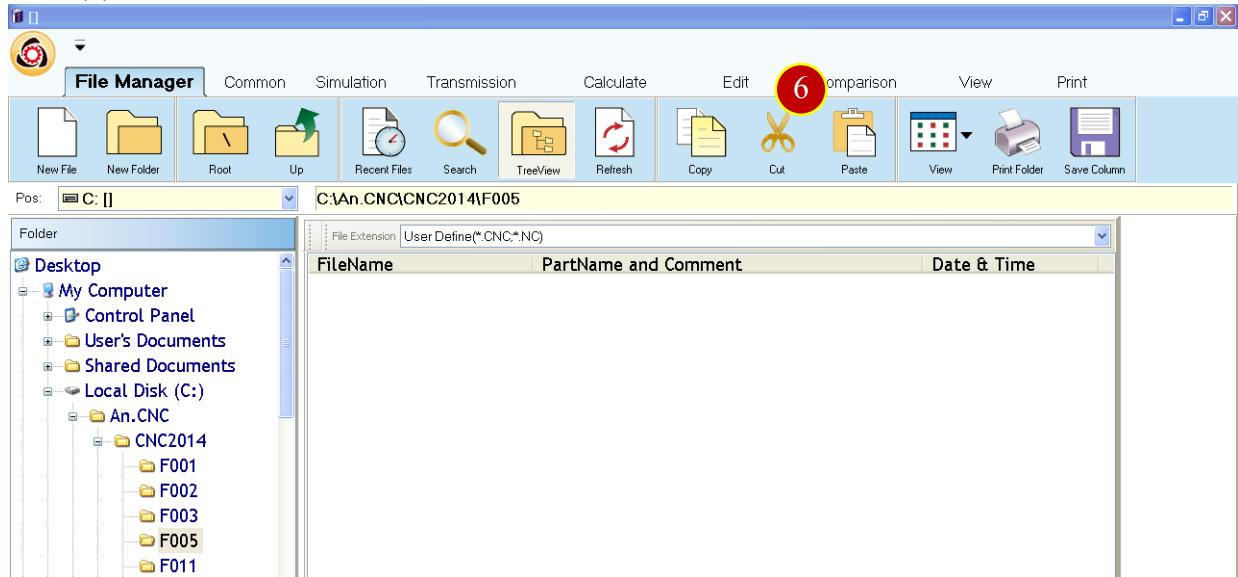
NcEditor MANUAL



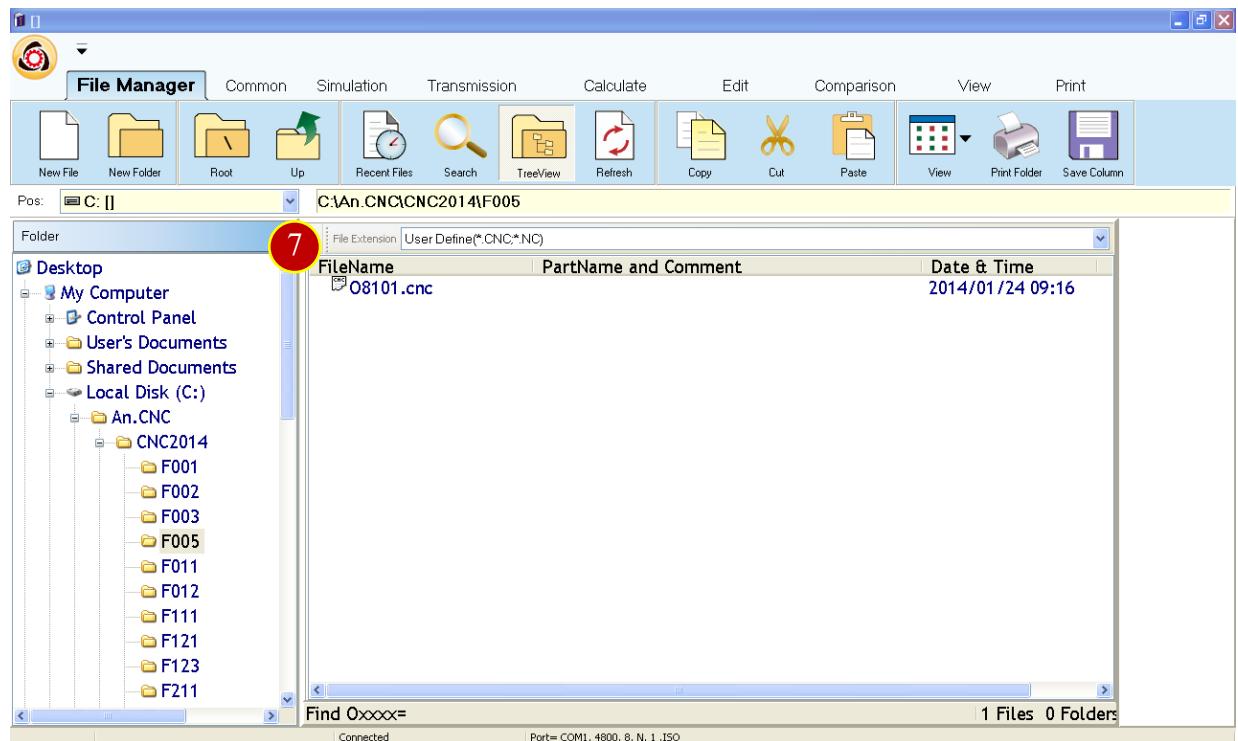
RenAn Information Technology Co., Ltd.
www.RenAn.com.tw

File Management

(6) Click 【Paste】



(7) File list displays the pasted file

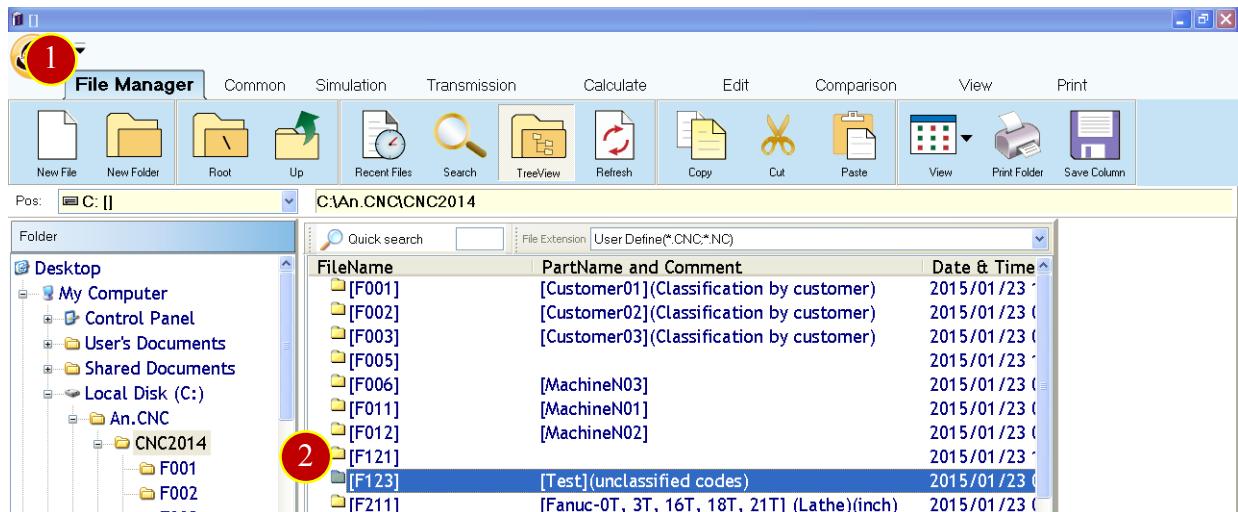


3.10.2 Move

e.g. Move F123\01012. CNC to folder F005

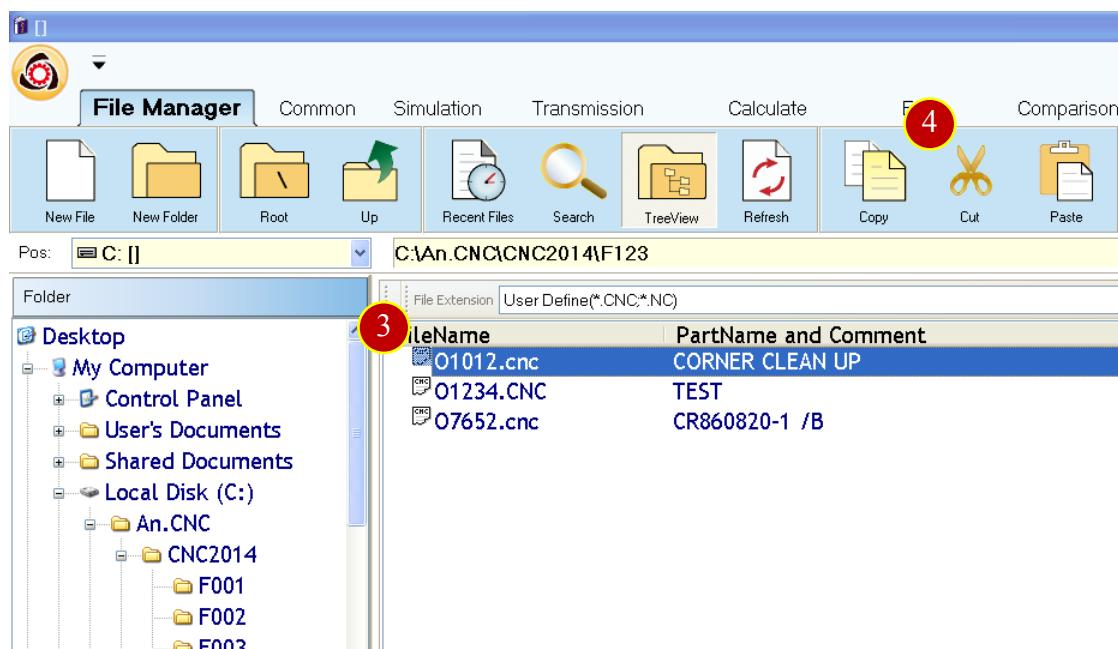
(1) Click 【File Manager】 tab

(2) Double click 【F123】 to enter



(3) Select 01012. CNC

(4) Click 【Cut】



NcEditor MANUAL

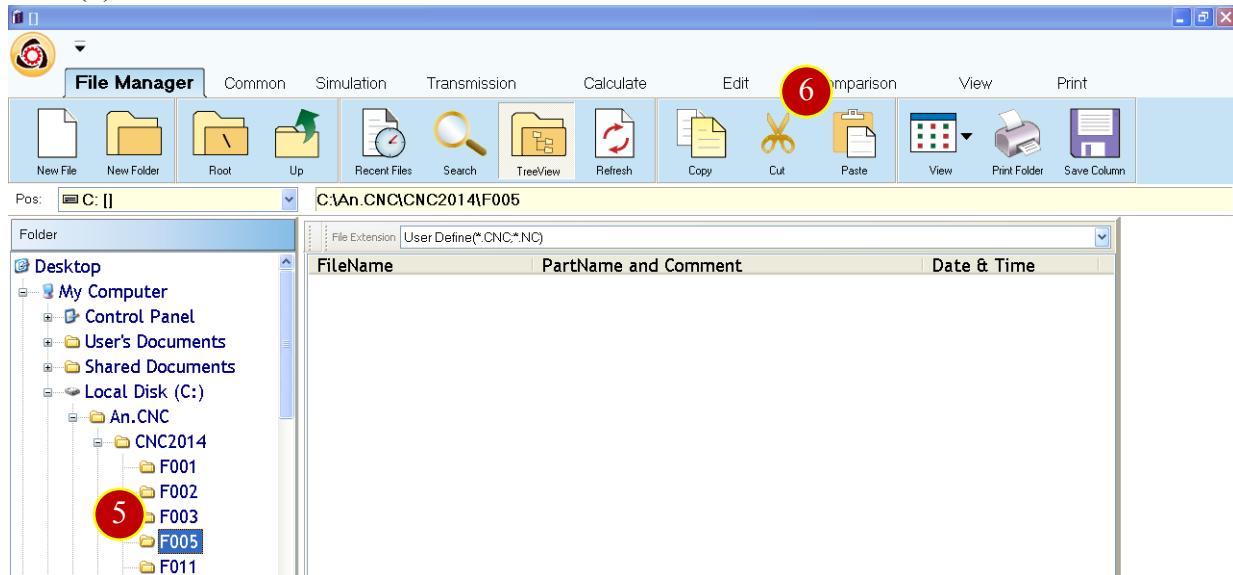


RenAn Information Technology Co., Ltd.
www.RenAn.com.tw

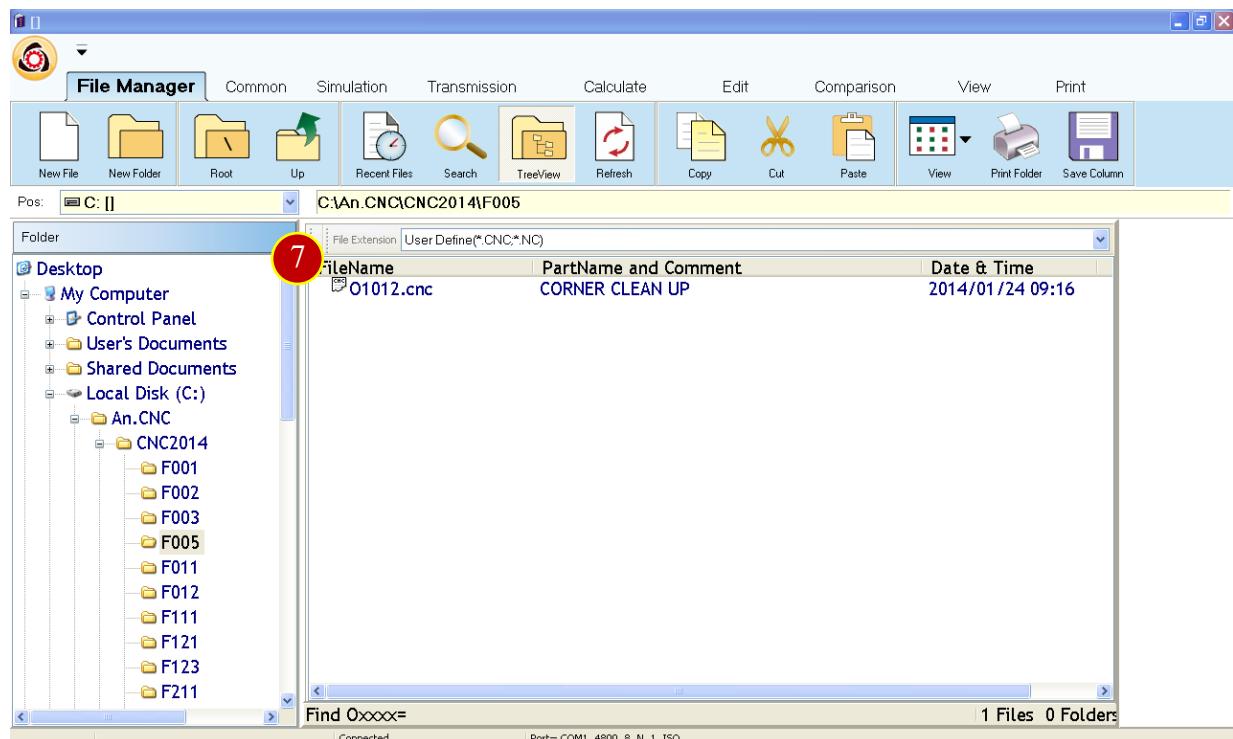
File Management

(5) Select folder 【F005】 to enter

(6) Click 【Paste】



(7) File list displays the pasted file



3.11 Merge File

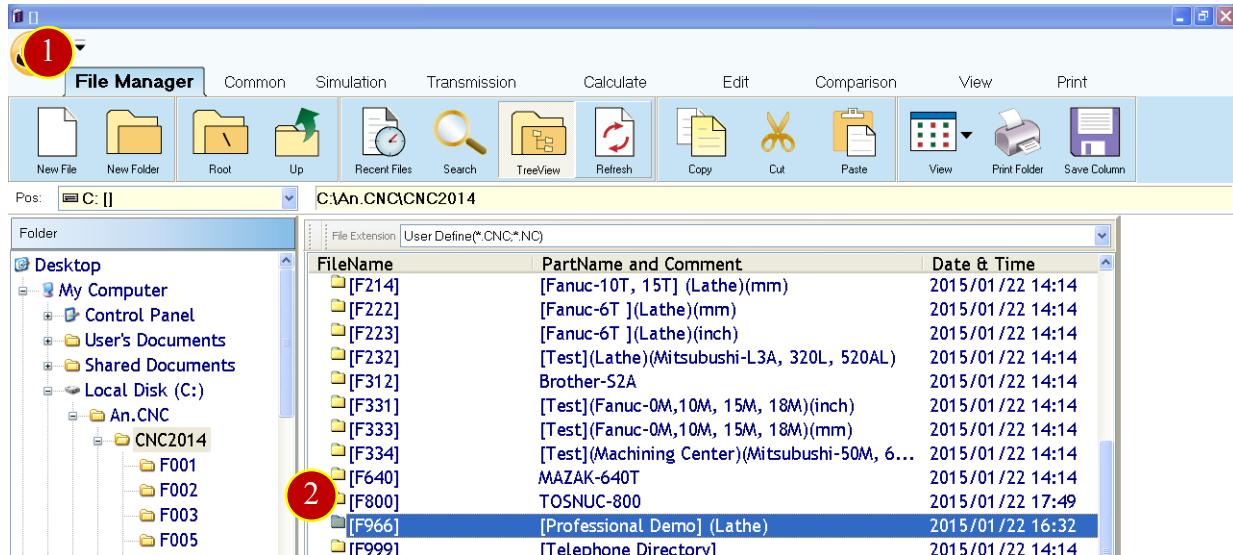
Use merge file to save time from opening, copying, pasting, etc when want to merge files which is bigger than 1MB or want to merge several files.

3.11.1 Merge File

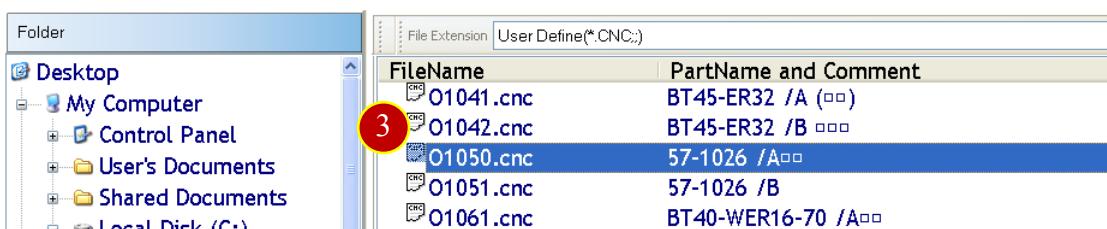
e.g. Merge F966\01050.CNC, 01051.CNC and 01065.CNC

(1) Click 【File Manager】 tab

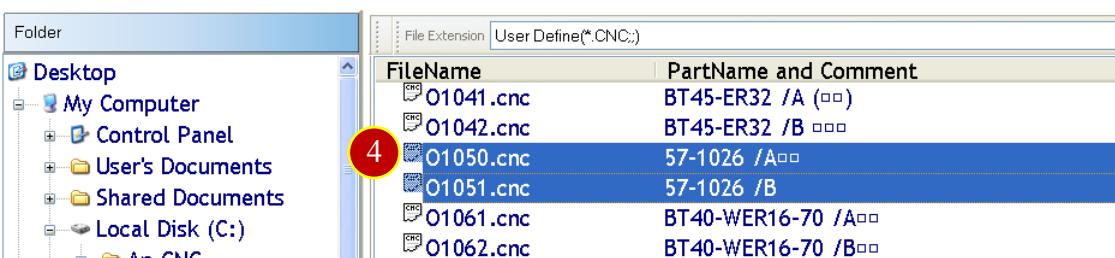
(2) Double click 【F966】 to enter



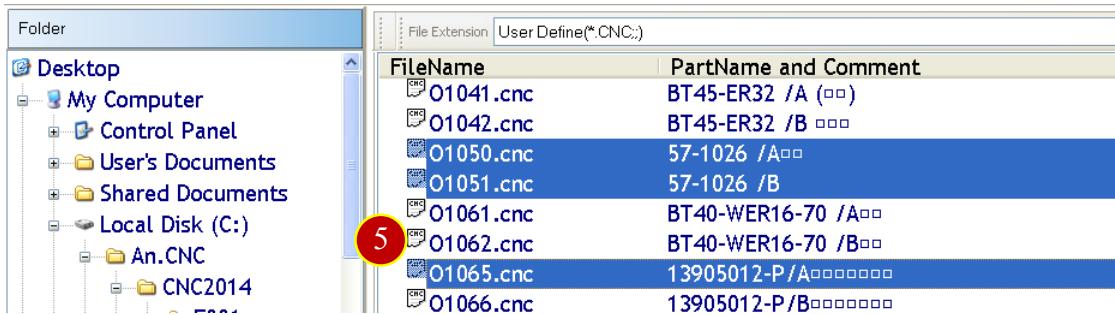
(3) Select 01050.CNC



(4) Press [Ctrl] and select 01051.CNC

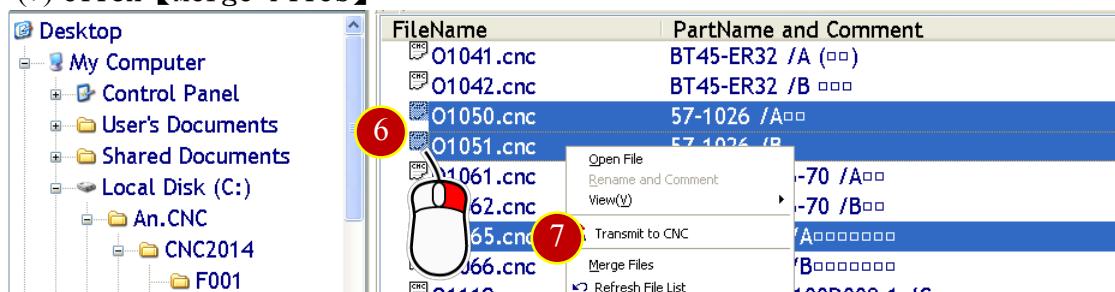


(5) And select 01065.CNC



(6) Right click at selected files

(7) Click 【Merge Files】



(8) In Merging Order, select 【01050.CNC】

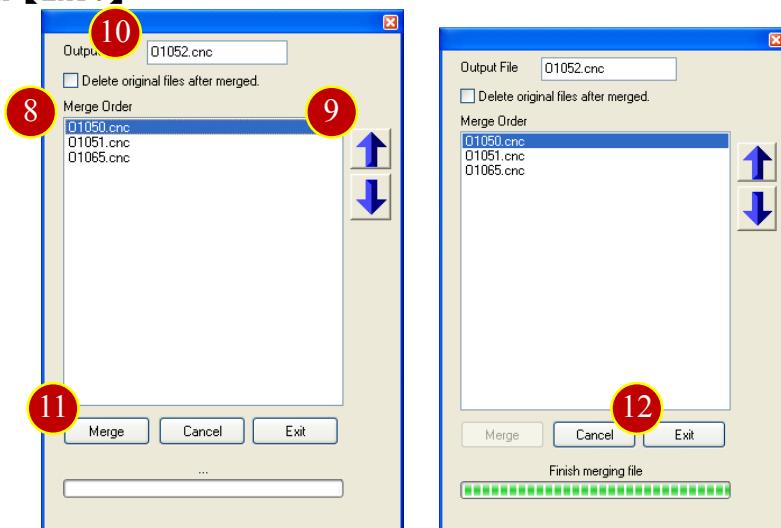
(9) Click arrow to change the merging order

*Move to the top means the location will be at the beginning of the program after merging

(10) Key in [01052.CNC] in Output File column

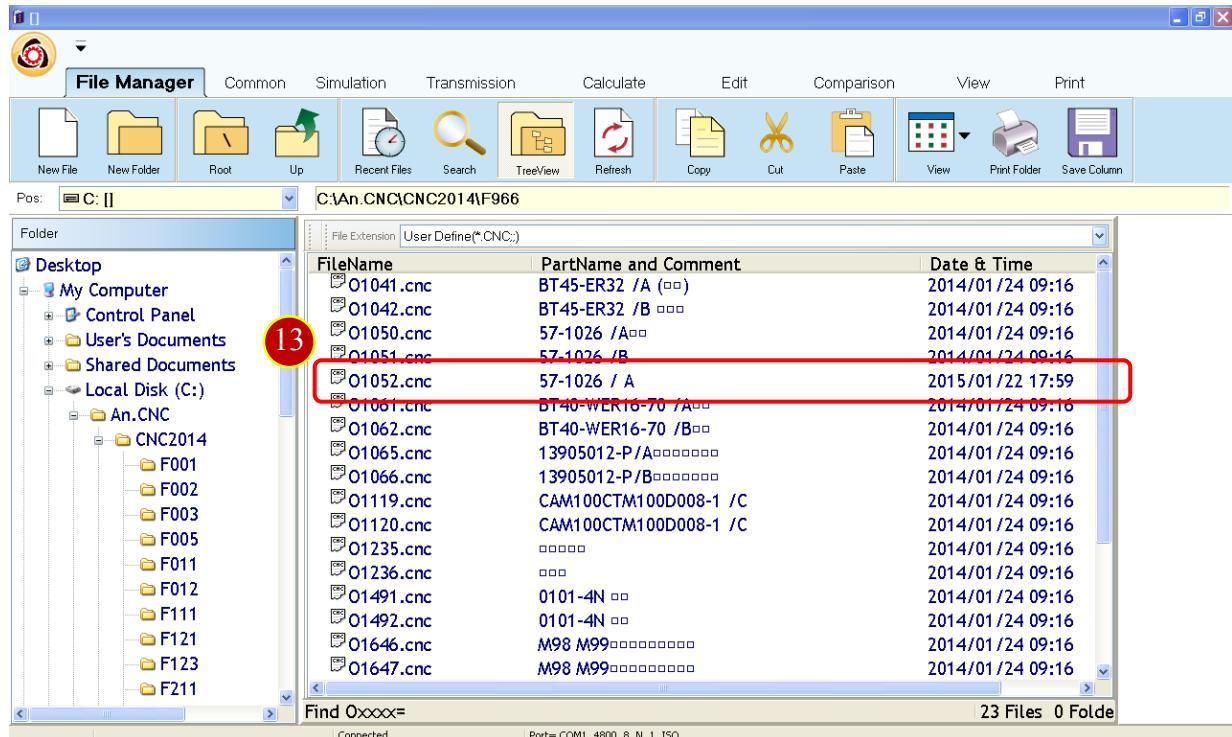
(11) Click 【Merge】

(12) Click 【Exit】



NcEditor MANUAL

(13) File list displays the merged file



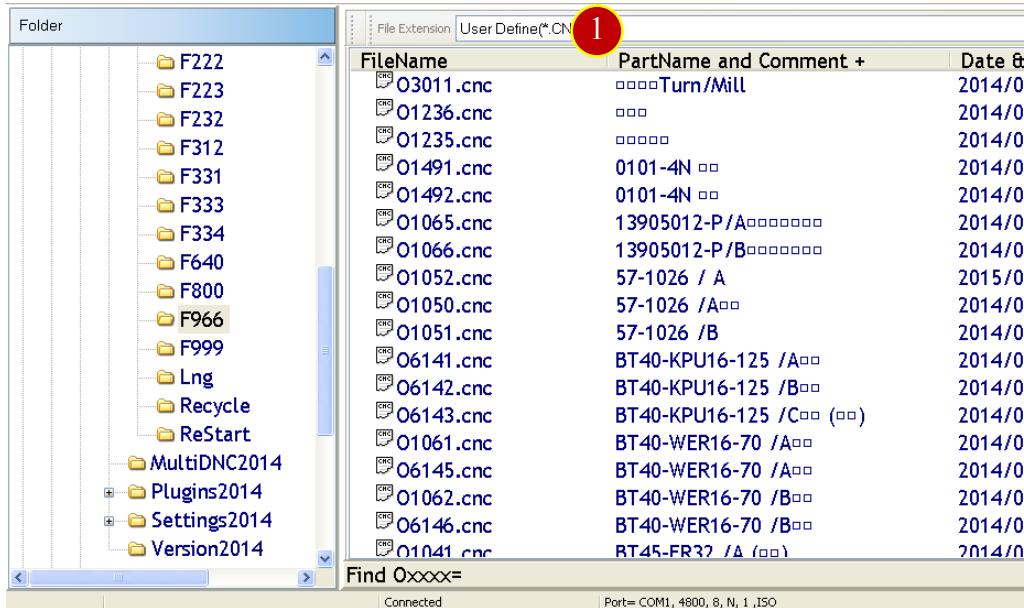
3.12 Sort Files and Save Columns Position

3.12.1 Sort Files

e.g. Based on the part name and comment

- (1) Click 【Part Name and Comment】 in file list

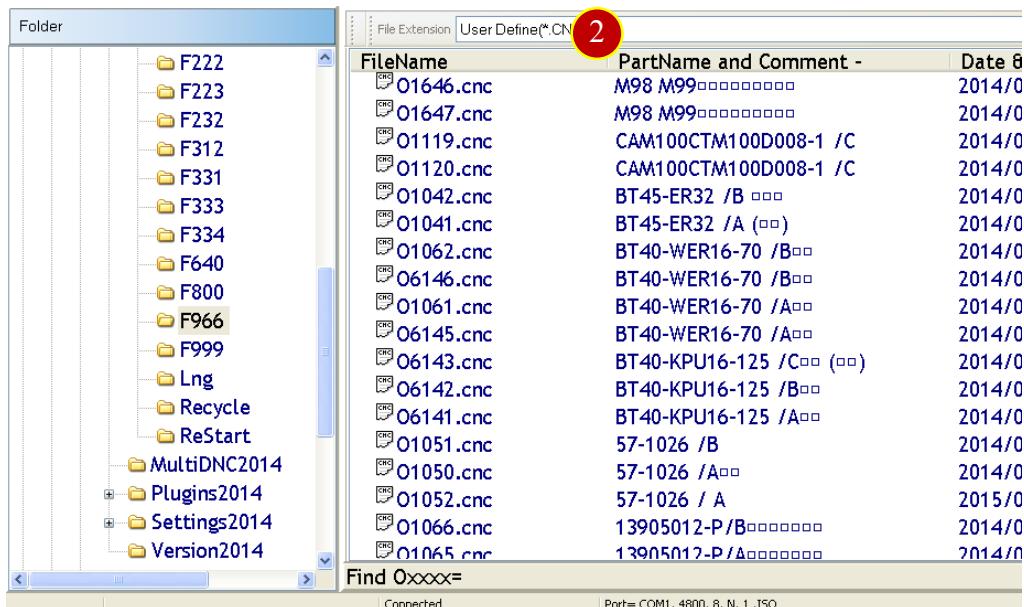
*System will automatically sort by Increment, the name in column display “+”



| FileName | PartName and Comment + | Date & |
|-----------|---------------------------|--------|
| O3011.cnc | □□□Turn/Mill | 2014/0 |
| O1236.cnc | □□□ | 2014/0 |
| O1235.cnc | □□□□□ | 2014/0 |
| O1491.cnc | 0101-4N □□ | 2014/0 |
| O1492.cnc | 0101-4N □□ | 2014/0 |
| O1065.cnc | 13905012-P/A□□□□□ | 2014/0 |
| O1066.cnc | 13905012-P/B□□□□□ | 2014/0 |
| O1052.cnc | 57-1026 / A | 2015/0 |
| O1050.cnc | 57-1026 / A□□ | 2014/0 |
| O1051.cnc | 57-1026 / B | 2014/0 |
| O6141.cnc | BT40-KPU16-125 / A□□ | 2014/0 |
| O6142.cnc | BT40-KPU16-125 / B□□ | 2014/0 |
| O6143.cnc | BT40-KPU16-125 / C□□ (□□) | 2014/0 |
| O1061.cnc | BT40-WER16-70 / A□□ | 2014/0 |
| O6145.cnc | BT40-WER16-70 / A□□ | 2014/0 |
| O1062.cnc | BT40-WER16-70 / B□□ | 2014/0 |
| O6146.cnc | BT40-WER16-70 / B□□ | 2014/0 |
| O1041.cnc | RT45-FR32 / A (□□) | 2014/0 |

- (2) Click 【Part Name and Comment】 once again in file list

*System will sort by minus, the name in column display “-”



| FileName | PartName and Comment - | Date & |
|-----------|---------------------------|--------|
| O1646.cnc | M98 M99□□□□□□ | 2014/0 |
| O1647.cnc | M98 M99□□□□□□ | 2014/0 |
| O1119.cnc | CAM100CTM100D008-1 / C | 2014/0 |
| O1120.cnc | CAM100CTM100D008-1 / C | 2014/0 |
| O1042.cnc | BT45-ER32 / B □□ | 2014/0 |
| O1041.cnc | BT45-ER32 / A (□□) | 2014/0 |
| O1062.cnc | BT40-WER16-70 / B□□ | 2014/0 |
| O6146.cnc | BT40-WER16-70 / B□□ | 2014/0 |
| O1061.cnc | BT40-WER16-70 / A□□ | 2014/0 |
| O6145.cnc | BT40-WER16-70 / A□□ | 2014/0 |
| O6143.cnc | BT40-KPU16-125 / C□□ (□□) | 2014/0 |
| O6142.cnc | BT40-KPU16-125 / B□□ | 2014/0 |
| O6141.cnc | BT40-KPU16-125 / A□□ | 2014/0 |
| O1051.cnc | 57-1026 / B | 2014/0 |
| O1050.cnc | 57-1026 / A□□ | 2014/0 |
| O1052.cnc | 57-1026 / A | 2015/0 |
| O1066.cnc | 13905012-P/B□□□□□ | 2014/0 |
| O1065.cnc | 13905012-P/A□□□□□ | 2014/0 |

3.12.2 Save Columns Position

After saving the position of the column, there's no need to adjust the column next time.

e.g. Adjust the file name, comment, date and time, digit number and save the column position

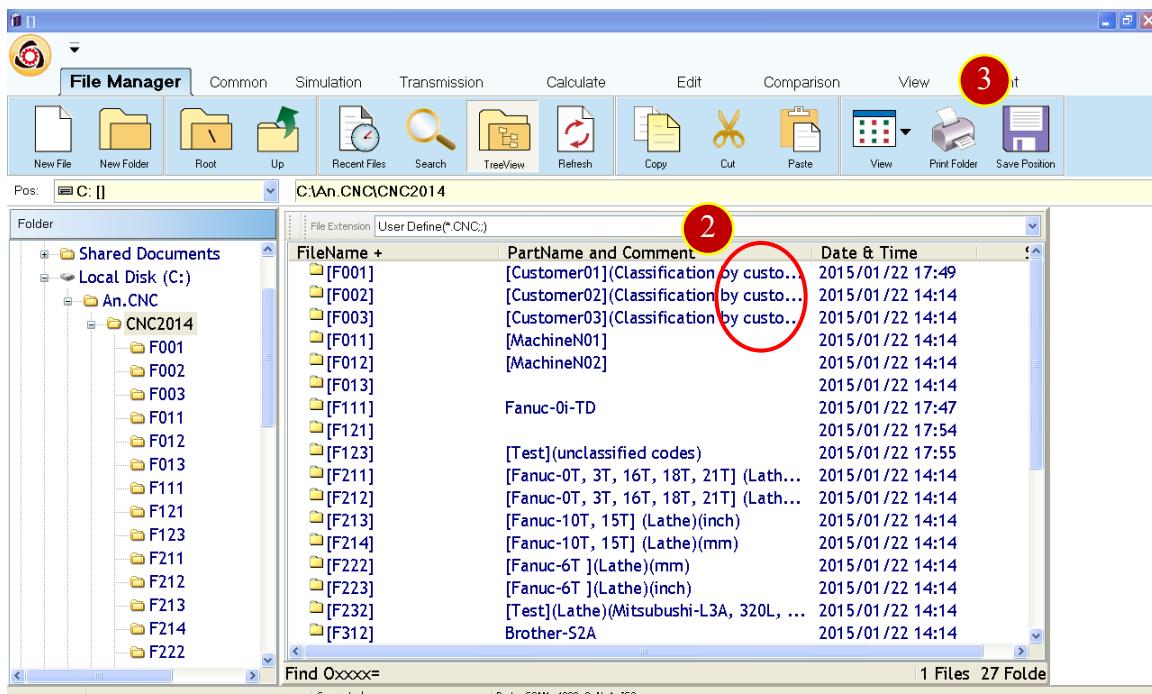
(1) Put the cursor on the line between 【Part Name and Comment】 and 【Date and Time】 ; it will appear “

The screenshot shows the NcEditor interface with the 'File Manager' tab selected. A red circle labeled '1' highlights the boundary between the 'PartName and Comment' column header and the 'Date & Time' column header. The 'PartName and Comment' column contains entries like '[Customer01](Classification by customer)' and '[Customer02](Classification by customer)'. The 'Date & Time' column contains dates and times such as '2015/05/06 08:54'.

(2) Press and drag to adjust it to proper width

*When the description is too long and the width of the column is too short, extra words will be displayed as “...”

(3) Click 【Save Position】 to save the width of the column



4. Program Edit

The operation of edit NC program, debug and command set

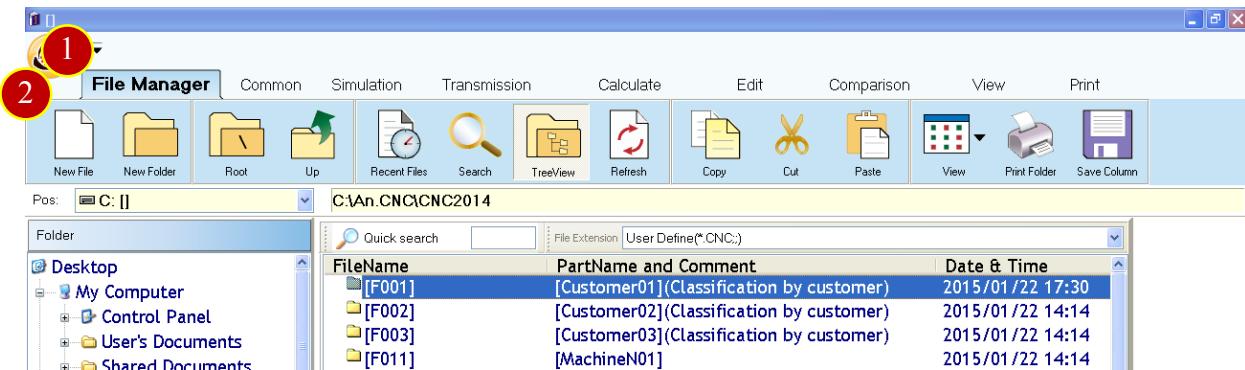
4.1 Command Set

4.1.1 Paste All

e.g. Add Program Start from Command Set

(1) Click 【File Manager】 tab

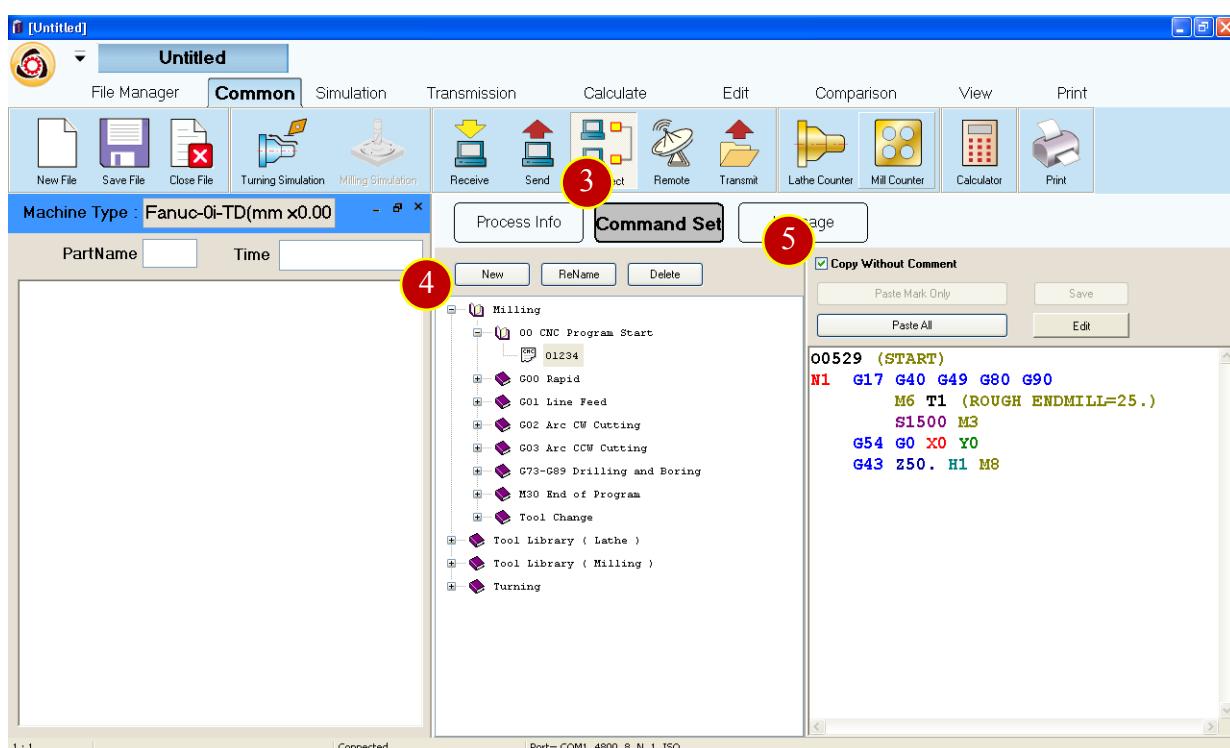
(2) Click 【New File】



(3) Click 【Command Set】

(4) Select 【Milling】 > 【00 Program Start】 > 【01234】

(5) Check 【Copy Without Comment】



NcEditor MANUAL

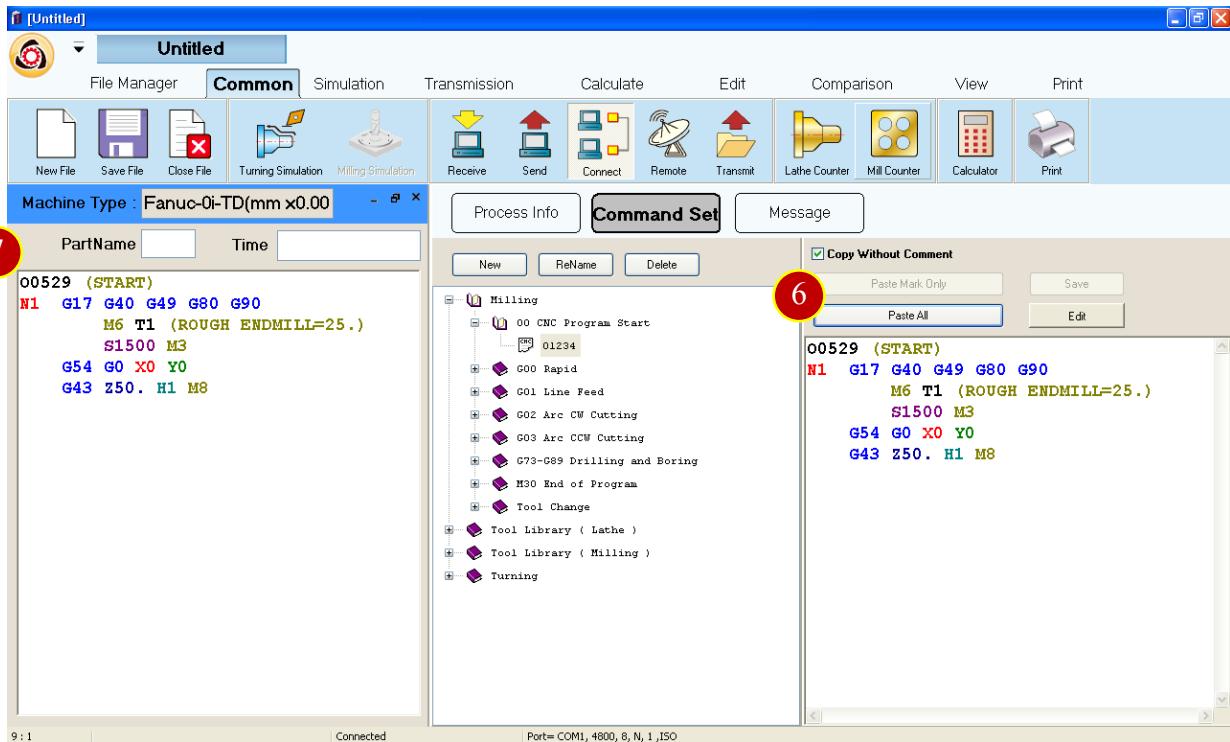


RenAn Information Technology Co., Ltd.
www.RenAn.com.tw

Program Edit

(6) Click 【Paste All】

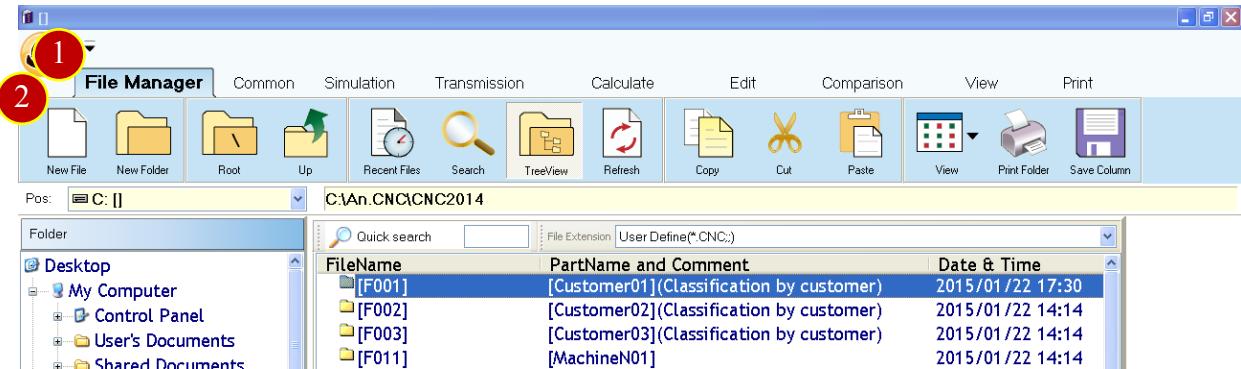
(7) Edit area displays paste program



4.1.2 Select Block Paste

e.g. Settings of add part of program start from command set

- (1) Click 【File Manager】 tab
- (2) Click 【New File】



- (3) Key in program number in edit area

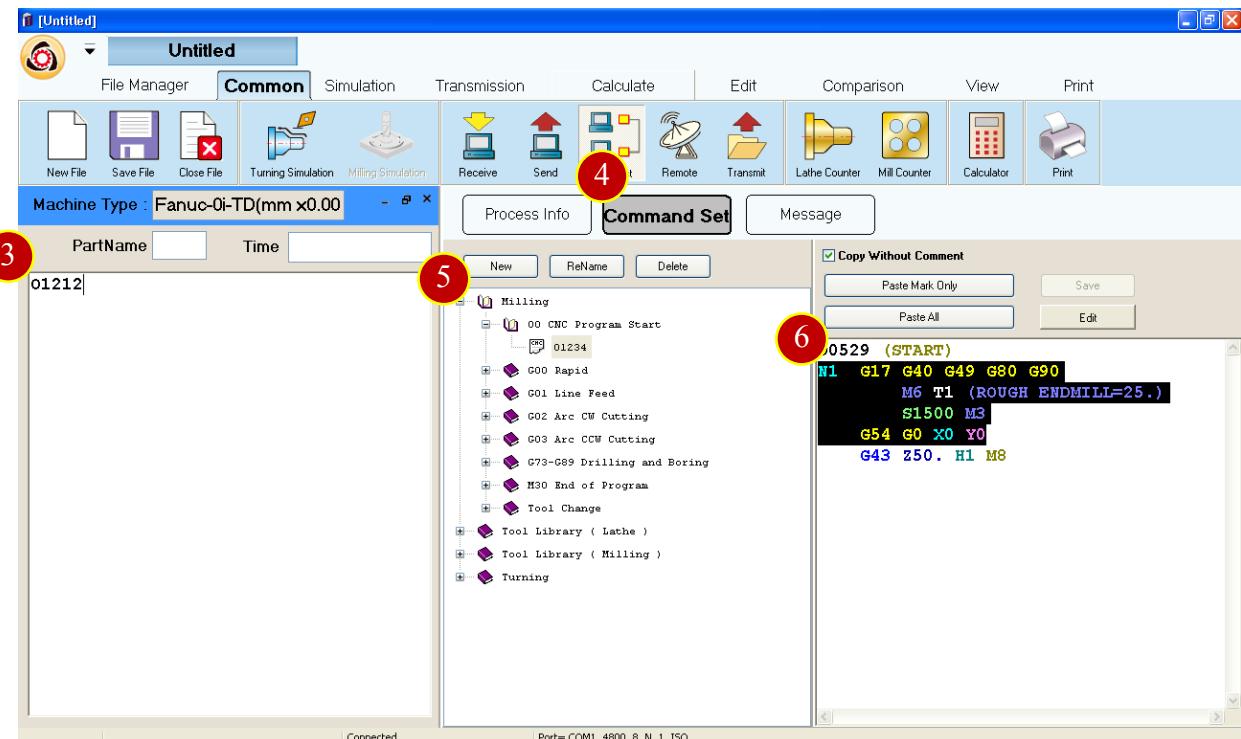
e.g. 01212

- (4) Click 【Command Set】

- (5) Select 【Milling】 > 【00 CNC Program Start】 > 【01234】

- (6) Select Block of Program

e.g. N1 G17 G40 G49...



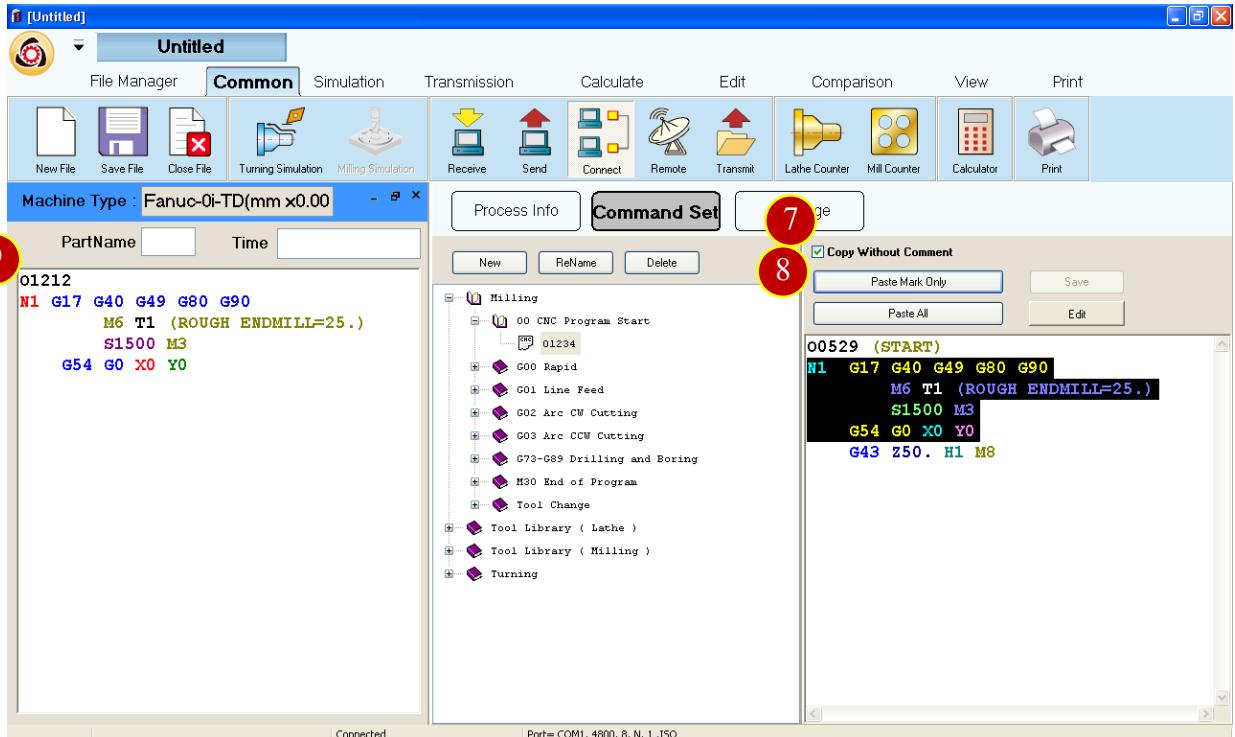
NcEditor MANUAL



RenAn Information Technology Co., Ltd.
www.RenAn.com.tw

Program Edit

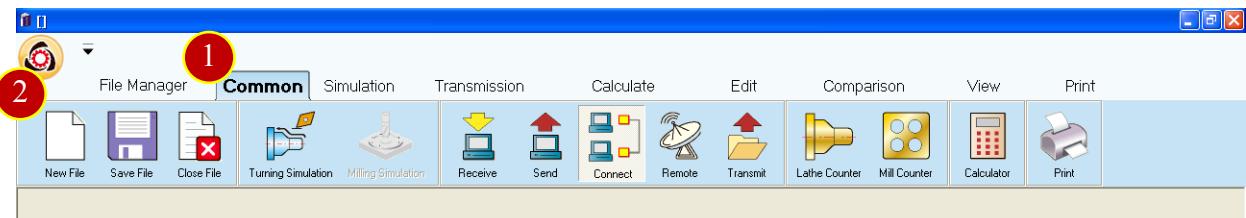
- (7) Check 【Copy Without Comment】
- (8) Click 【Paste Mark Only】
- (9) Edit area displays paste program



4.1.3 Rename Command Set

e.g. Rename from 【00 program start】 > 【01234】 to 【0 program start】

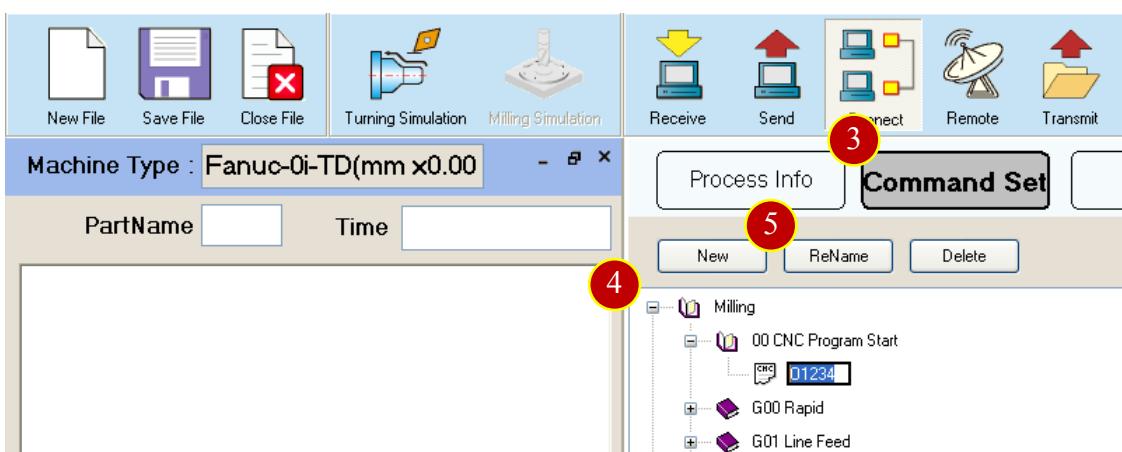
- (1) Click 【Common】 tab
- (2) Click 【New File】



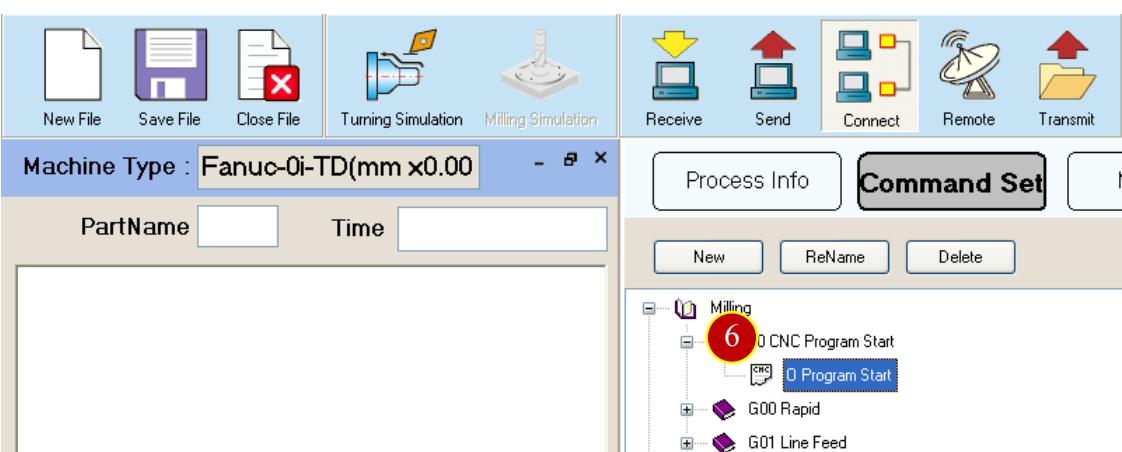
- (3) Click 【Command Set】

(4) Select 【Milling】 > 【00 CNC Program Start】 > 【01234】

- (5) Click 【Rename】



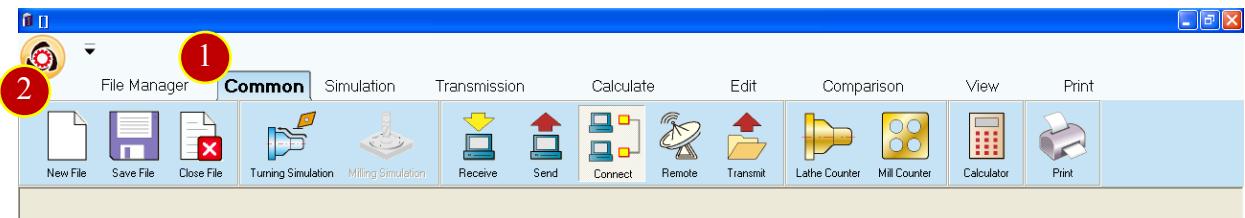
- (6) Key in 【0 Program Start】 and press [Enter]



4.1.4 Add Command Set

e.g. Add 【Optional cycle】after 【End of Program】

- (1) Click 【Common】 tab
- (2) Click 【New File】

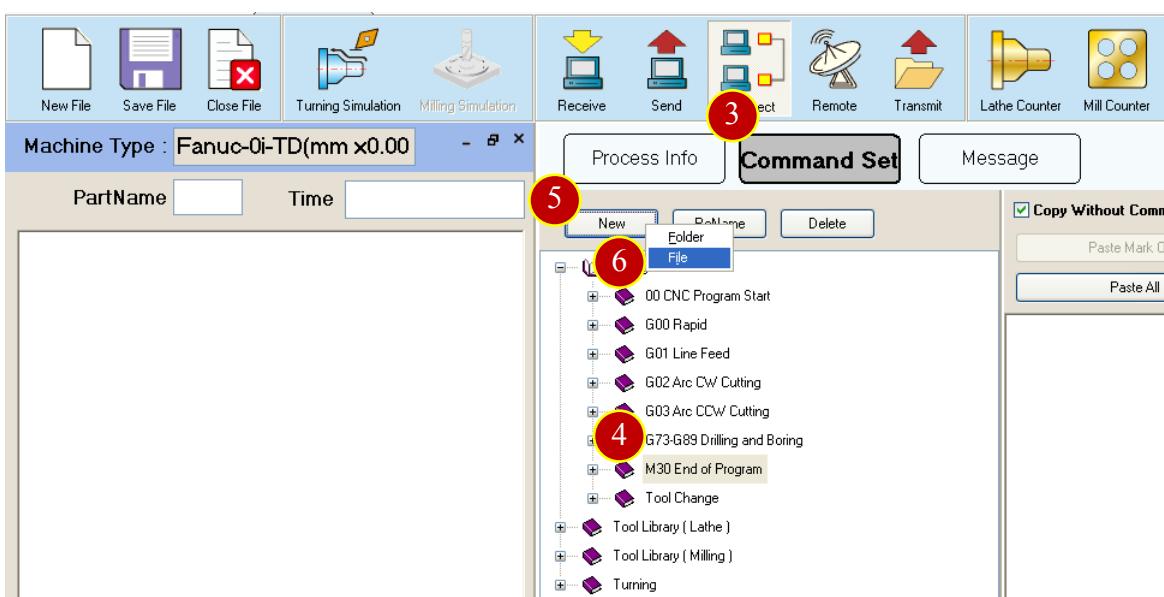


- (3) Click 【Command Set】

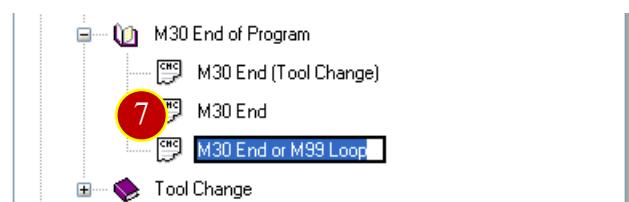
(4) Select 【M30 End of Program】

- (5) Click 【New】

- (6) Click 【File】



- (7) Key in [M30 End or M99 Loop] and press Enter



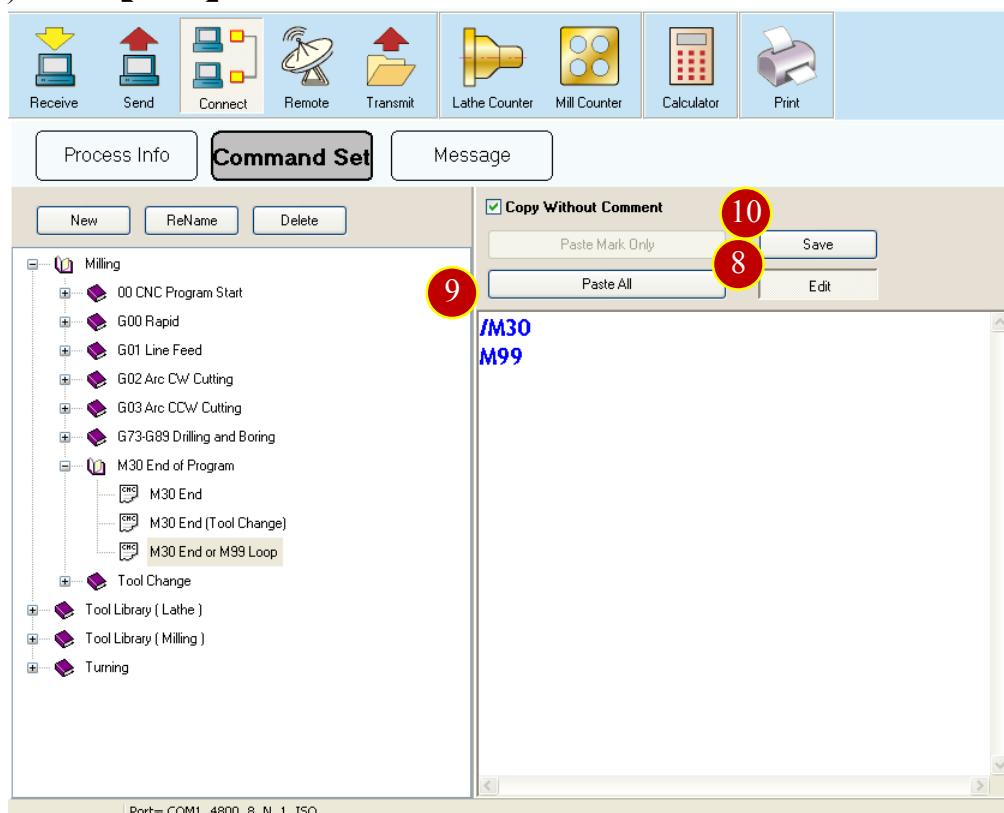
NcEditor MANUAL



RenAn Information Technology Co., Ltd.
www.RenAn.com.tw

Program Edit

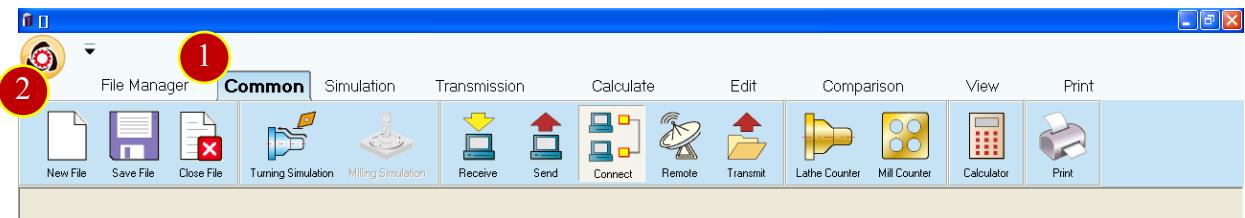
- (8) Click 【Edit】
- (9) Key in NC code
e.g. /M30...
- (10) Click 【Save】



4.1.5 Delete Command Set

e.g. Delete command set 【0 Program Start】 of 【Program Start】

- (1) Click 【Common】 tab
- (2) Click 【New File】

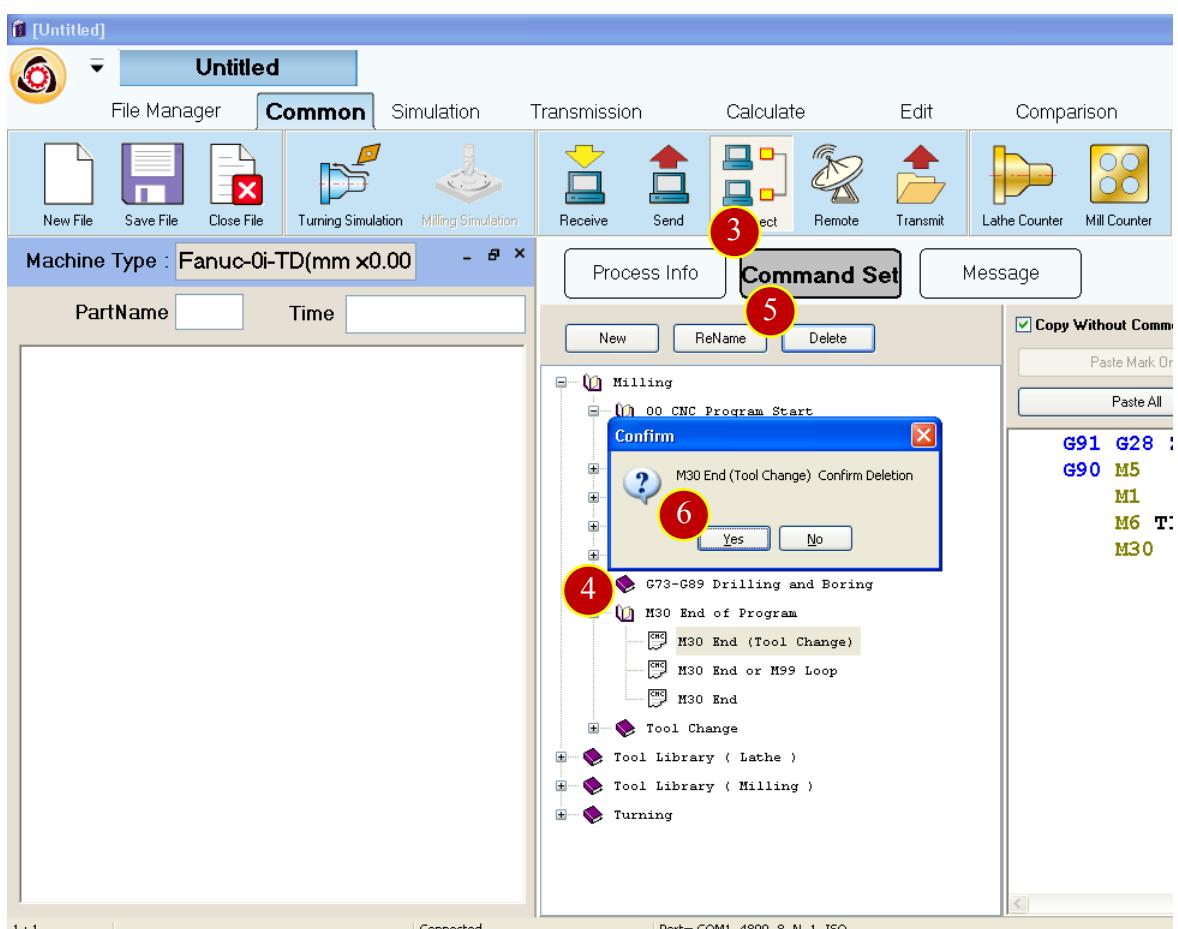


- (3) Click 【Command Set】

(4) Select 【M30 End of Program】 > 【M30 End(Tool Change)】

- (5) Click 【Delete】

- (6) Click 【Yes】 and complete



4.2 Save File, Save As and Refresh

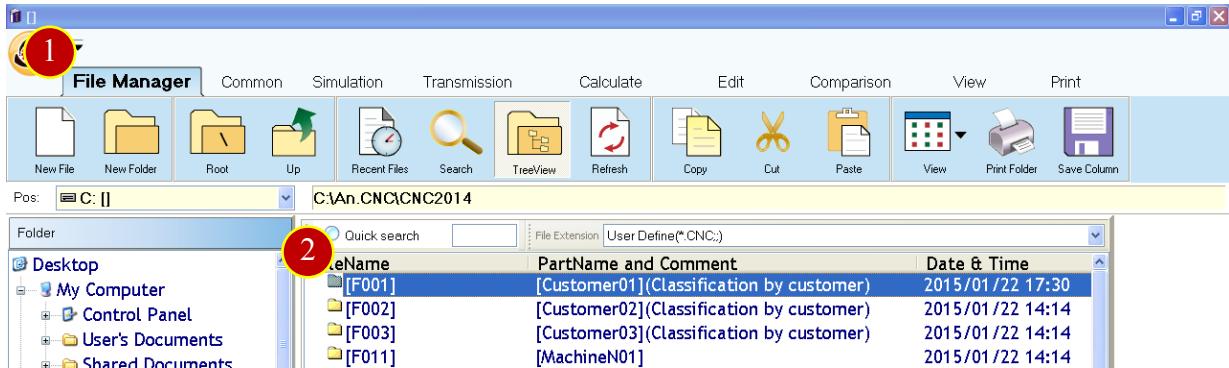
4.2.1 Save

Save current CNC program

e.g. Edit F001\00701.CNC and save file

(1) Click 【File Manager】 tab

(2) Double click 【F001】 to enter folder



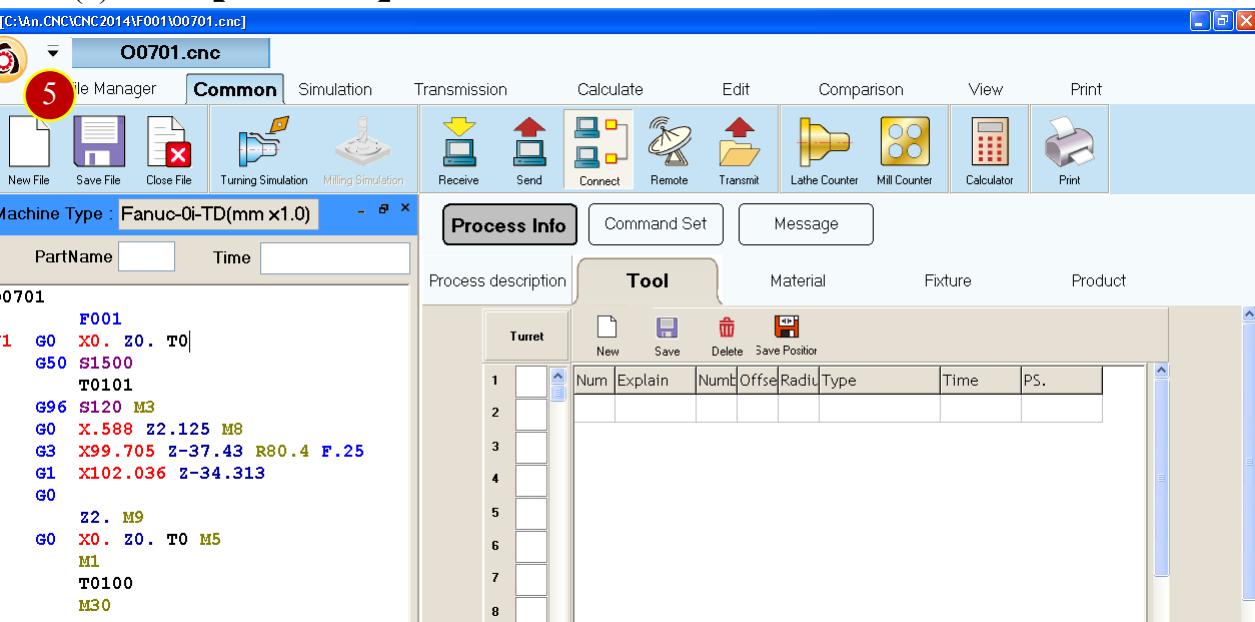
(3) Double click 【00701.CNC】 file to open and edit



(4) Edit program

e.g. Add comment (ST80-07/A)

(5) Click 【Save File】



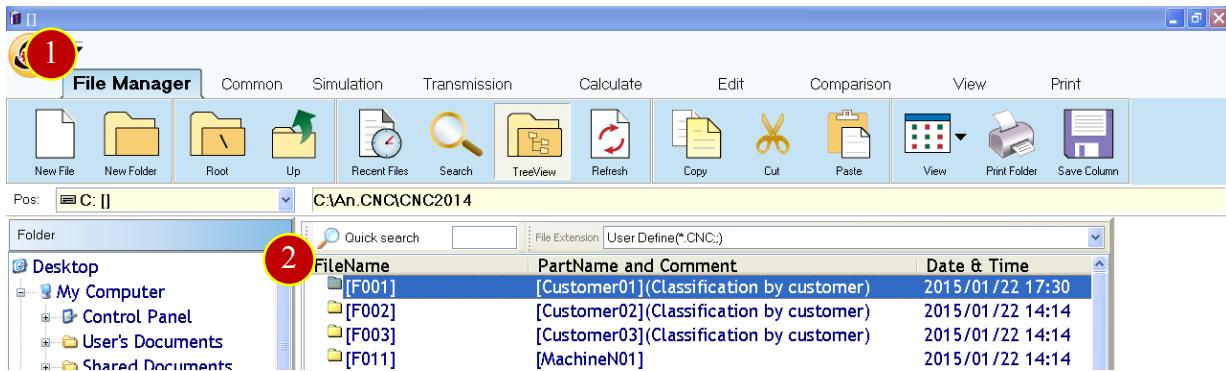
4.2.2 Save As

Save current CNC program to another file

e.g. Save F001\00701.CNC to F002

(1) Click **【File Manager】**

(2) Double click **【F001】** to enter



(3) Double click 00701.CNC to open and edit

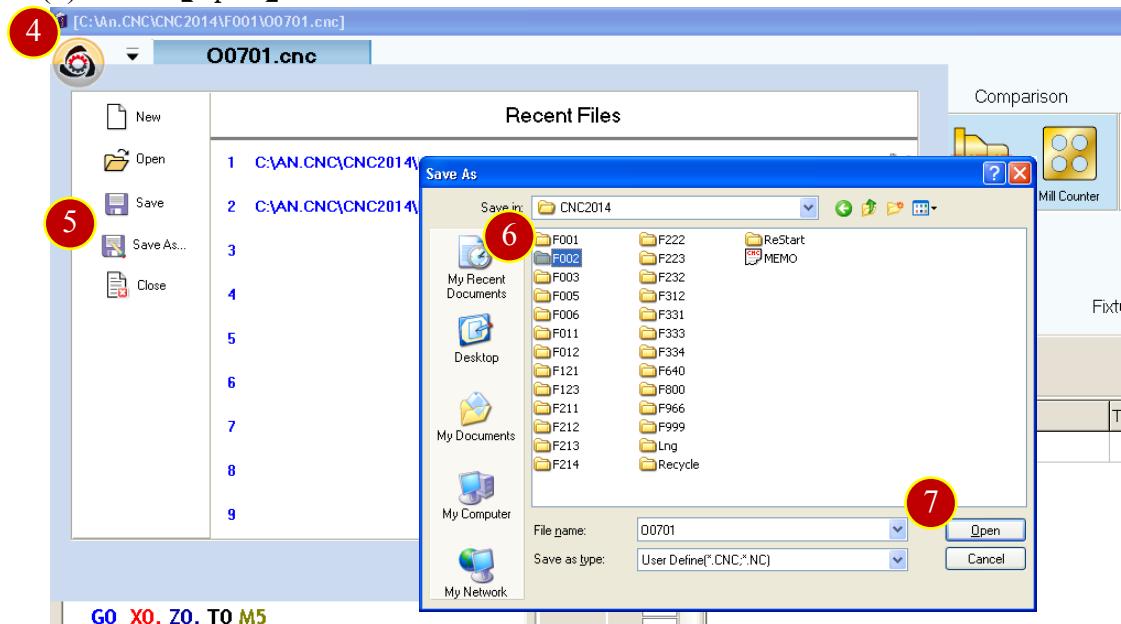


(4) Click RenAn Logo

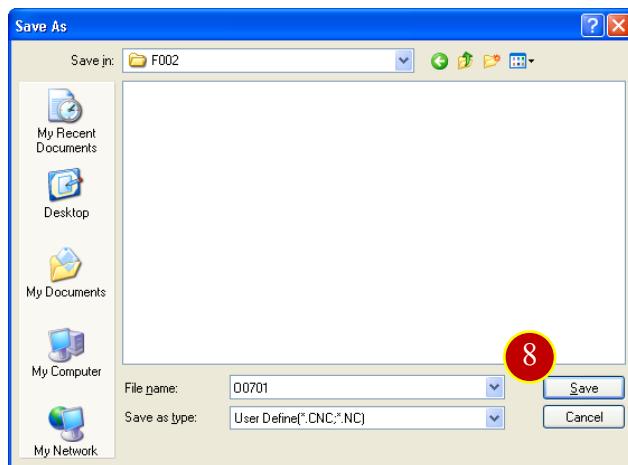
(5) Select **【Save As】**

(6) Select **【F002】**

(7) Click **【Open】**



(8) Click 【Save】



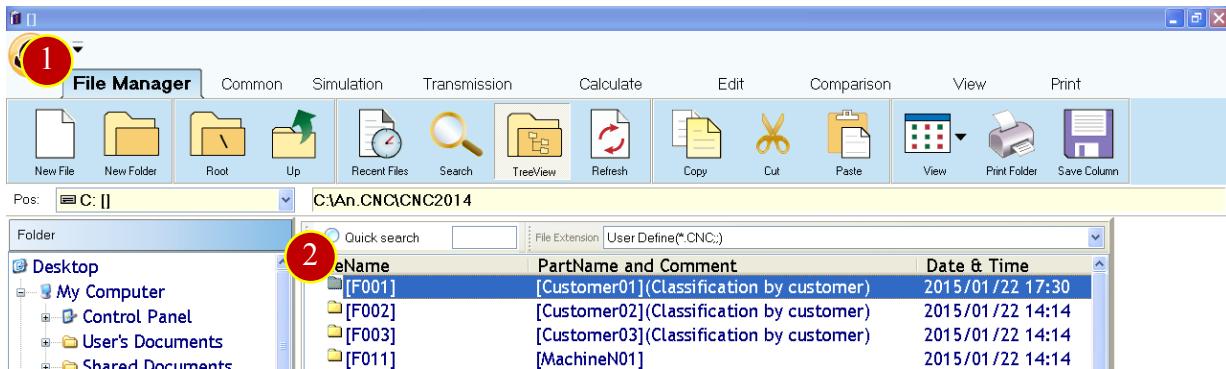
4.2.3 Refresh

Refresh current CNC program, include color, font and size

e.g. Refresh

(1) Click 【File Manager】 tab

(2) Double click 【F001】 to enter

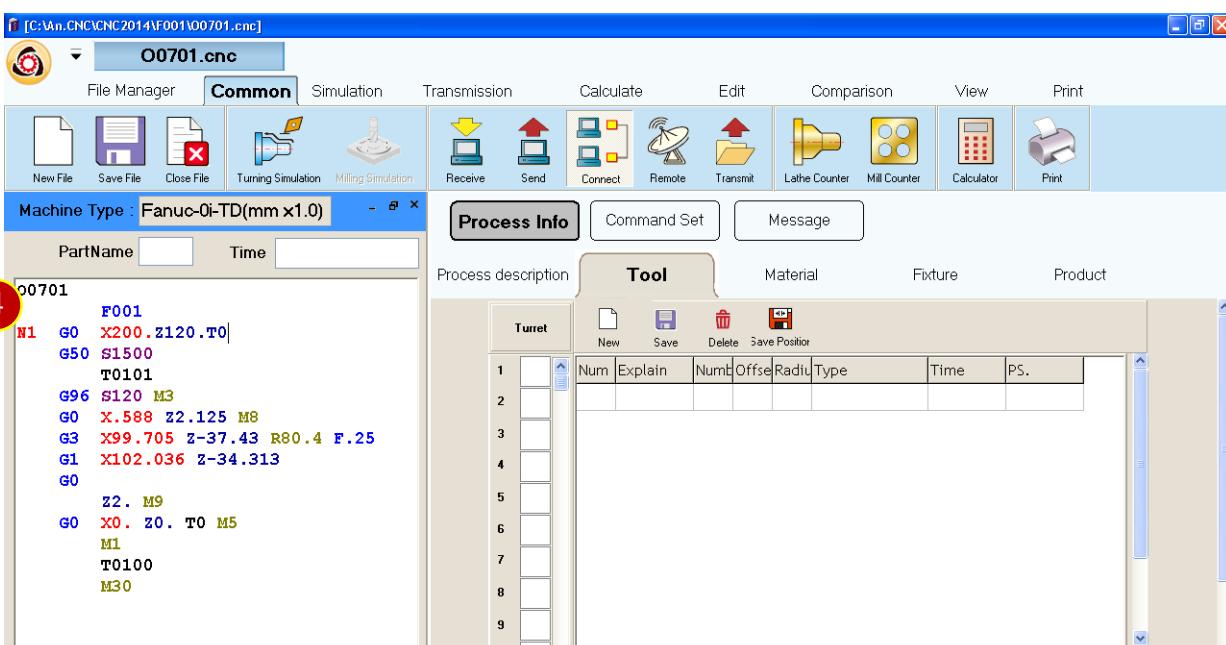


(3) Double click 【00701.CNC】 to open and edit status



(4) Modify CNC program

e.g. N1 G0 X200. Z120. T0



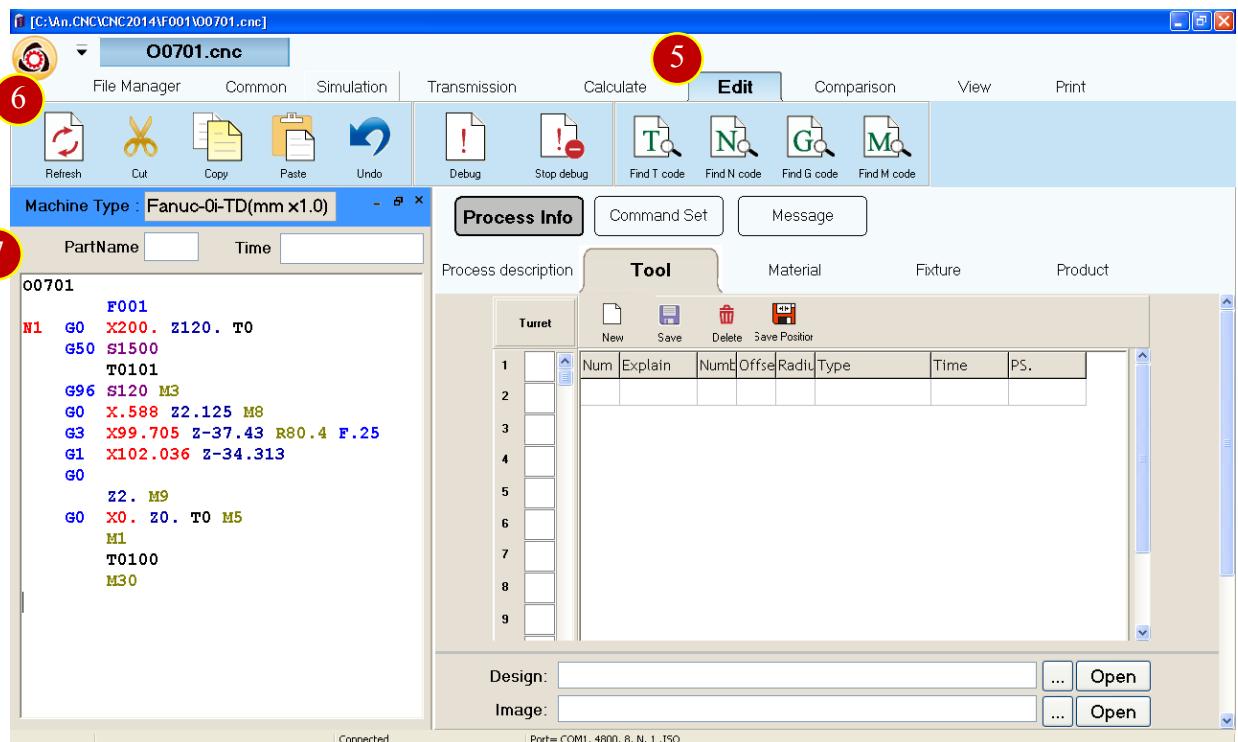
NcEditor MANUAL



RenAn Information Technology Co., Ltd.
www.RenAn.com.tw

Program Edit

- (5) Click 【Edit】 tab
- (6) Click 【Refresh】
- (7) The program color, size, and position modify finish

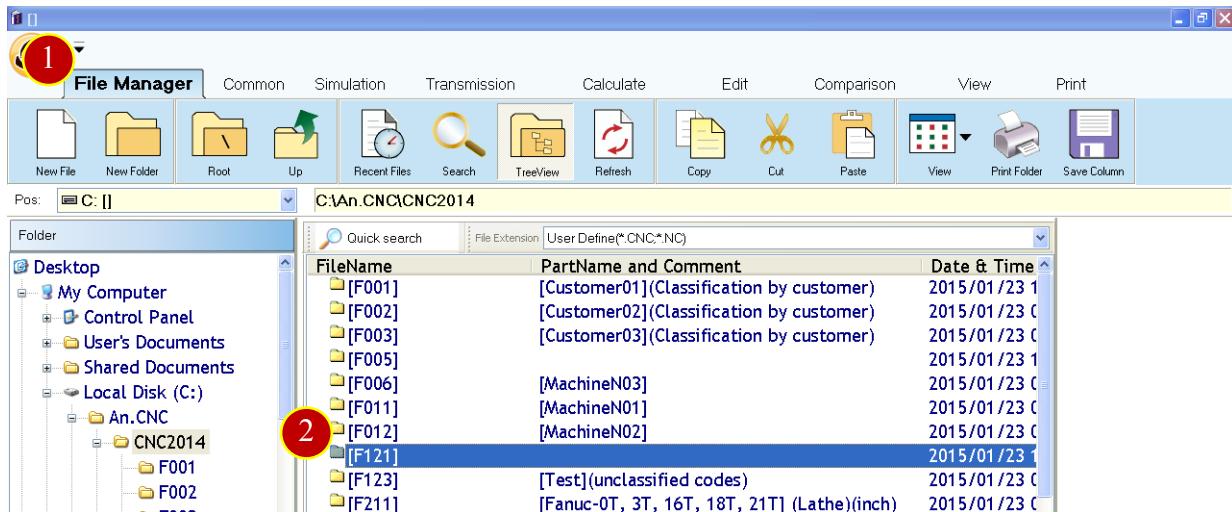


4.3 Copy, Move and Undo

4.3.1 Copy

e.g. Reuse part of the content in F121\03802.CNC

- (1) Click 【File Manager】 tab
- (2) Double click folder 【F121】



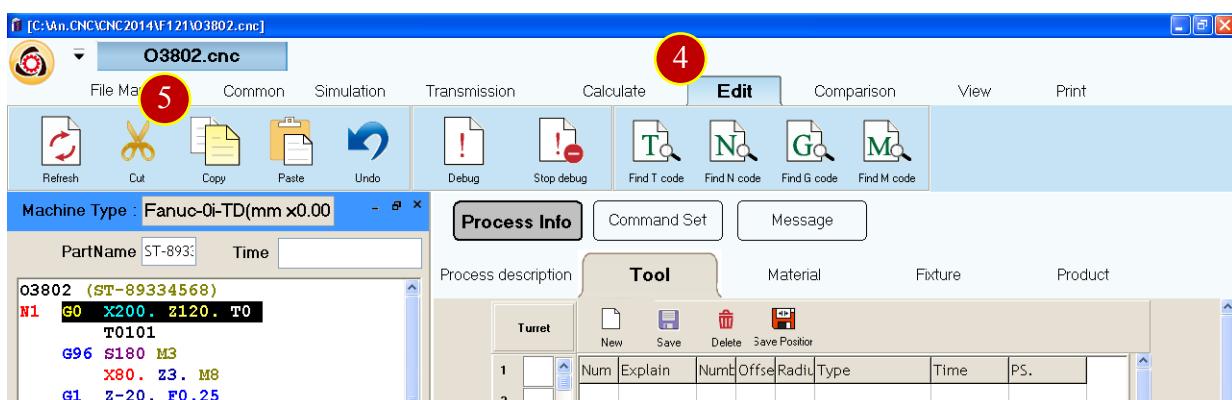
- (3) Double click 【03802.CNC】 edit file



- (4) Select a block of the program

e.g. G0 X200. Z120. T0

- (5) Click 【Edit】 tab
- (6) Click 【Copy】



NcEditor MANUAL



RenAn Information Technology Co., Ltd.
www.RenAn.com.tw

Program Edit

(7) Key in [N9] after process N8

```

Machine Type : Fanuc-0i-TD(mm x0.00)
PartName ST-8933 Time

G71 U2. R0.3
G71 P7 Q8 U-.2 W.1 F.25
N7 G0 X61.064
G1 Z0.000
G3 X20.523 Z-9.082 R49.200
G2 X16.000 Z-11.829 R2.800
G1 Z-20.800
N8 G1 U-1.
G0 Z20.
X63.0 Z120. T0
M1
N9 G0 X63.0 Z120. T0
T0303
G96 S180 M3
G0 X60.533 Z1.000
G1 Z0 F.2
G3 X19.877 Z-9.074 R49.600
G2 X16.000 Z-11.429 R2.400
G1 Z-20.400 M9
G0 U-.5 Z20. M5

```

(8) Click 【Paste】

(9) Display the copied program

```

Machine Type : Fanuc-0i-TD(mm x0.00)
PartName ST-8933 Time

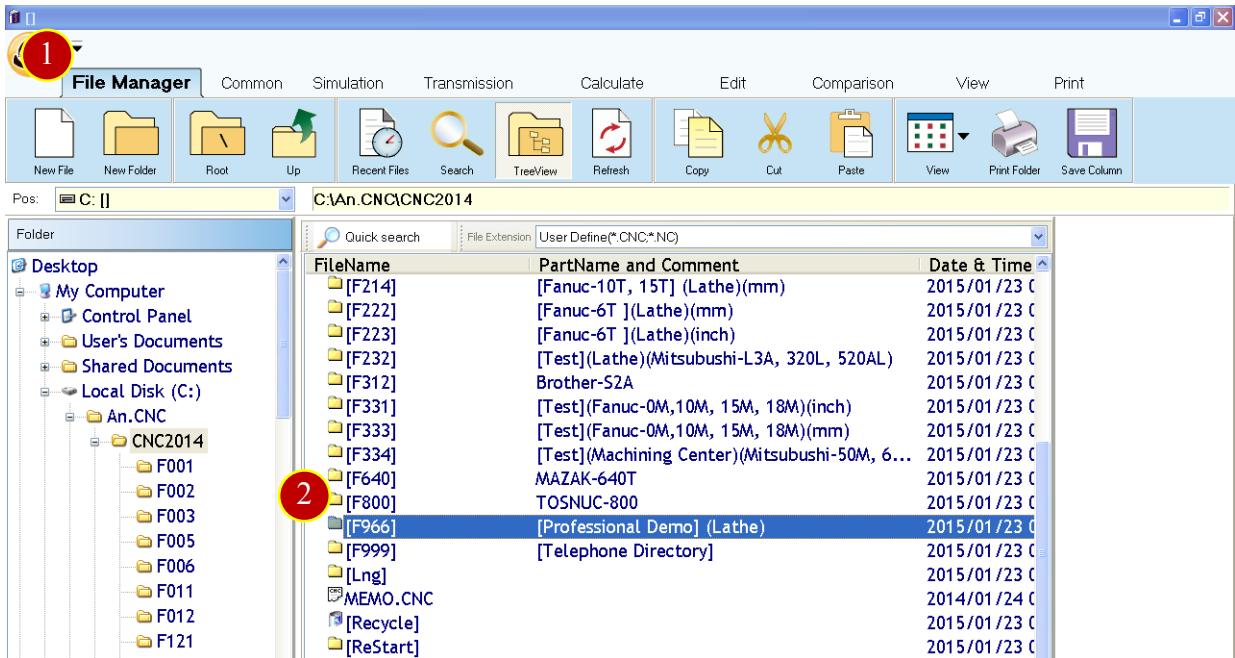
G71 U2. R0.3
G71 P7 Q8 U-.2 W.1 F.25
N7 G0 X61.064
G1 Z0.000
G3 X20.523 Z-9.082 R49.200
G2 X16.000 Z-11.829 R2.800
G1 Z-20.800
N8 G1 U-1.
G0 Z20.
X63.0 Z120. T0
M1
N9 G0 X200. Z120. T0
T0303
G96 S180 M3
G0 X60.533 Z1.000
G1 Z0 F.2
G3 X19.877 Z-9.074 R49.600
G2 X16.000 Z-11.429 R2.400
G1 Z-20.400 M9
G0 U-.5 Z20. M5

```

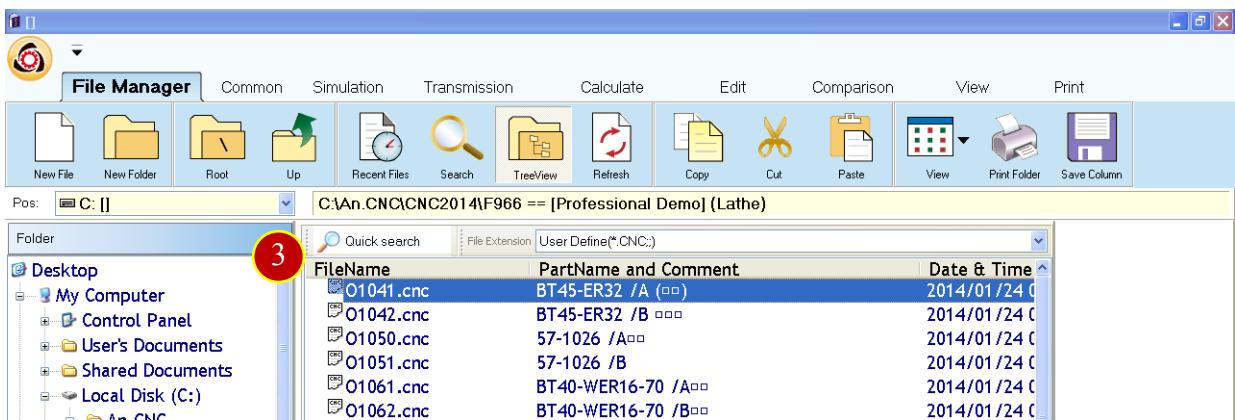
4.3.2 Move

e.g. Move the content of F966\01041.CNC from N2 to N3

- (1) Click 【File Manager】 tab
- (2) Double click folder 【F966】 to enter



- (3) Double click 【01041.CNC】



NcEditor MANUAL



RenAn Information Technology Co., Ltd.
www.RenAn.com.tw

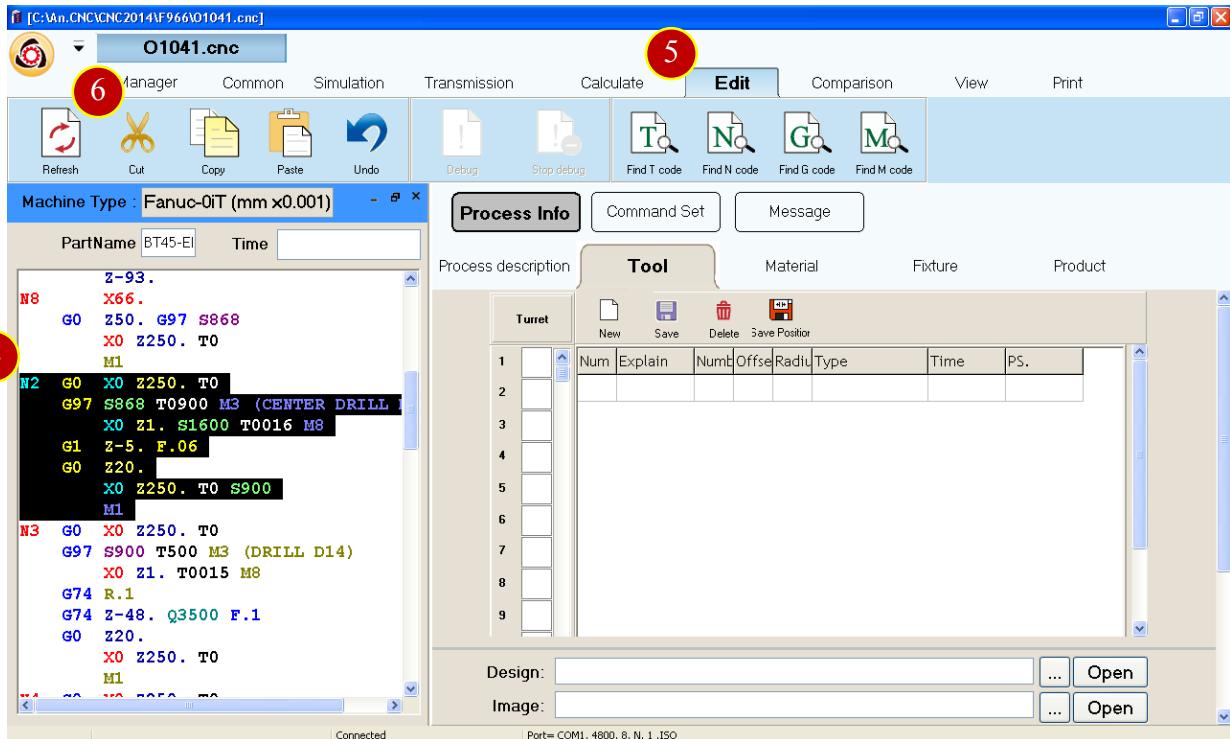
Program Edit

(4) Select the program to be moved

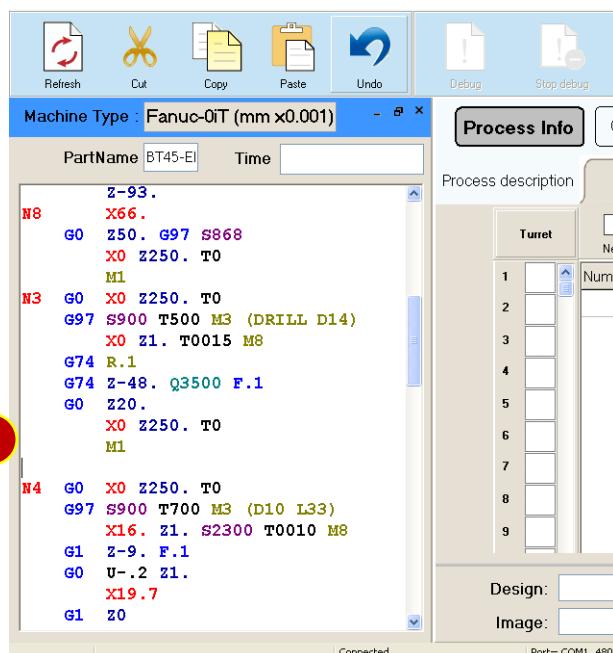
e.g. N2 G0 X0 Z250. T0...

(5) Click 【Edit】 tab

(6) Click 【Cut】



(7) Press [Enter] after process N3



NcEditor MANUAL

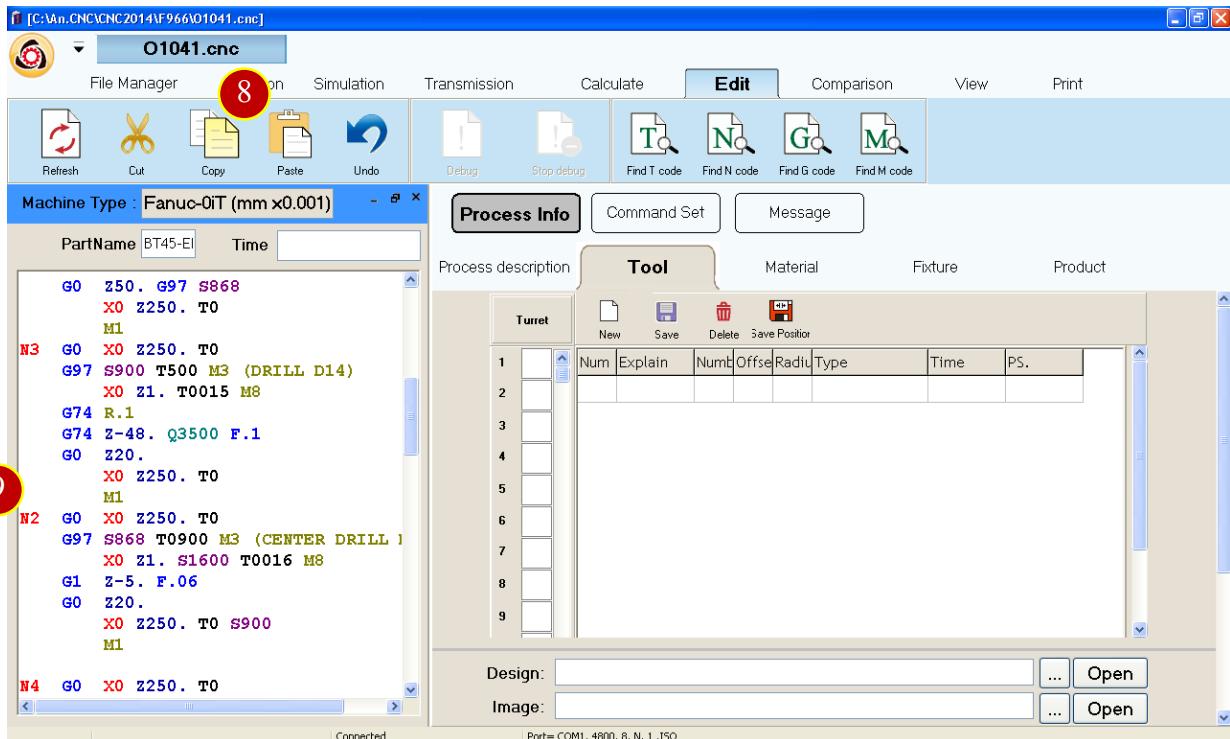


RenAn Information Technology Co., Ltd.
www.RenAn.com.tw

Program Edit

(8) Click 【Paste】

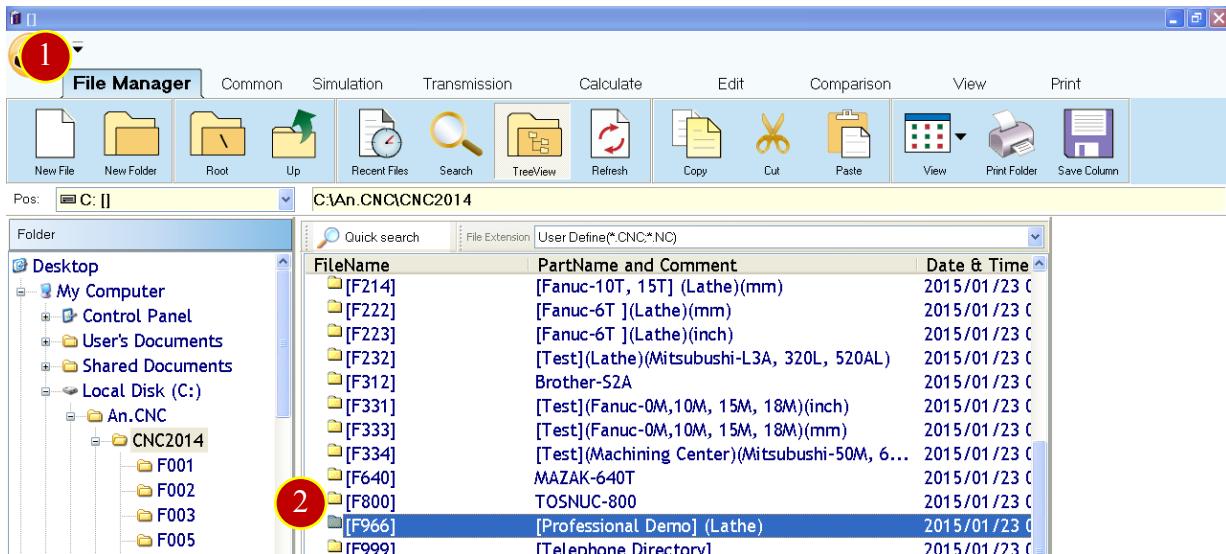
(9) Display the program



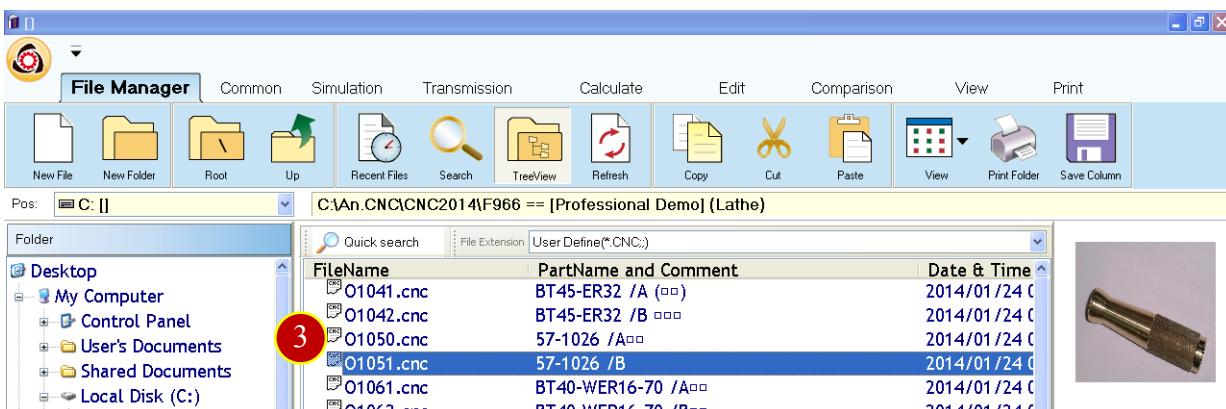
4.3.3 Undo

Undo the action and return to previous status
e.g. Undo the deleted program of F966\01051.CNC

- (1) Click 【File Manager】 tab
- (2) Double click folder 【F966】 to enter

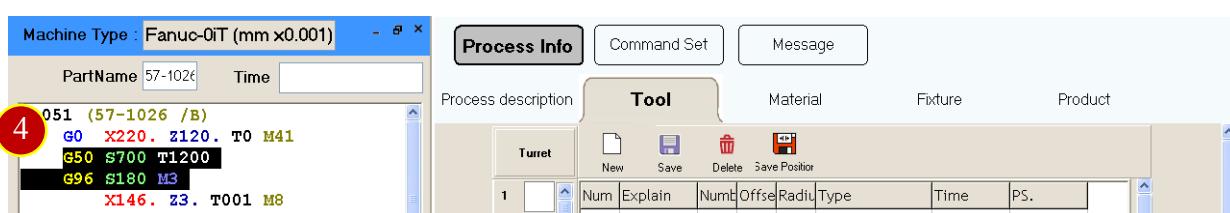


- (3) Double click 【01051.CNC】 to enter and edit



- (4) Select a part of program

e.g. G50 S700 T1200...



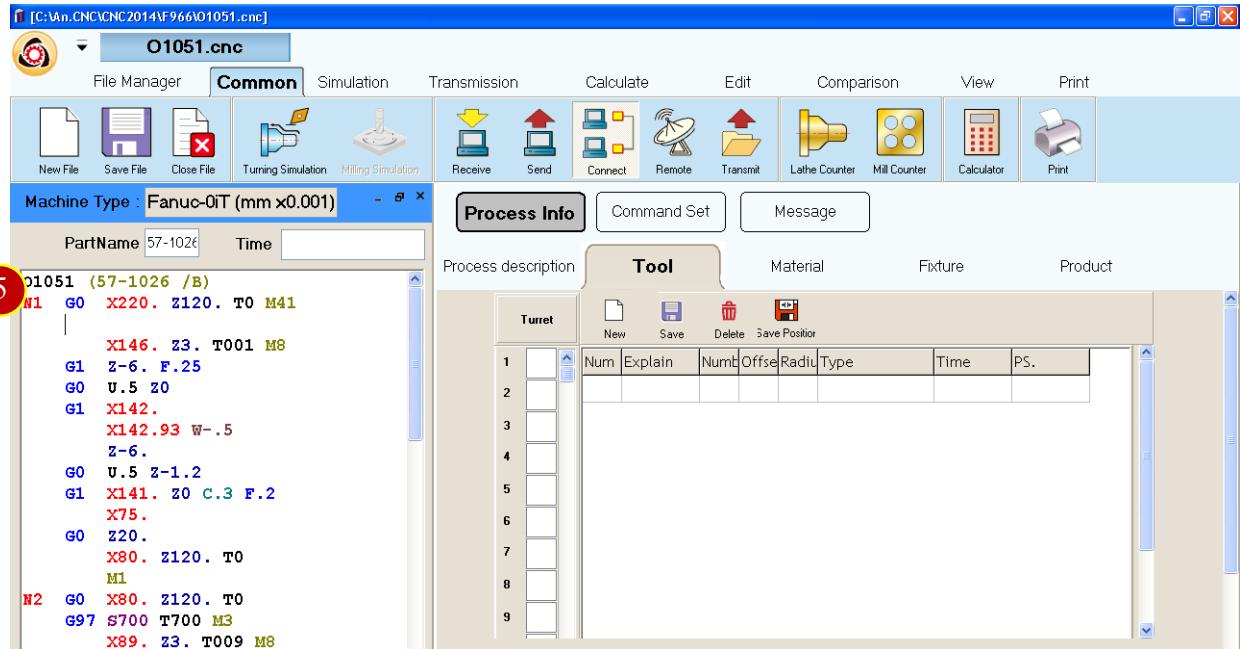
NcEditor MANUAL



RenAn Information Technology Co., Ltd.
www.RenAn.com.tw

Program Edit

(5) Press [Delete]

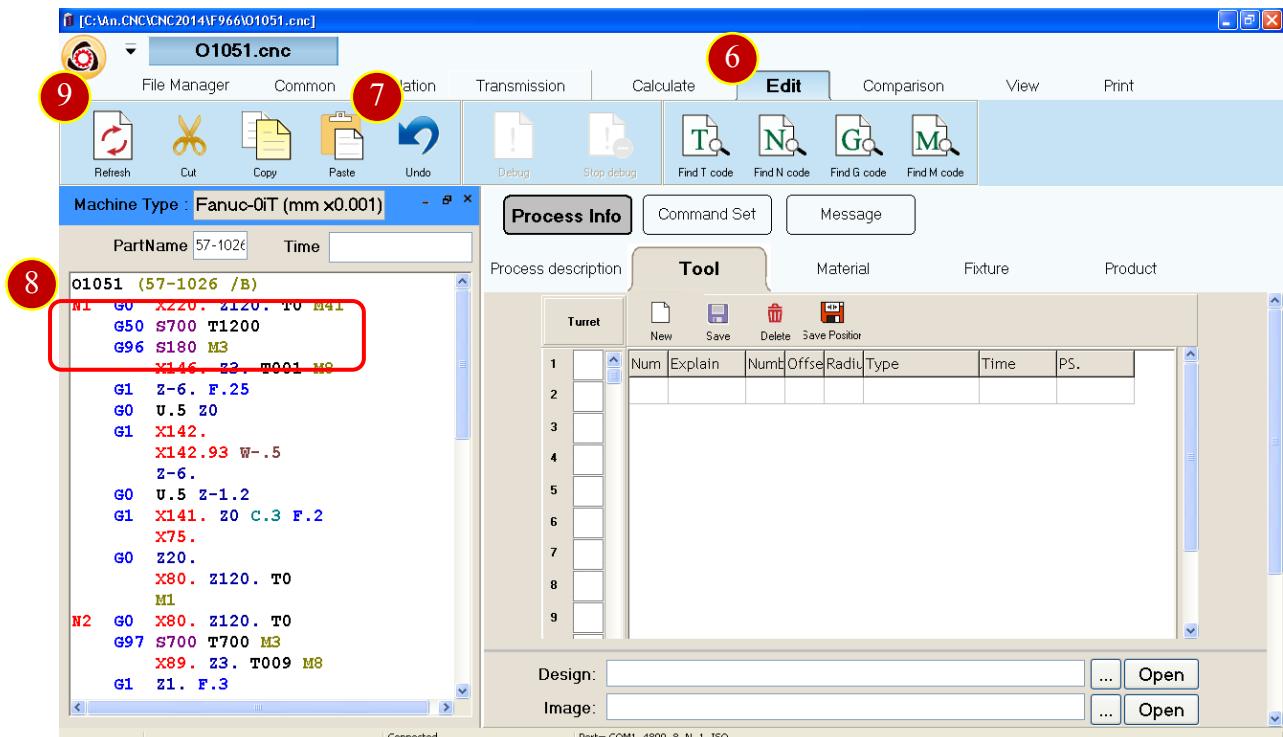


(6) Click [Edit] tab

(7) Click [Undo]

(8) The program be undone

(9) Click [Refresh]



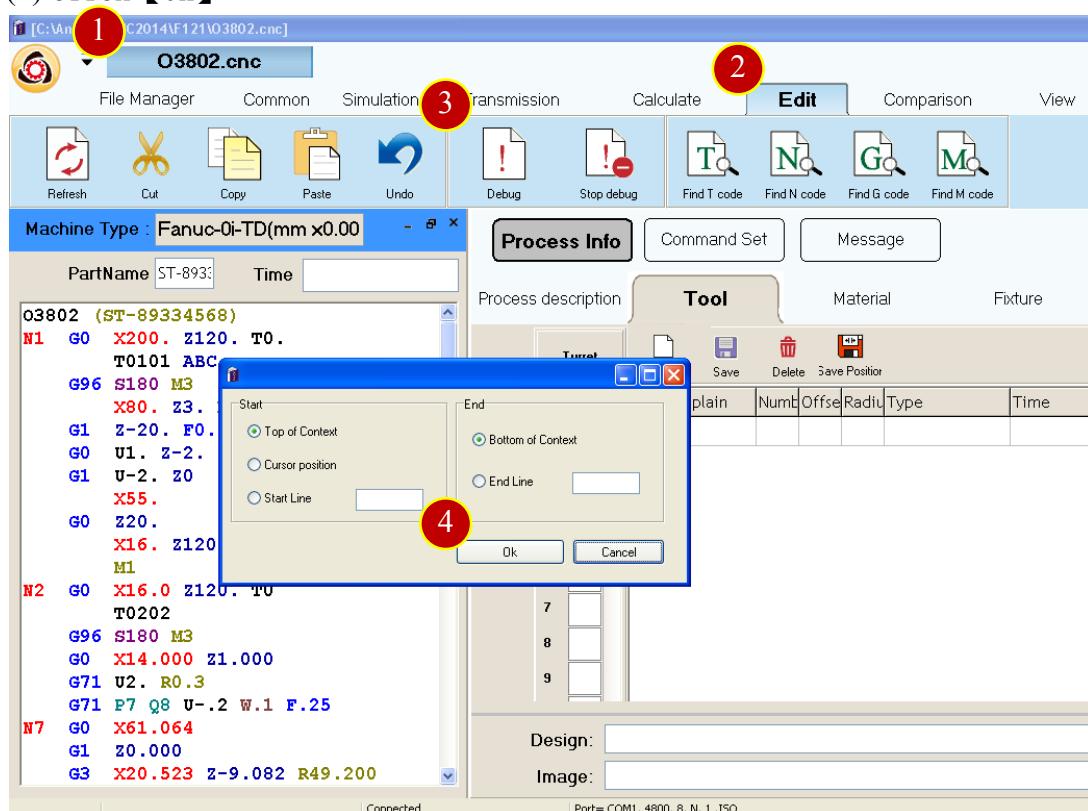
4.4 Syntax Debug

Based on the content of CNC program to debug in order to avoid illegality or collision

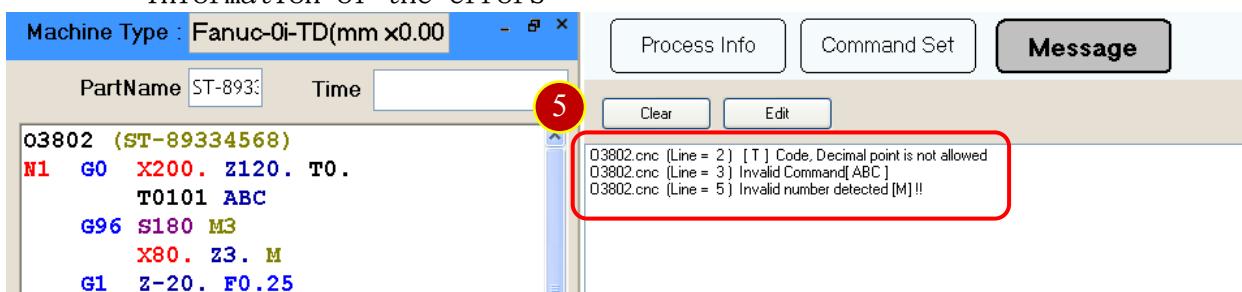
4.4.1 Debug

e.g. Debug file F121\00382.CNC

- (1) Open F121\00382.CNC
- (2) Click 【Edit】 tab
- (3) Click 【Debug】
- (4) Click 【OK】



- (5) When errors, message tab will automatically pop-up and display the information of the errors



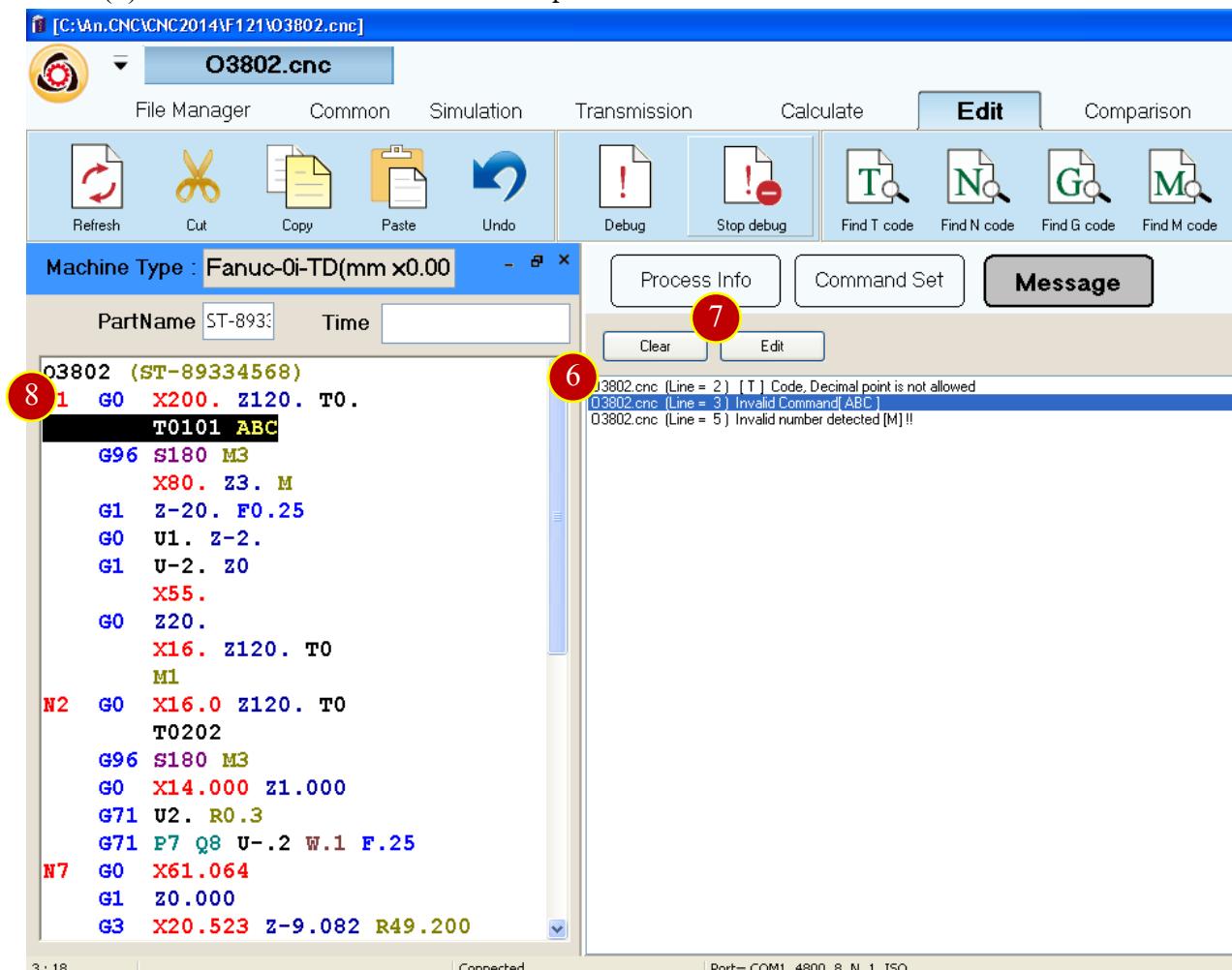
NcEditor MANUAL



RenAn Information Technology Co., Ltd.
www.RenAn.com.tw

Program Edit

- (6) Select one error information
e.g. Invalid Command (ABC)
- (7) Click 【Edit】
- (8) Cursor move to the error position



4.4.2 Stop Debug

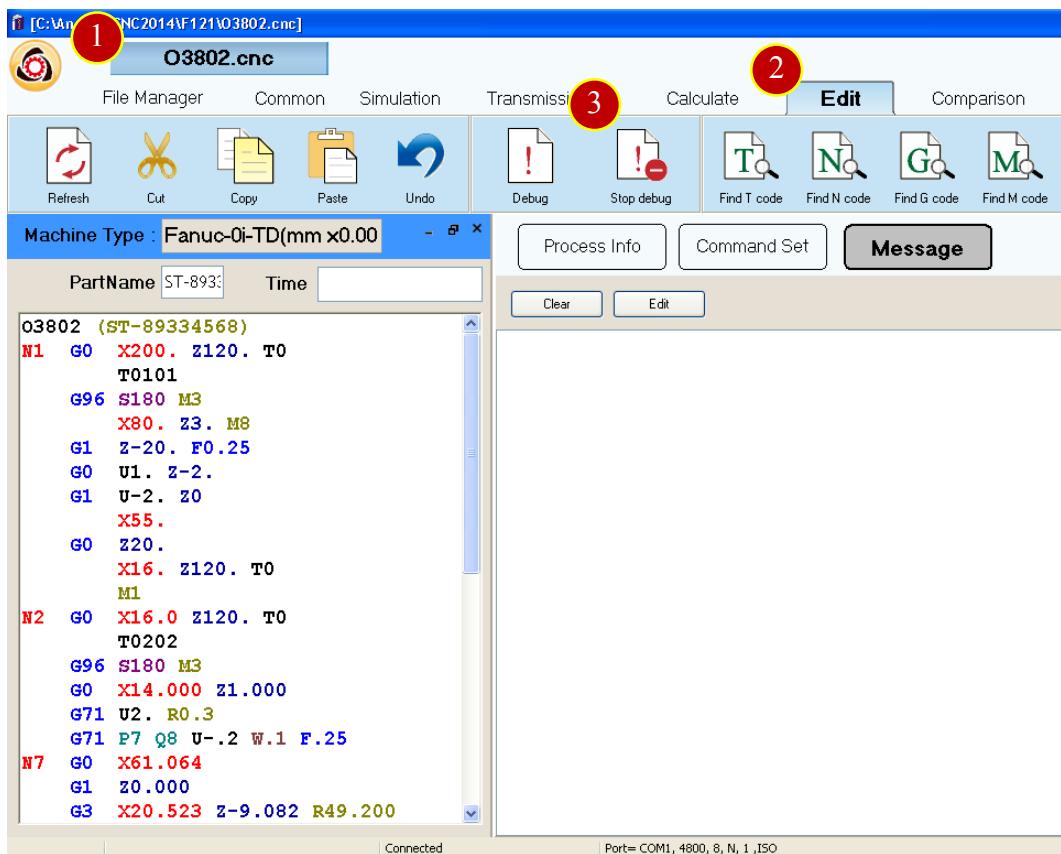
e.g. Stop Debug

(1) Open one file

e.g. F121\00382.CNC

(2) Click 【Edit】 tab

(3) Click 【Stop Debug】



4.5 Search

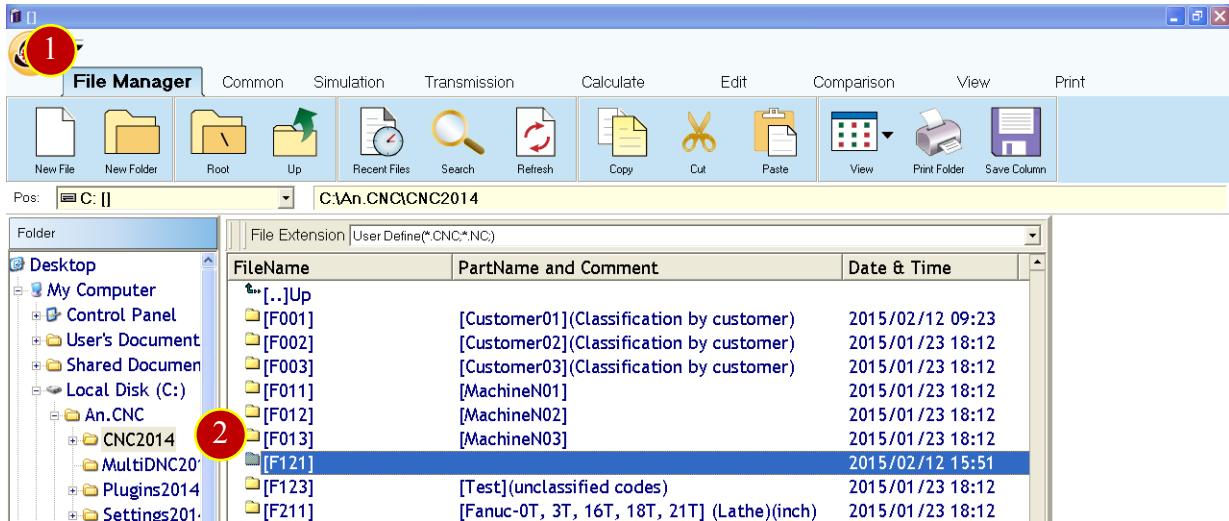
Search T, N, G and M code

4.5.1 Search Tool Number [T] code

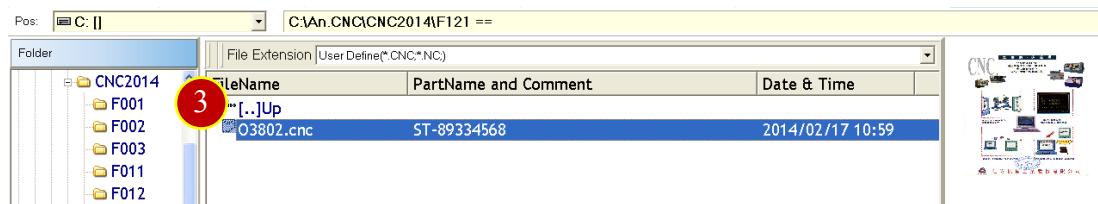
e.g. Search T03 in F121\00382.CNC

(1) Click 【File Manager】 tab

(2) Double click folder 【F121】 to enter



(3) Double click file 【03802.CNC】 to open

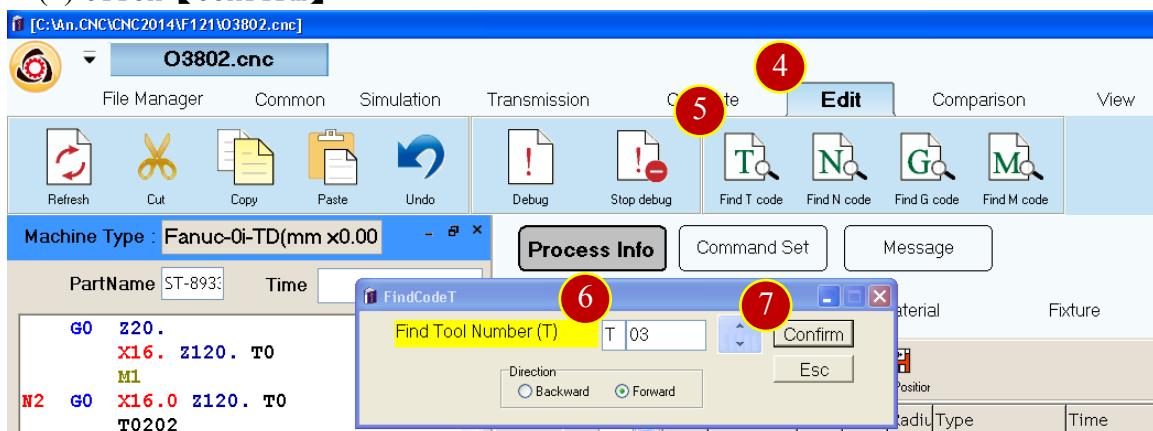


(4) Click 【Edit】 tab

(5) Click 【Find Code T】

(6) Key in [03]

(7) Click 【Confirm】



NcEditor MANUAL

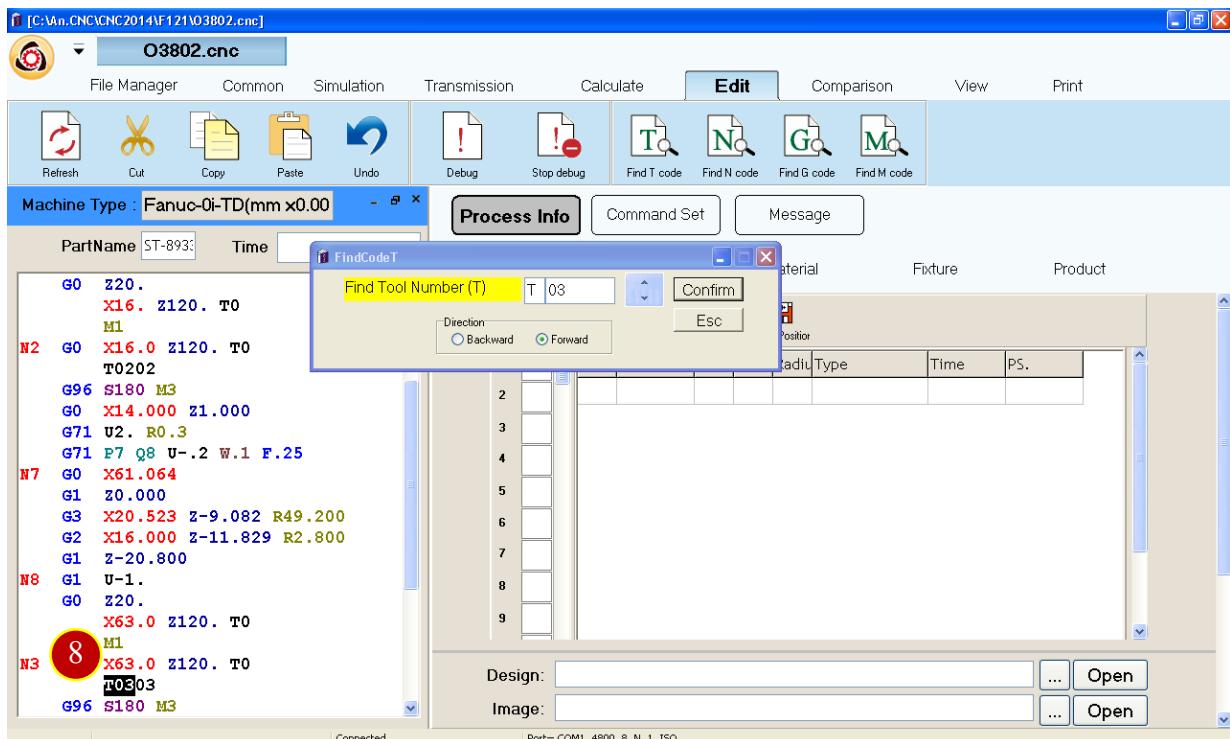


RenAn Information Technology Co., Ltd.
www.RenAn.com.tw

Program Edit

(8) Cursor move to destined position automatically

* Click 【Confirm】 or [F5] for next

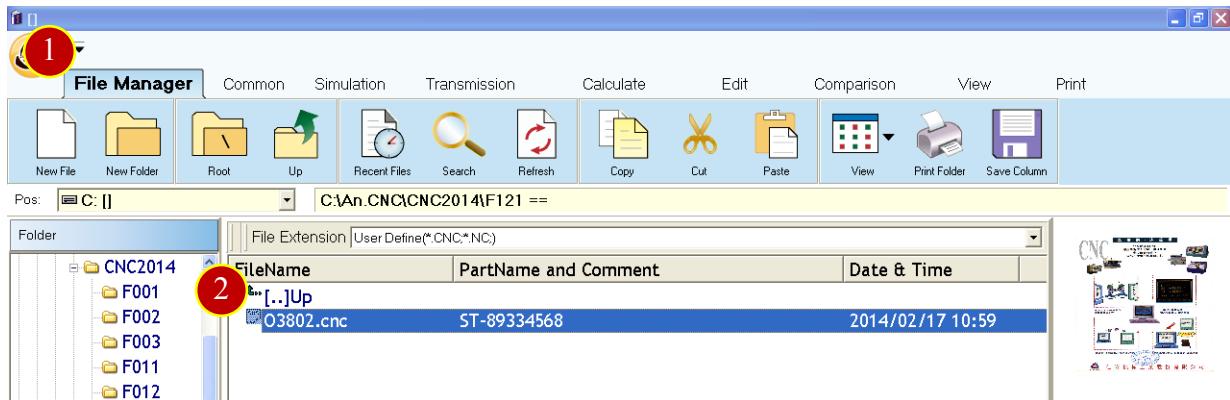


4.5.2 Search Sequence Number [N] code

e.g. Search N8 in F121\03802.CNC

(1) Click 【File Manager】 tab

(2) Double click folder 【F121】 > 【03802.CNC】 to open



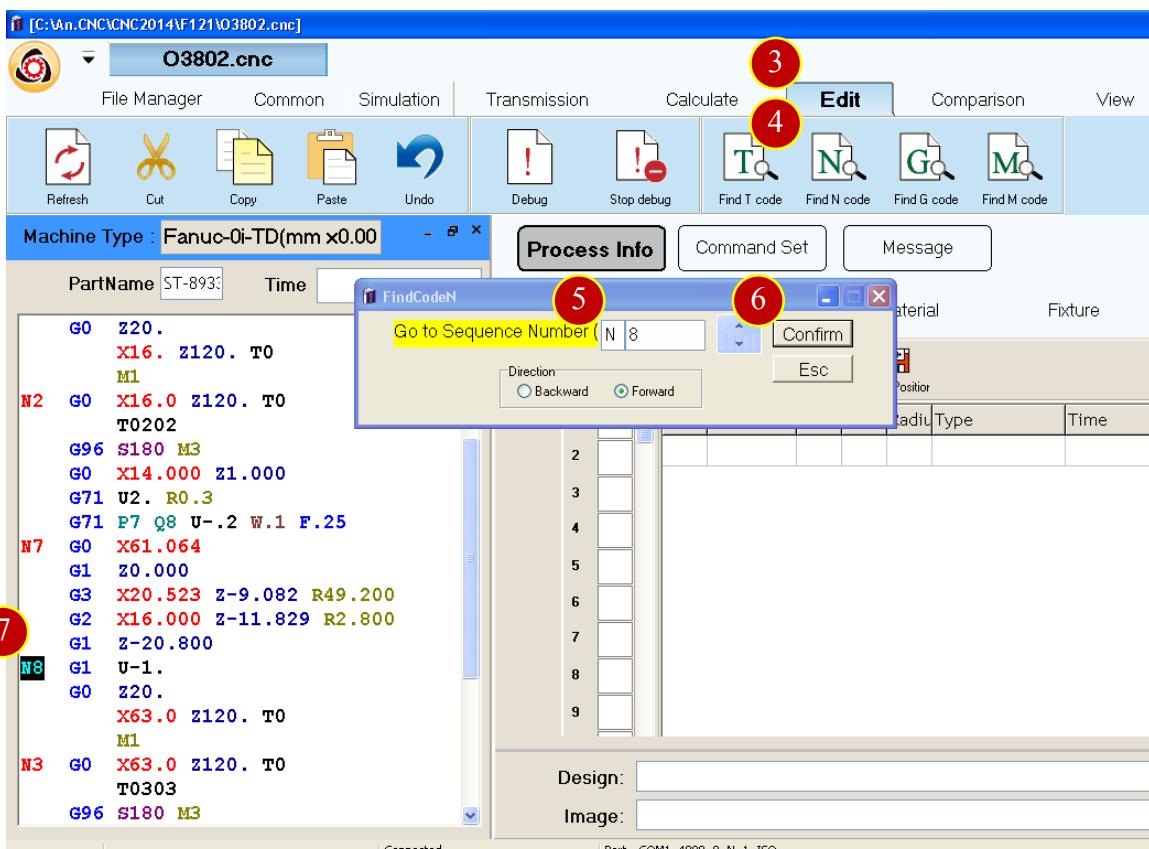
(3) Click 【Edit】 tab

(4) Click 【Find Code N】

(5) Key in [8]

(6) Click 【Confirm】

(7) Cursor move to destined position automatically

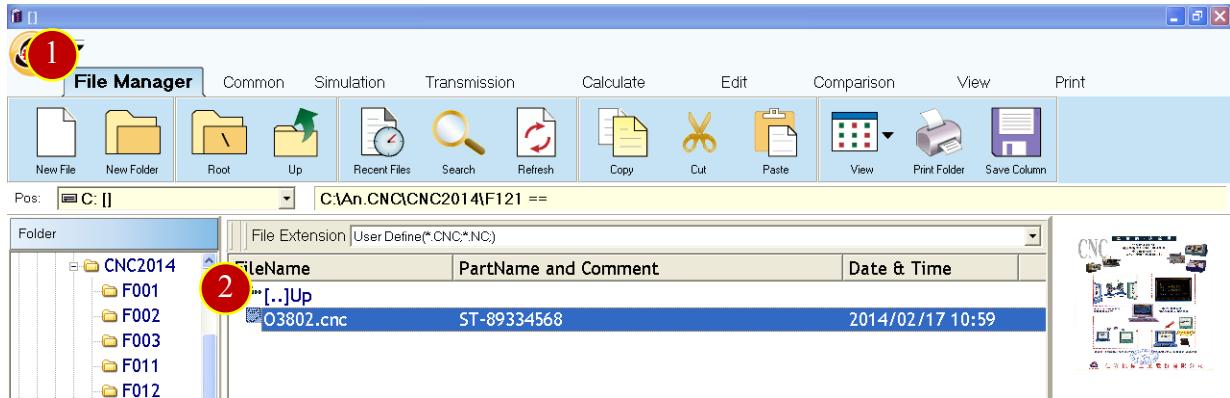


4.5.3 Search [G] code

e.g. Search G71 in F121\03802.CNC

(1) Click 【File Manager】 tab

(2) Double click 【03802.CNC】 of folder 【F121】 to open



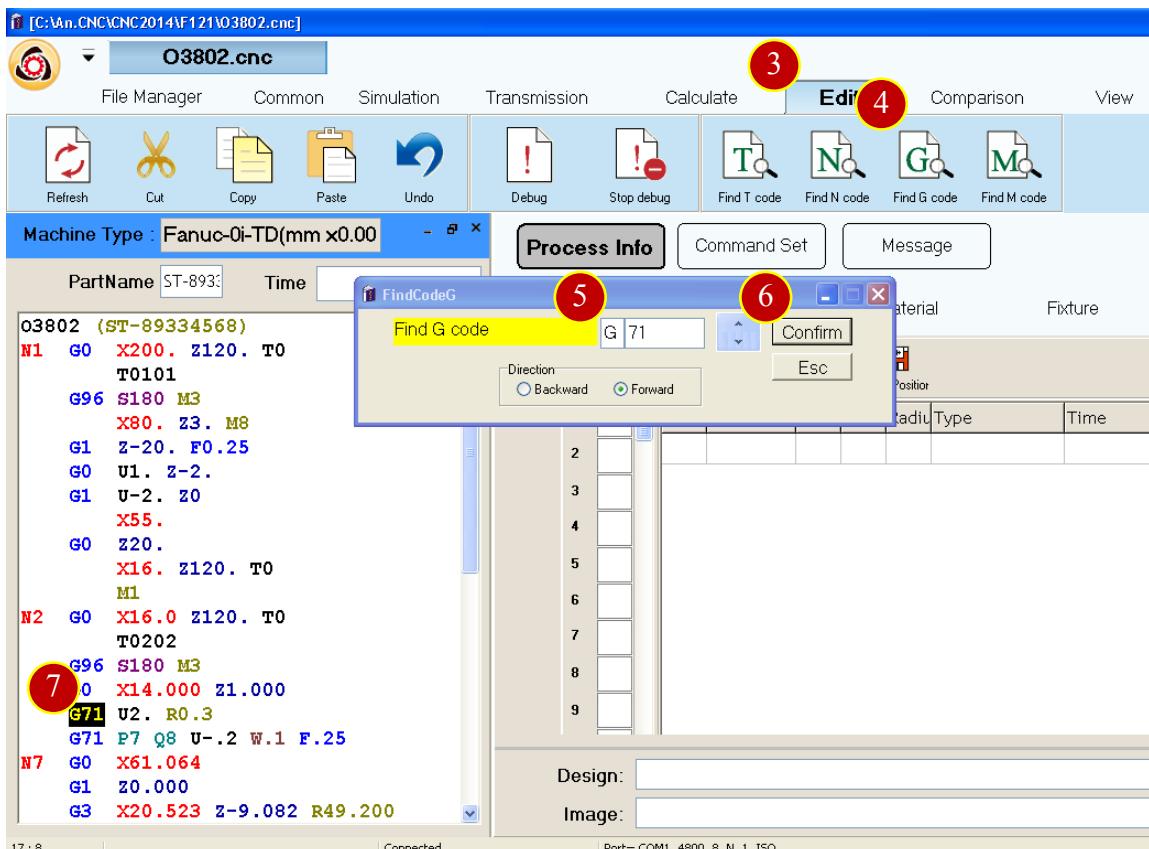
(3) Click 【Edit】 tab

(4) Click 【Find Code G】

(5) Key in [71]

(6) Click 【Confirm】

(7) Cursor move to destined position automatically

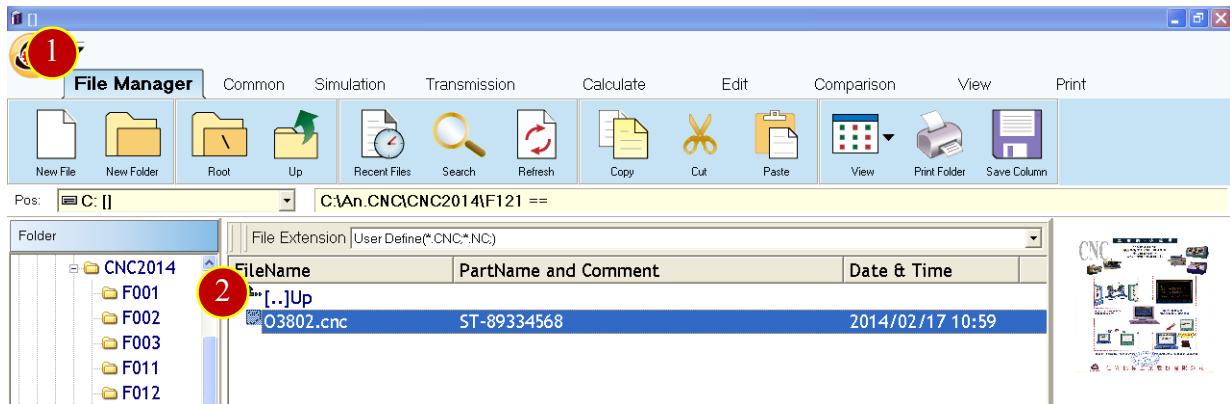


4.5.4 Search [M] code

e.g. Search M5 in F121\03802.CNC

(1) Click 【File Manager】 tab

(2) Double click 【03802.CNC】 of folder 【F121】 to open



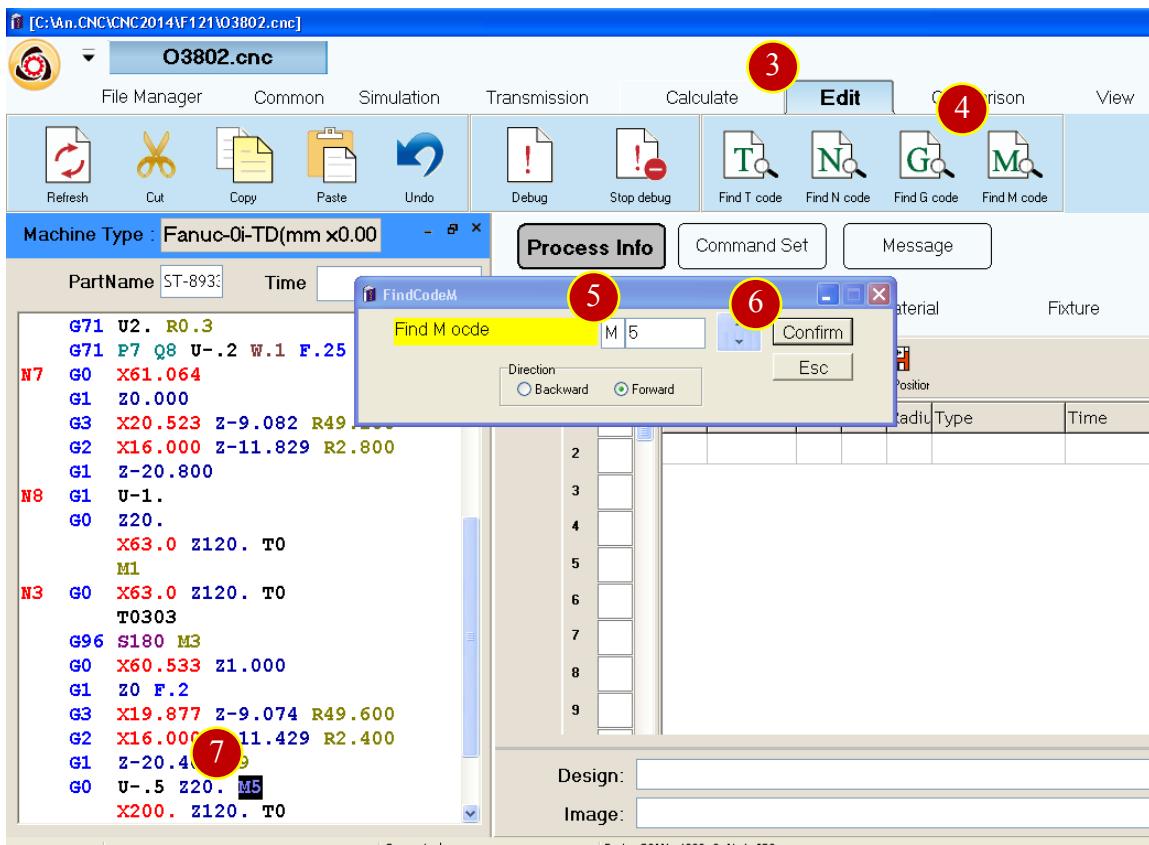
(3) Click 【Edit】 tab

(4) Click 【Find Code M】

(5) Input [5]

(6) Click 【Confirm】

(7) Cursor move to destined position automatically



4.6 Process Description, Fixture and Product

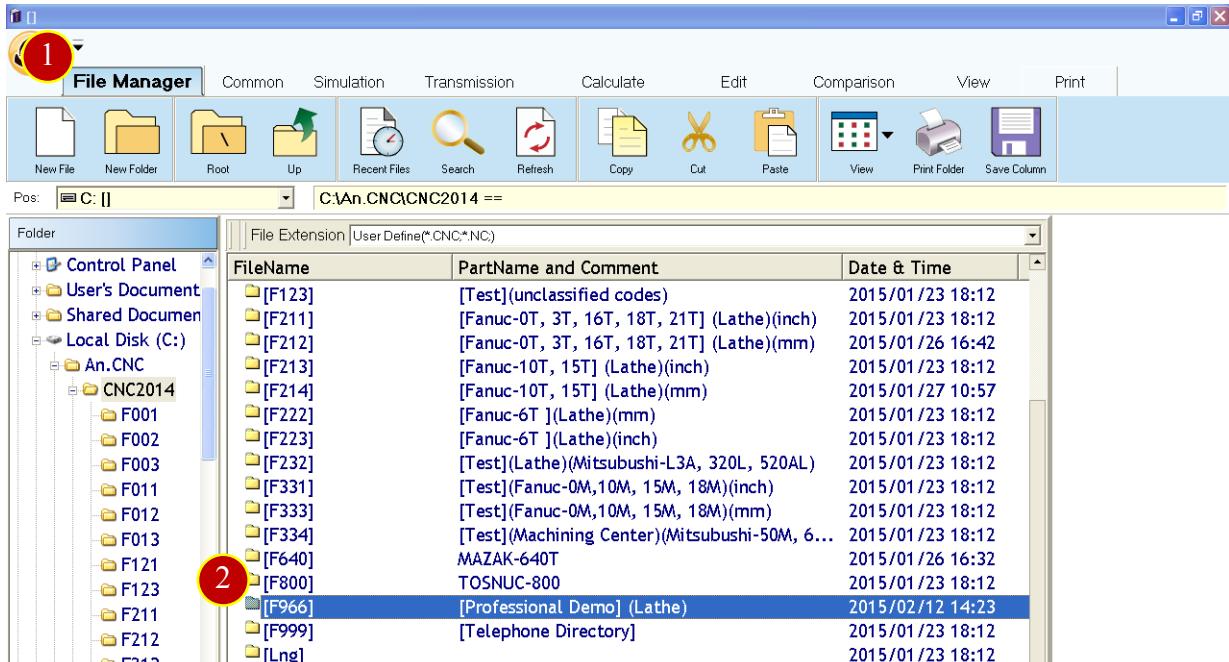
Write Process Description, Fixture and Product

4.6.1 Modify Process Description

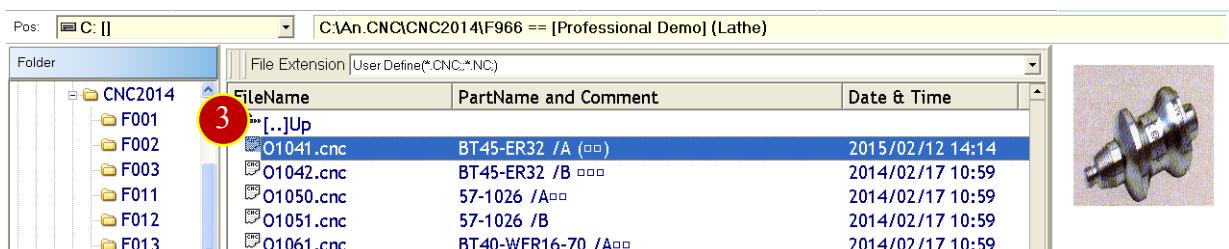
e.g. Modify F966\01041.CNC Process Description

(1) Click 【File Manager】 tab

(2) Double click 【F966】 to enter

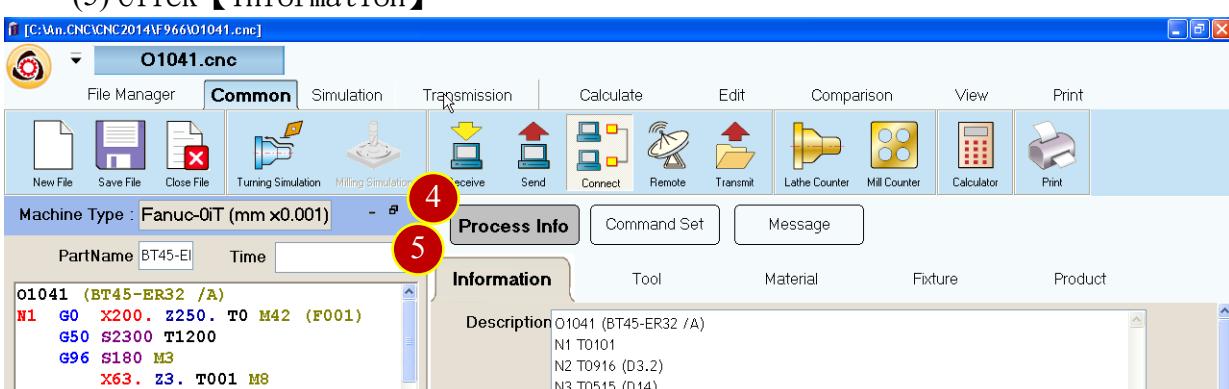


(3) Double click 【01041.CNC】 to open



(4) Click 【Process Info】

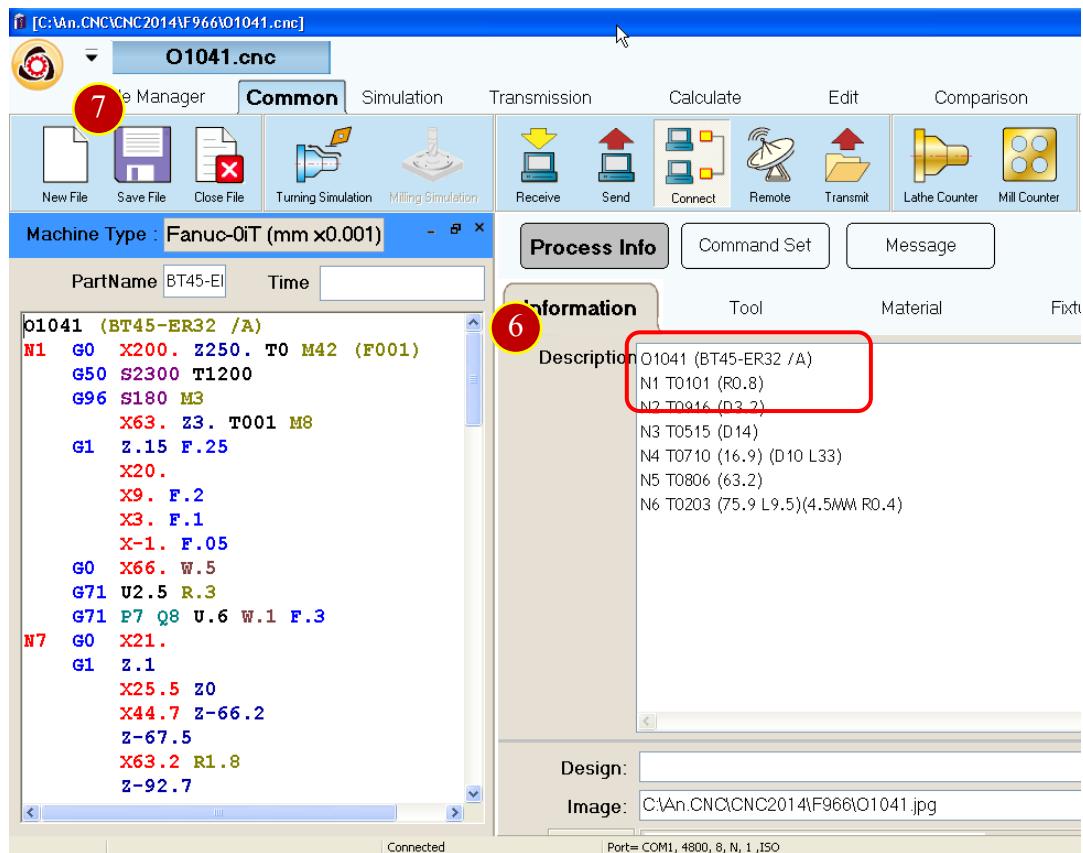
(5) Click 【Information】



(6) Modify 【Description】

e. g. N1 T0101 (R0.8)

(7) Click 【Save File】



4.7 Design

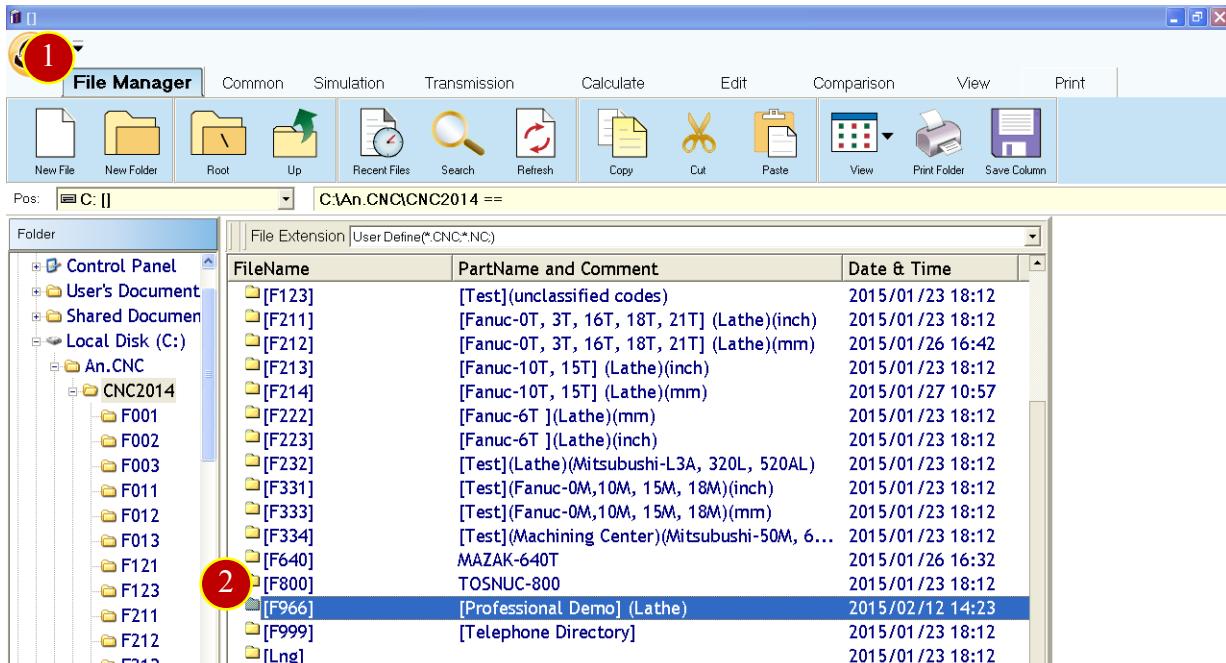
Add the picture in process descriptions, fixture and product to give a concrete image

4.7.1 Display design image in Process Descriptions

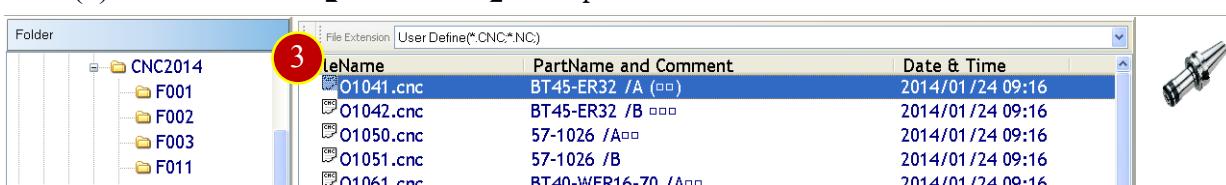
e.g. Display the design image of Process Description in F966\01041.CNC

(1) Click 【File Manager】 tab

(2) Double click 【F966】 to enter

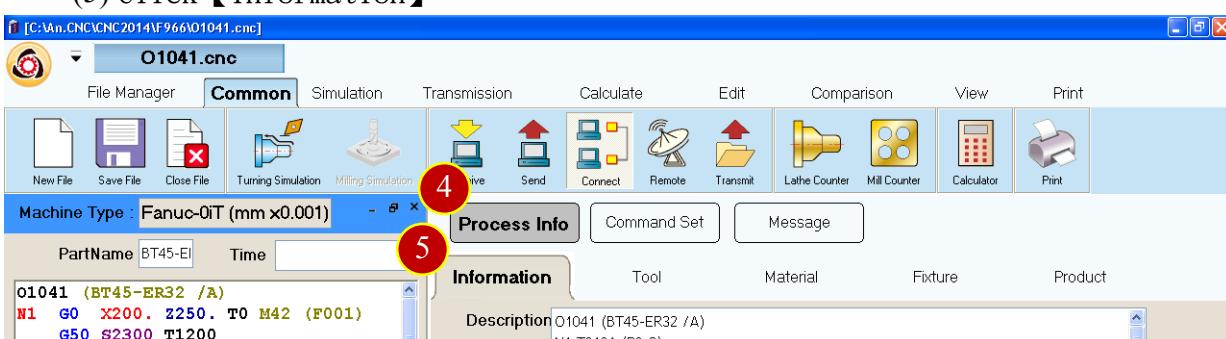


(3) Double click 【01041.CNC】 to open



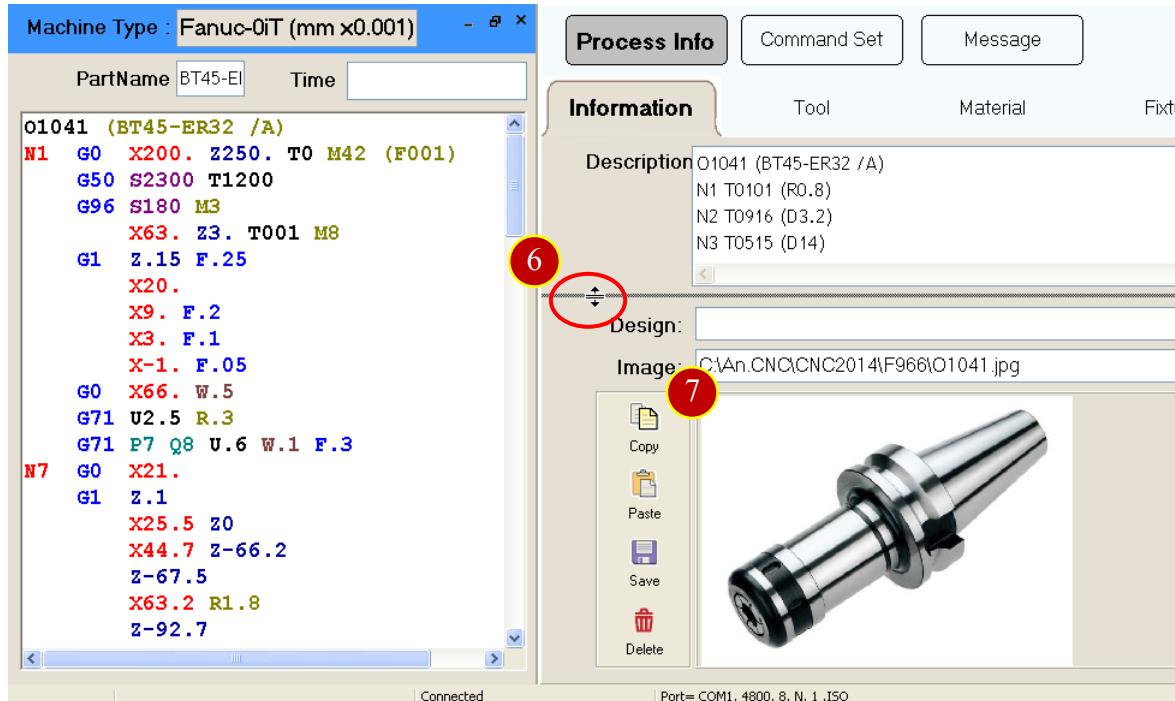
(4) Click 【Process Info】

(5) Click 【Information】



(6) Drag the line to adjust the position

(7) Image



4.7.2 Operating Design Image



4.8 Calculator

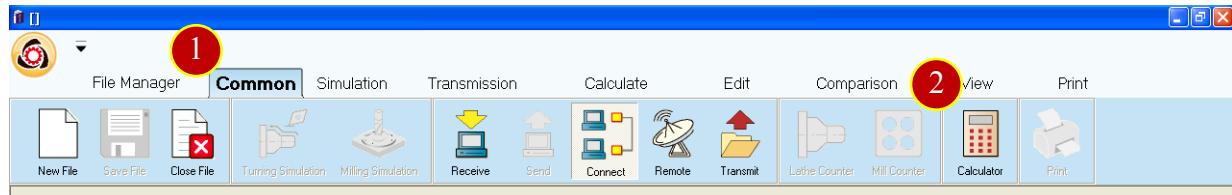
Open calculator when calculate and convert unit manually

4.8.1 Open Calculator

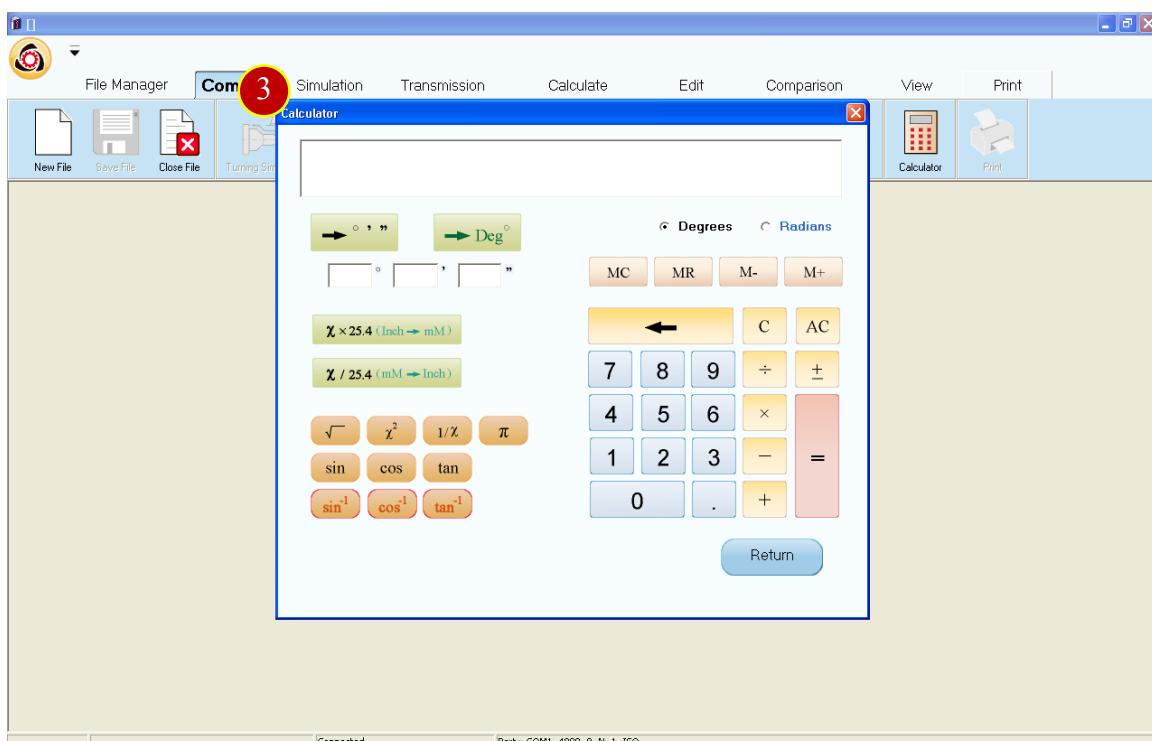
e.g. Execute calculator

(1) Click 【Common】 tab

(2) Click 【Calculator】



(3) Display calculator



4.8.2 Operating Calculator

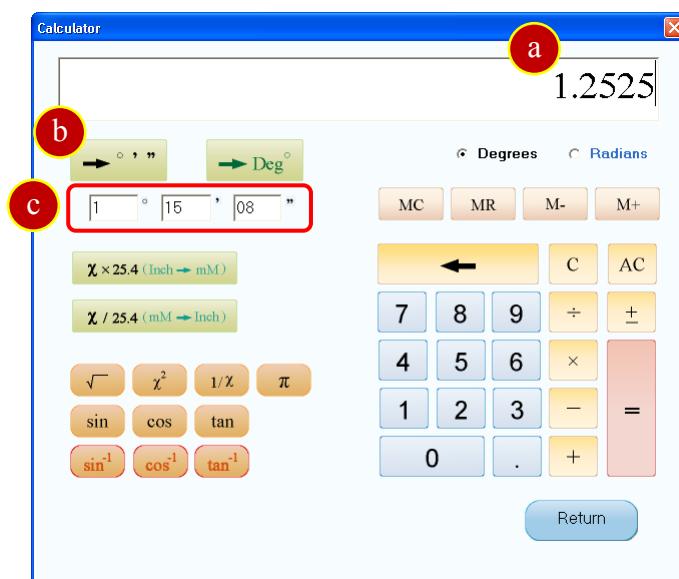
Functions displayed as



(1) Unit Conversion Area - Convert Decimal Degrees into DMS

e.g. Convert 1.2525 degrees into DMS

- Input [1.2525]
- Click [$\rightarrow^{\circ} \text{ } ' \text{ } ''$]
- Display DMS



(2) Unit Conversion Area - Convert DMS into Decimal Degrees

e.g. Convert $1^{\circ} 15' 09''$ into degrees

(a) Input $1^{\circ} 15' 09''$

(b) Click $\rightarrow \text{Deg}^{\circ}$

(c) Display Decimal Degrees



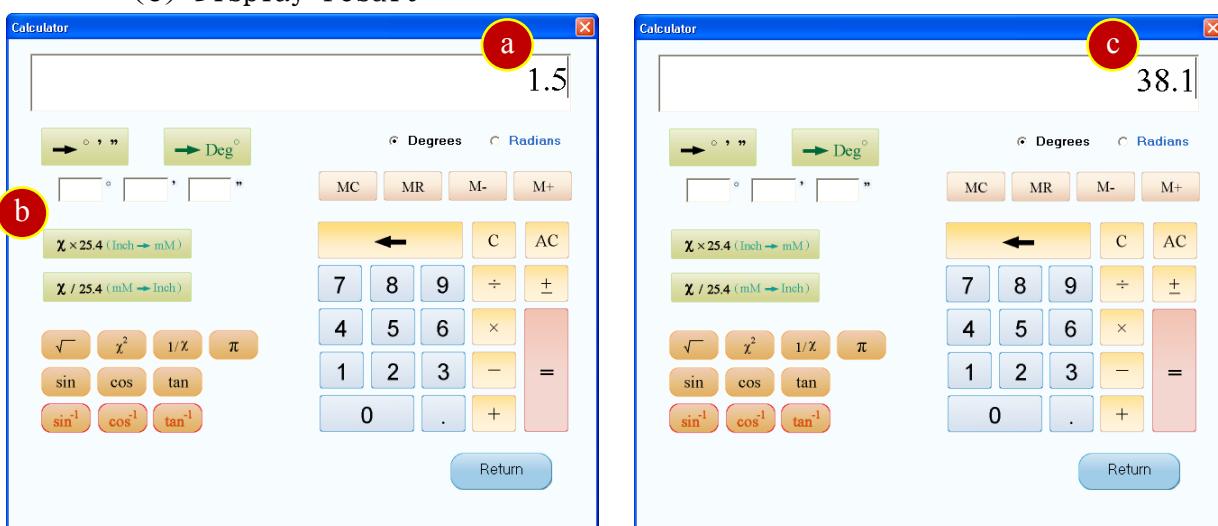
(3) Unit Conversion Area - Convert inch into mm unit

e.g. Convert 1.5 inches into mm

(a) Key in [1.5]

(b) Click $\times 25.4 (\text{Inch} \rightarrow \text{mm})$

(c) Display result



(4) Unit Conversion Area – Convert inch into mm unit

e.g. Convert 1.5 inches into mm

(a) Key in [1.5]

(b) Click **[X/25.4(mM→Inch)]**

(c) Display result



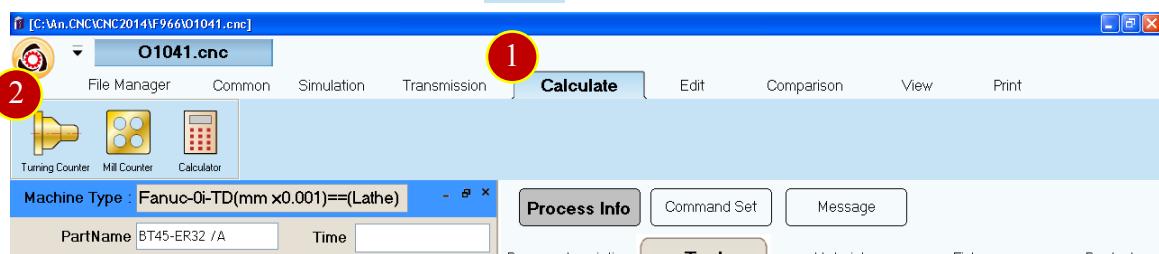
5. Part Contour Calculation

5.1 Counter Turn

We use the pattern guiding interface. According to the input of the drawing part dimension, system will calculate the related coordinates automatically. Then, choose the directions of cutting tools and cuttings and it generates CNC code. It is quite efficient to calculate arc intersection program of turning.

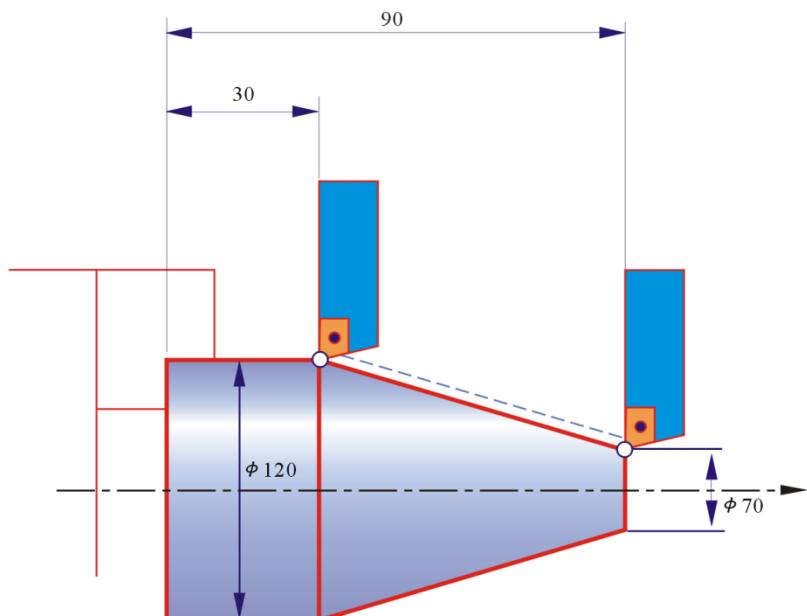
5.1.1 Open Turning Counter

- (1) Switch the tab to 【Calculate】
- (2) Click 【Turning Counter】  to enter counter window



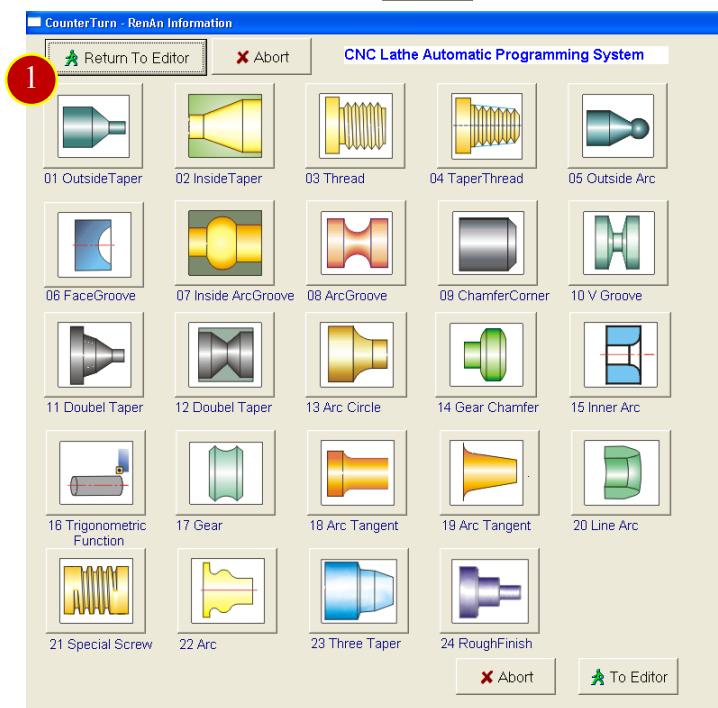
5.1.2 Select the processing type

Offers various processing in turning intersection calculation



(1) Pattern types

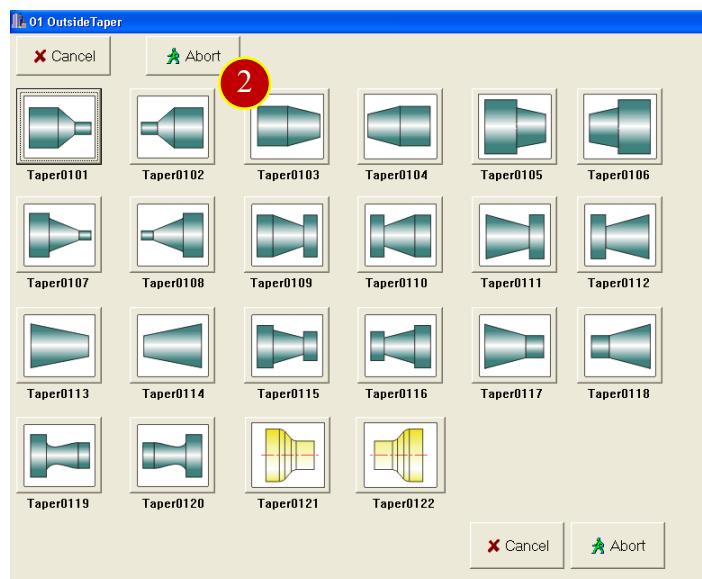
e.g. Click **[01 Outside Taper]** to enter **【Detailed patterns】**



(2) Select the processing pattern

e.g. Click **【Taper0103】** , enter **【Intersection calculation】**

P.S. According to the choosing pattern, it will show the pattern in details



5.1.3 Key in the processing term

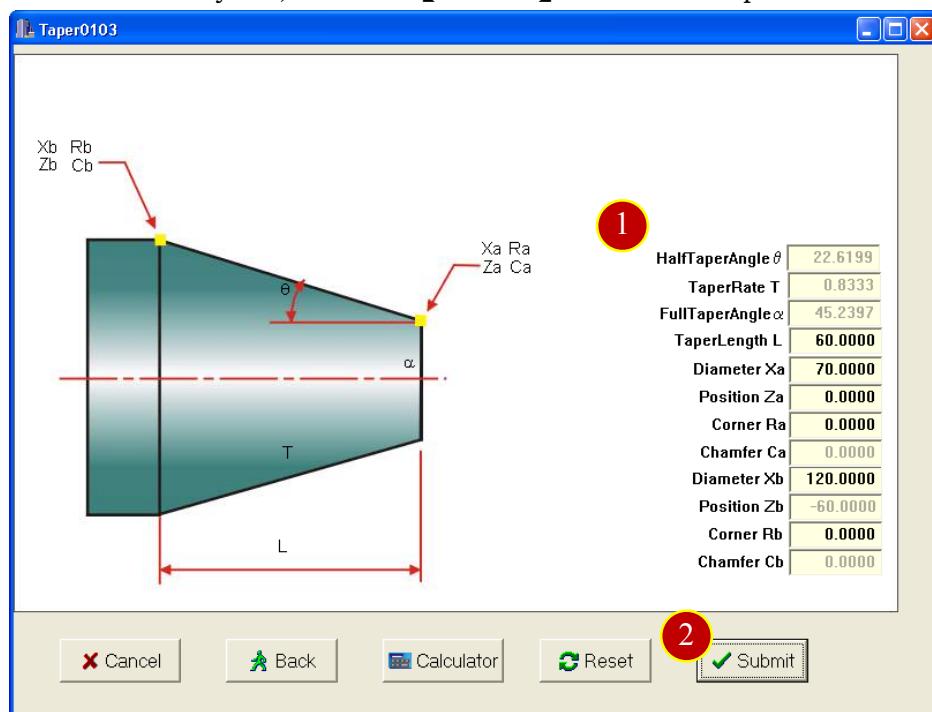
(1) Key in the drawing size term and system will calculate automatically according to the input terms.

You have to key in the important size first and then key in the related size in sequent.

e.g.

| Item | Input value | Instruction |
|-------------------|------------------------|-------------------------------------|
| Half Taper Angle= | [Enter] | Unknown size, press [Enter] to next |
| Taper Rate T= | [Enter] | Unknown, press [Enter] to next |
| Full Taper Angle= | [Enter] | Unknown, press [Enter] to next |
| Taper Length L= | <u>60.000</u> [Enter] | Key in $90-30 = 60$ |
| Diameter Xa= | <u>70.000</u> [Enter] | Diameter = 70 |
| Position Za= | <u>0.0000</u> [Enter] | Z of end face = 0 |
| Corner Ra= | <u>0.0000</u> [Enter] | Key in 0 |
| Chamfer Ca= | 0.0000 | Automatically show 0 |
| Diameter Xb= | <u>120.000</u> [Enter] | Diameter = 120 |
| Position Zb= | -60.0000 | Automatically show -60 |
| Corner Rb= | <u>0.0000</u> [Enter] | Key in 0 |
| Chamfer Cb= | 0.0000 | Automatically show 0 |

(2) After finish key in, click 【Submit】 to next step

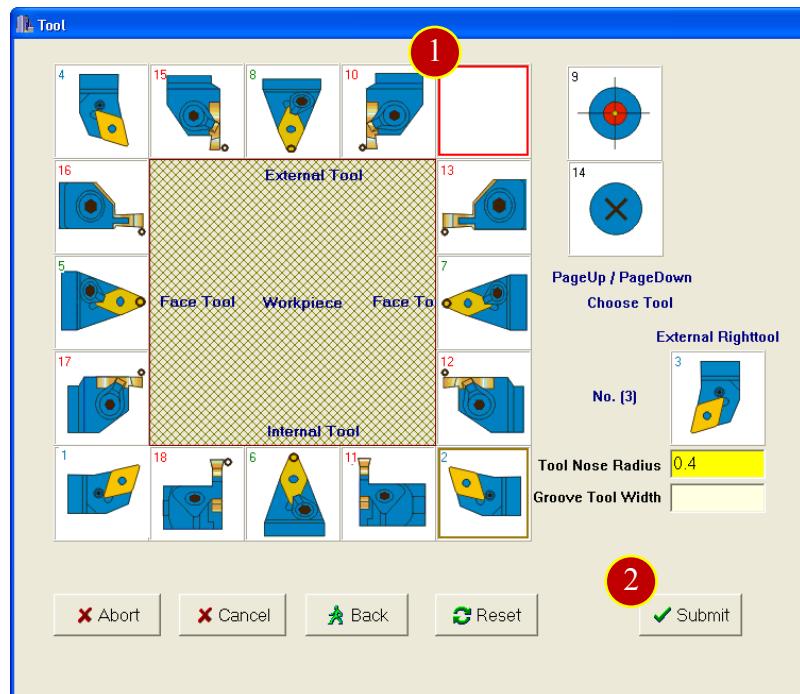


5.1.4 Choose the direction of the cutting and Tool

(1) Choose the appropriate Tool

e.g. External tool for right hands, and tool nose radius is 0.4

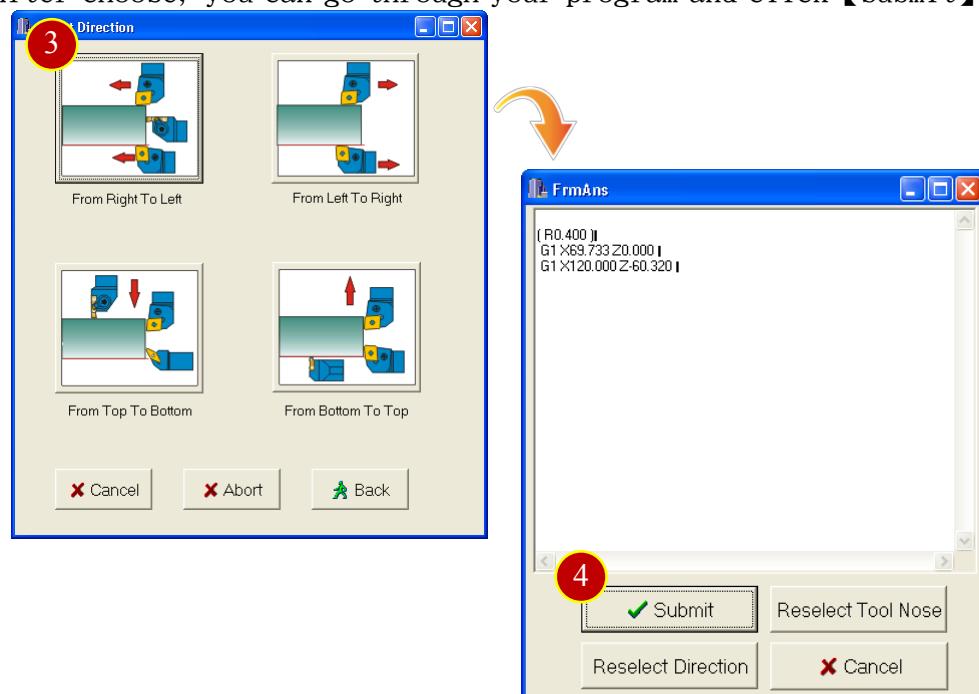
(2) Click 【Submit】



(3) Choose the cutting direction

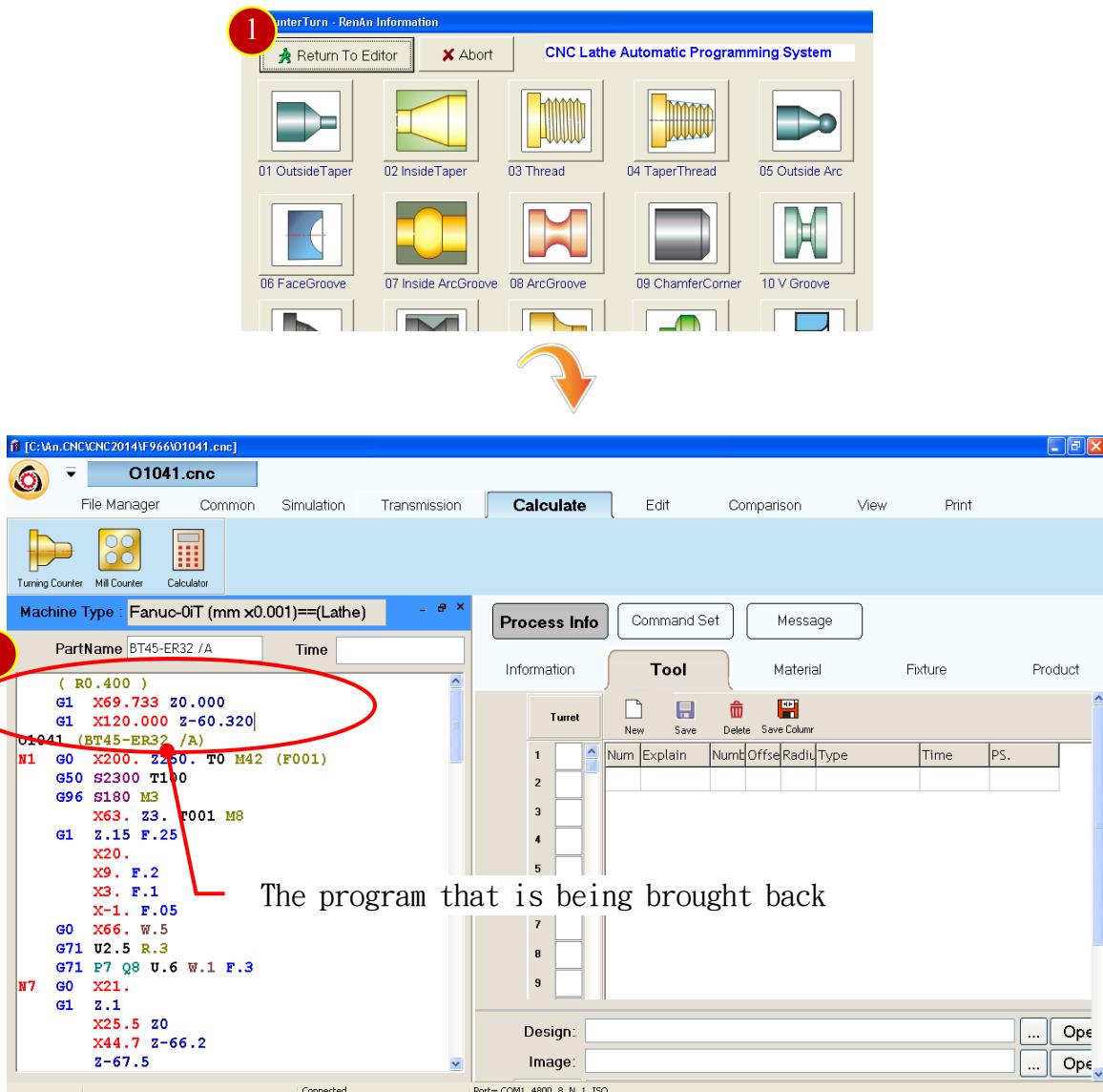
e.g. Click 【From Right To Left】

(4) After choose, you can go through your program and click 【Submit】 Complete.



5.1.5 Return to NcEditor system

- (1) Click 【Return To Editor】
- (2) System will copy your program to editor automatically



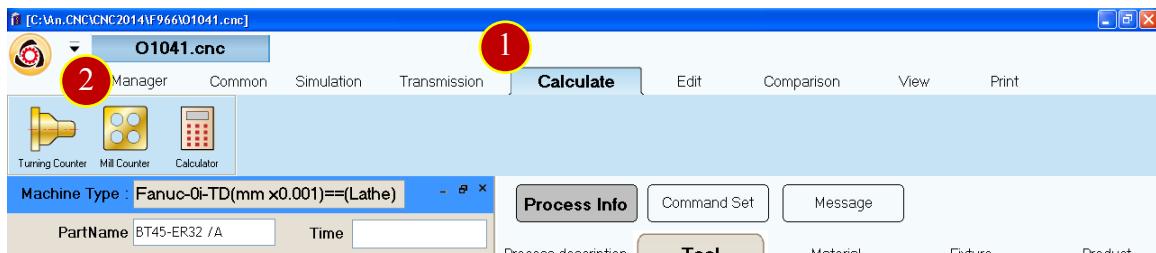
5.2 Counter Mill

We use the pattern guiding interface. According to the input of the drawing document, system will calculate the related coordinates automatically. Then, input the milling parameter and it generates CNC milling program.

5.2.1 Open Milling Counter

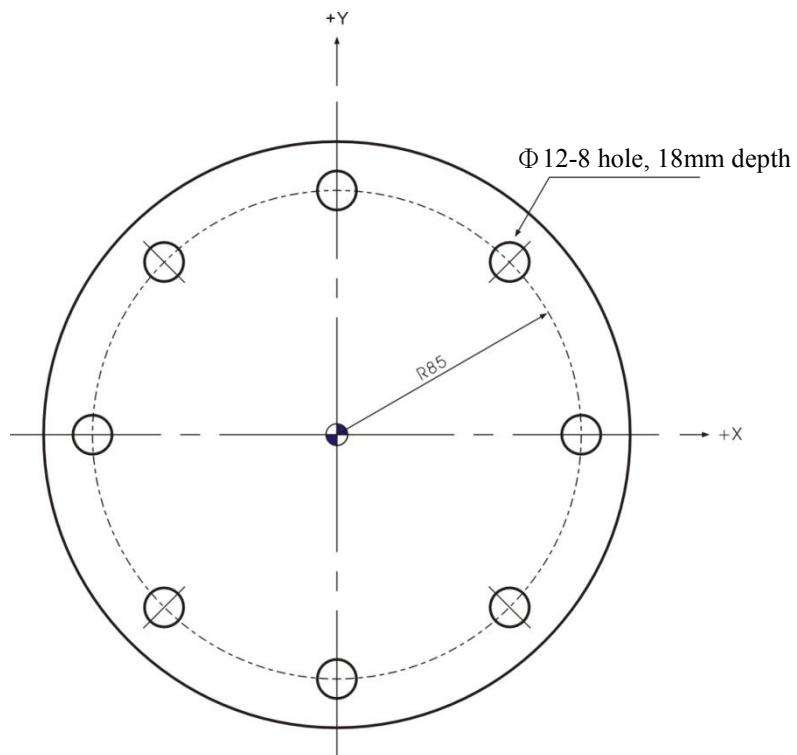
(1) Click 【Calculate】

(2) Click 【Mill counter】  to enter 【Milling intersection calculation】



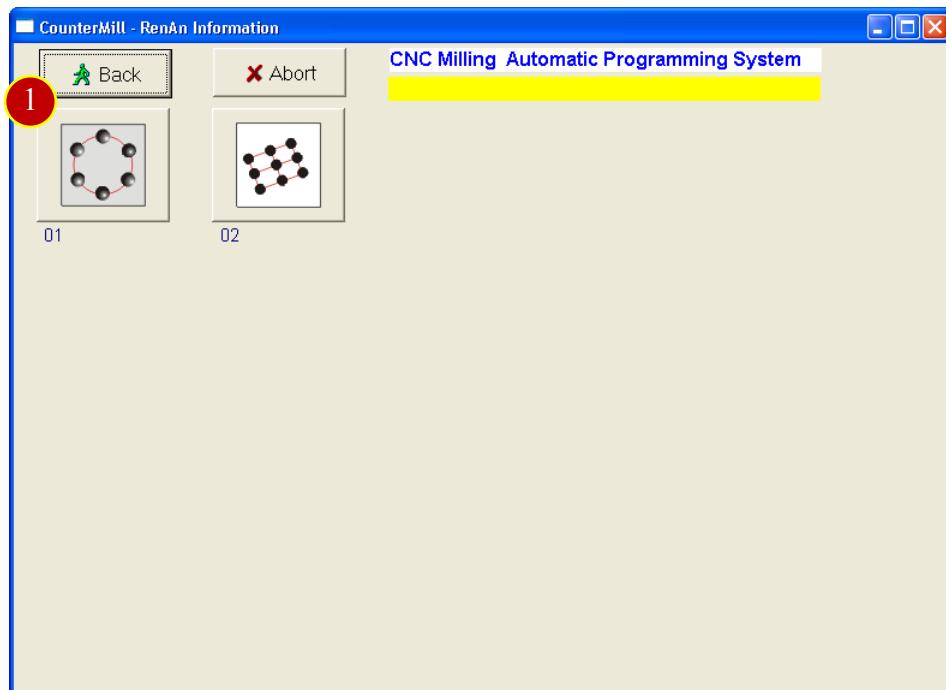
5.2.2 Select the processing type

System provide hole circle, hole line



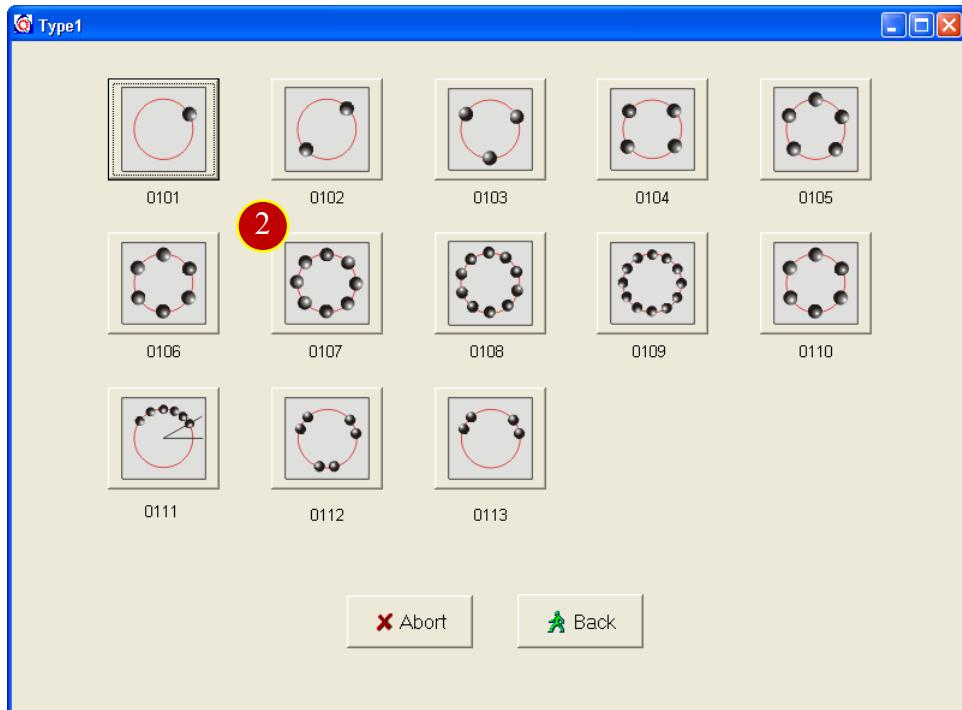
(1) Pattern type

e.g. Click 【01 Hole circle】 to enter 【Detailed patterns】



(2) Select the processing pattern

e.g. Click 【0107】 to enter 【Intersection calculation】



5.2.3 Key in the processing term

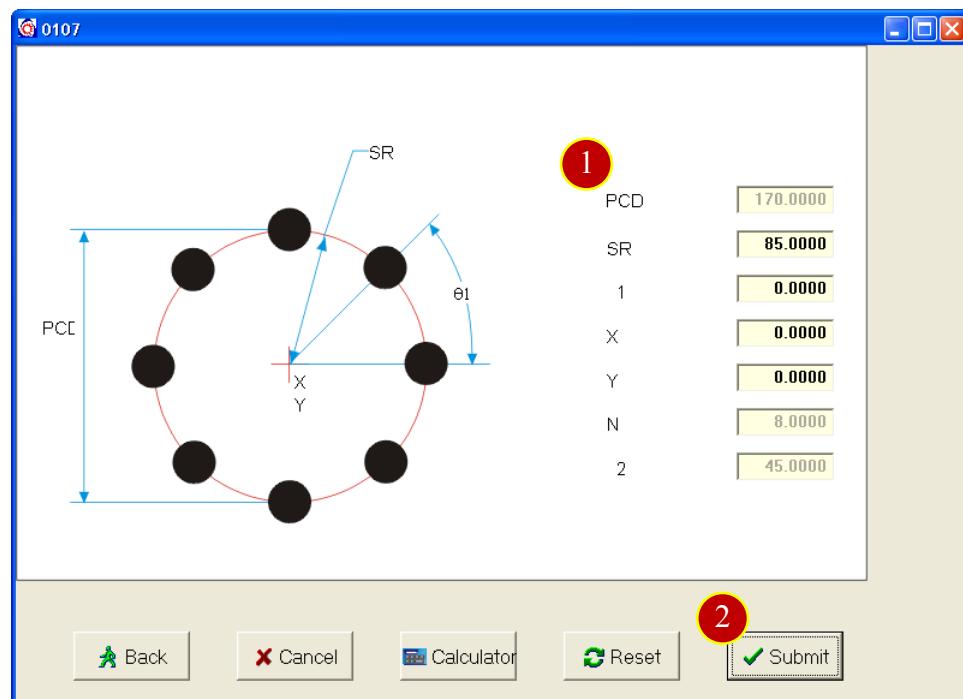
(1) Key in the drawing size term and system will calculate automatically according to the input terms.

You have to key in the important size first and then key in the related size in sequent.

e.g.

| Item | Input Value | Instruction |
|------------------|-------------|-------------------------------------|
| PCD= | [Enter] | Unknown size, click [Enter] to next |
| SR= | 85. [Enter] | Circle radius R = 85 |
| Begin angle 1= | 0 [Enter] | Begin hole of angle = 0 |
| Coordinate X= | 0 [Enter] | Coordinate X of center = 0 |
| Coordinate Y= | 0 [Enter] | Coordinate Y of center = 0 |
| Quantity N= | 8 [Enter] | Automatically show 8 |
| Average angle 2= | 45 [Enter] | Automatically show 45 |

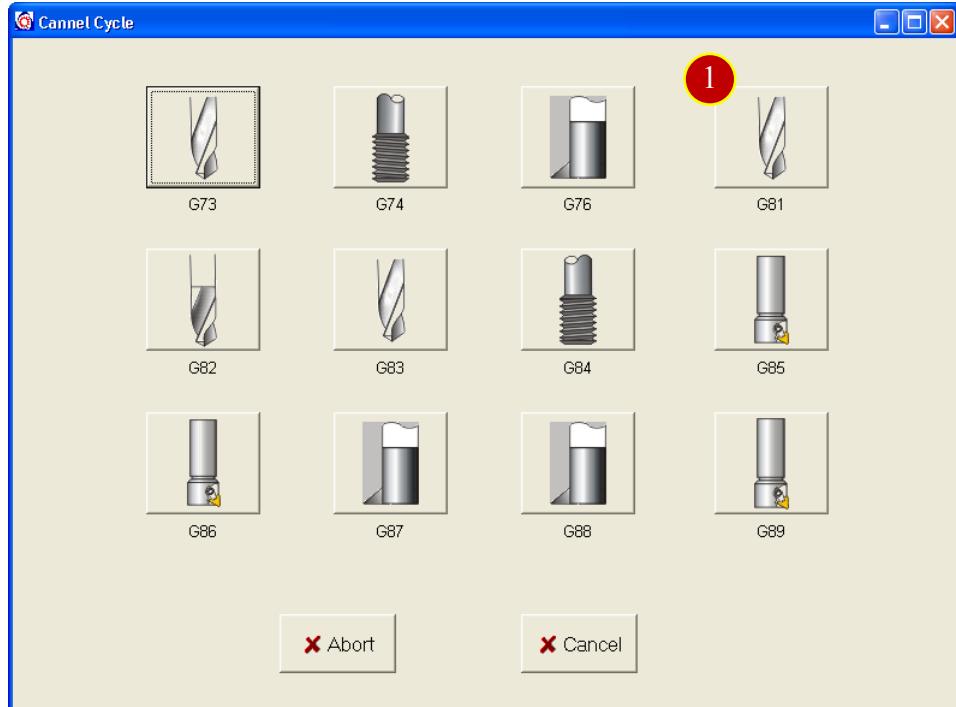
(2) Click 【Submit】 to next



5.2.4 Select

(1) Select G81 type

e.g. Click **[G81]** to next



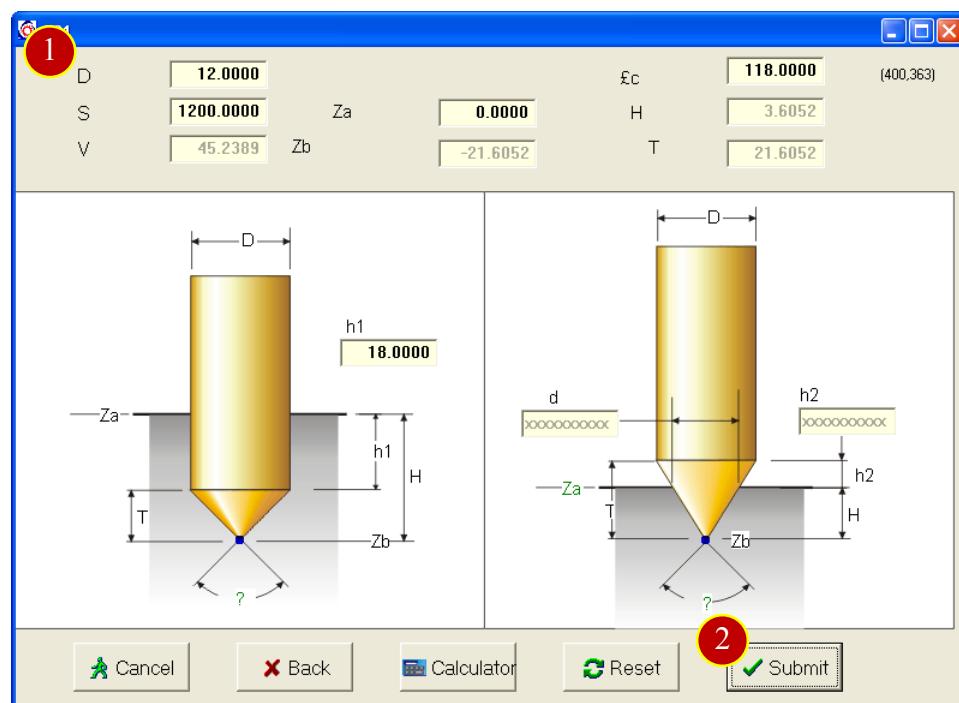
5.2.5 Key in the milling parameter

(1) Key in the tool parameter

e.g.

| Item | Input value | Instruction |
|---------------------|---------------------|---------------------------------|
| Diameter D= | <u>12</u> [Enter] | Unknown, click [Enter] to next |
| Spindle speed S= | <u>1200</u> [Enter] | Spindle speed, key in 1200 |
| V mm/min= | <u>45.2389</u> | Automatically show 45.2389 |
| Za= | <u>0</u> [Enter] | Surface, key in 0 |
| Zb= | | Unknown, click [Enter] to next |
| Drill nose angle Θ= | <u>118</u> [Enter] | Angle of drill nose, key in 118 |
| Depth H= | <u>3.6052</u> | Automatically show 3.6052 |
| T= | | Unknown, click [Enter] to next |
| Depth in effect h1= | <u>18</u> [Enter] | Depth in effect, key in 18 |

(2) Click 【Submit】 to next



(3) Key in the cutting parameter

e.g. Begin position Zs=50

R position ZR=5

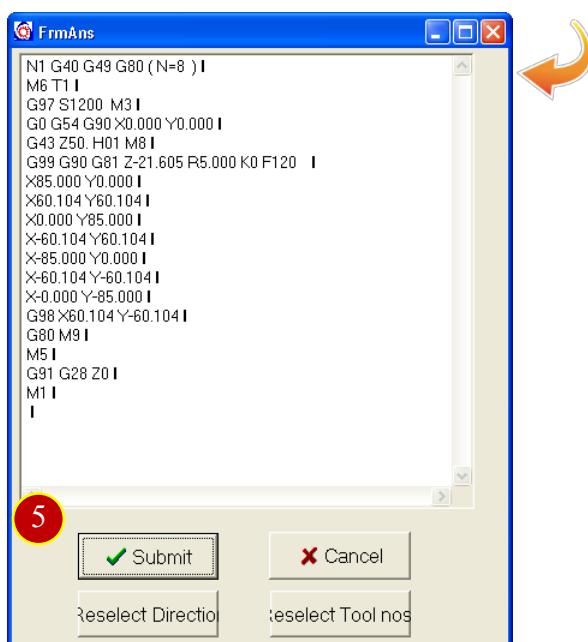
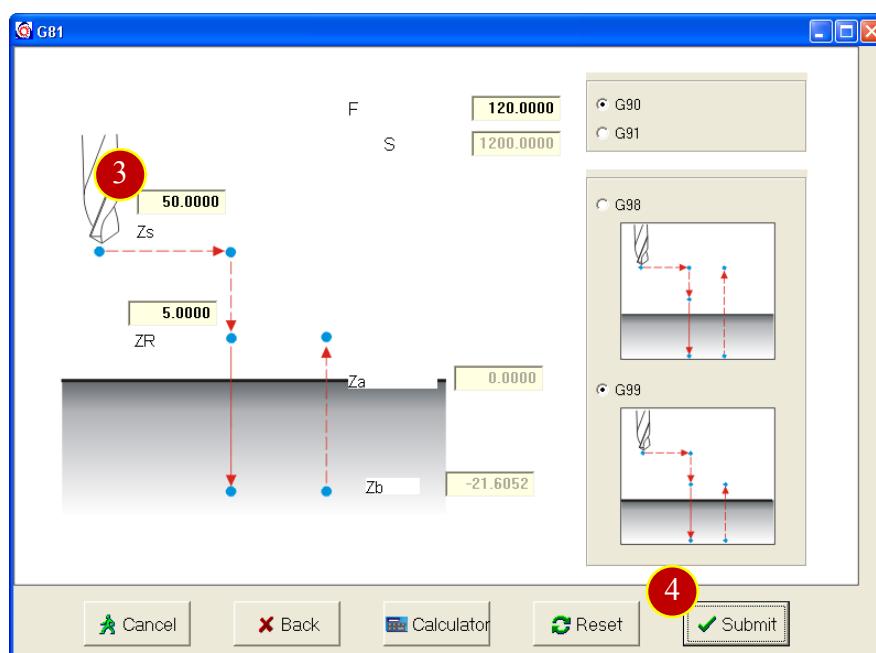
Feed rate F=120

Select 【G90】

Select 【G99】

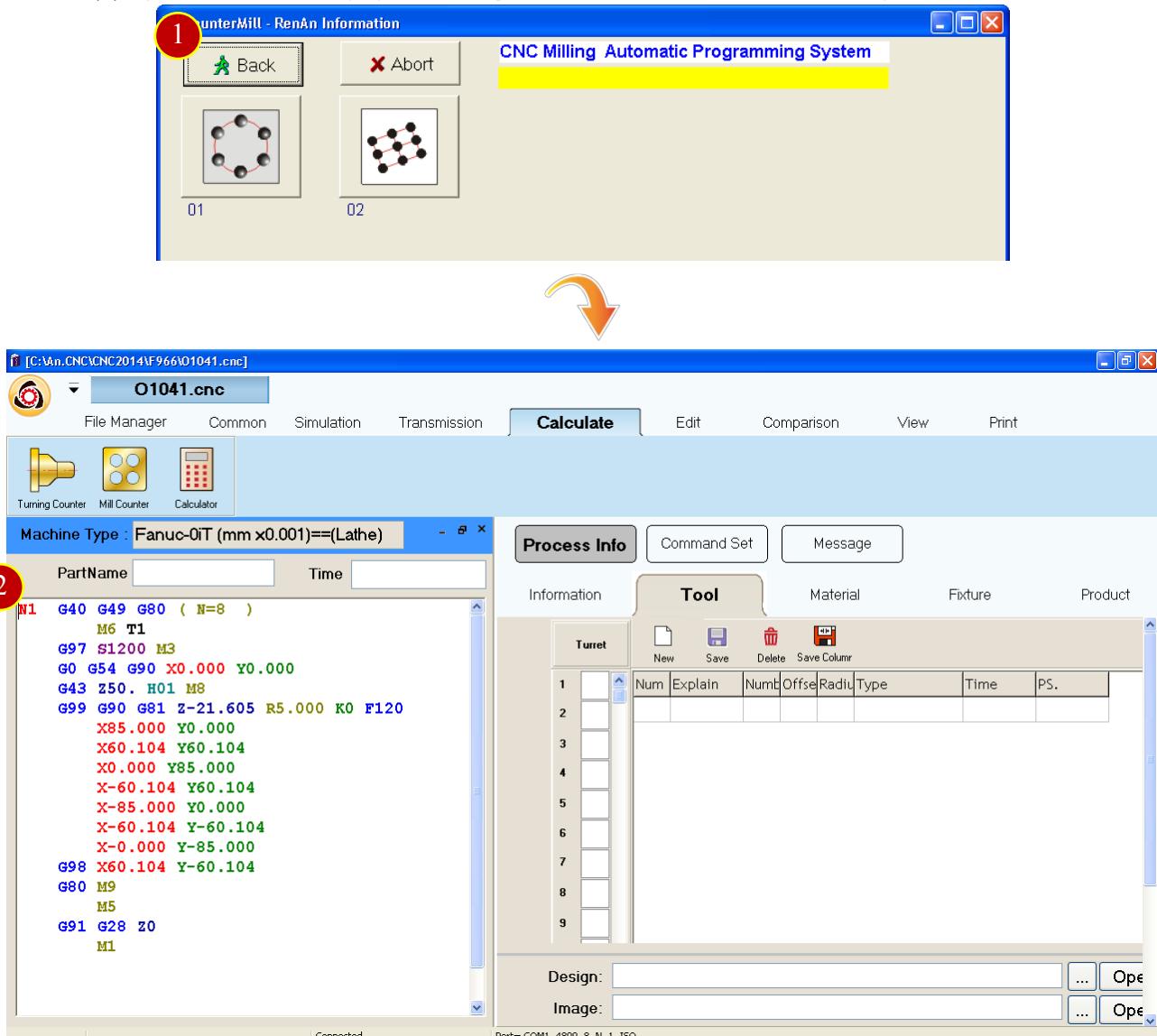
(4) Click 【Submit】 and preview the program

(5) Click 【Submit】 to Complete



5.2.6 Return to NcEditor system

- (1) Click 【Back】
- (2) System will copy your program to editor automatically



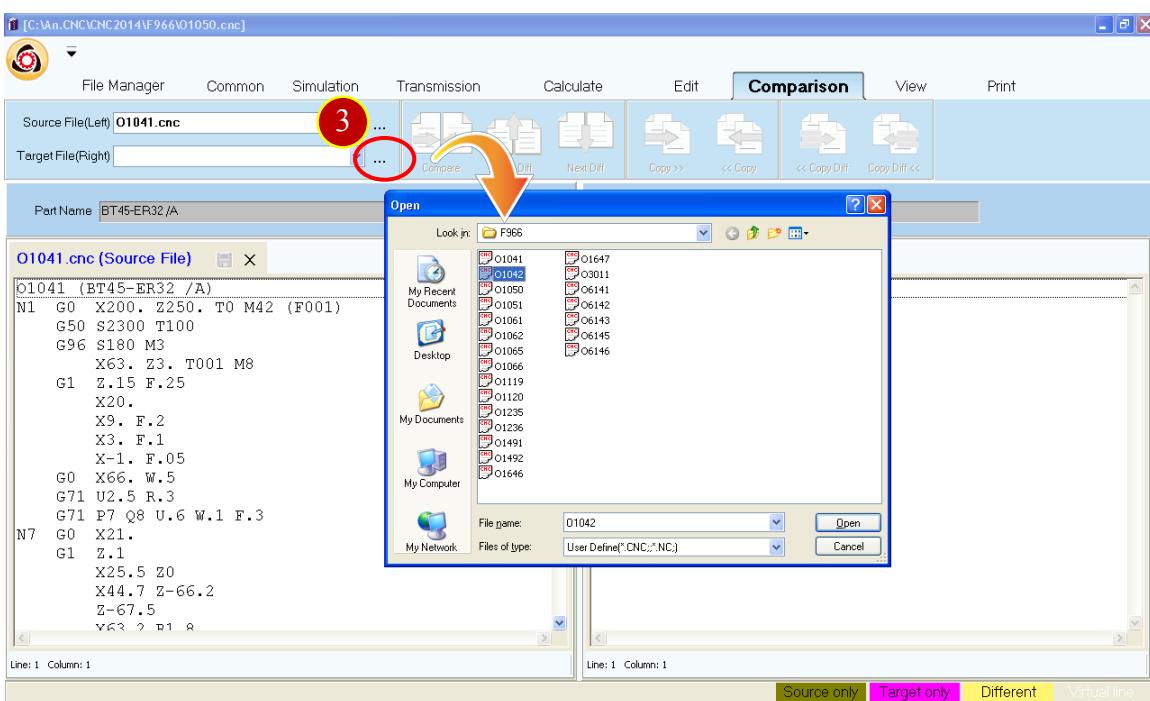
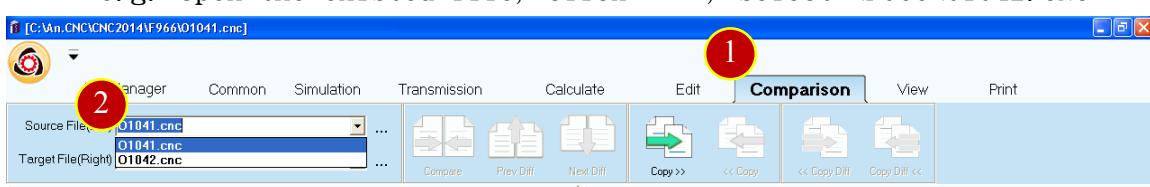
6. NC Code Comparison

6.1 Files comparison

The function "comparison" can compare the differences between two programs.

6.1.1 Open the comparison file

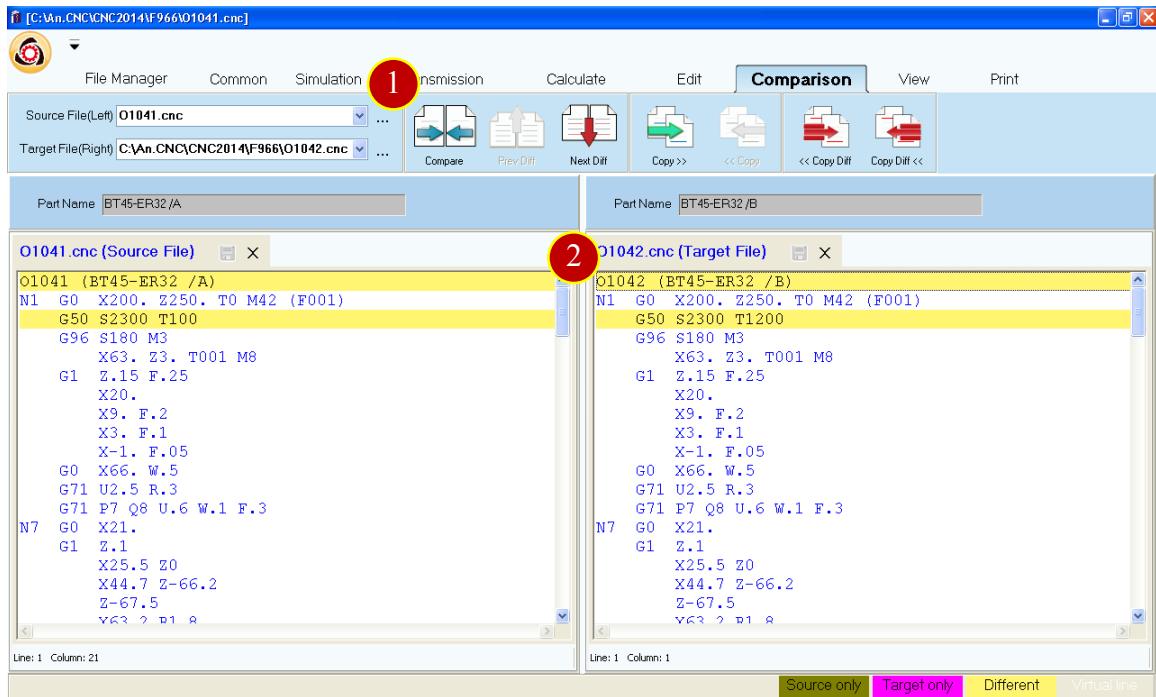
- (1) Change Function Tab to 【Comparison】
- (2) Select the source file (Left side)
 - e.g. select the file from the list, select 01014.CNC
- (3) Select the target file (Right side)
 - e.g. open the existed file, click ..., select \F966\01042.CNC



6.1.2 Compare

(1) Click 【Compare】 

(2) After the comparison, the differences will be shown with different colors
e.g. the yellow color is the different area



6.1.3 Difference editing

After comparing with each other, you can find a different area. Then, you can edit or copy it.

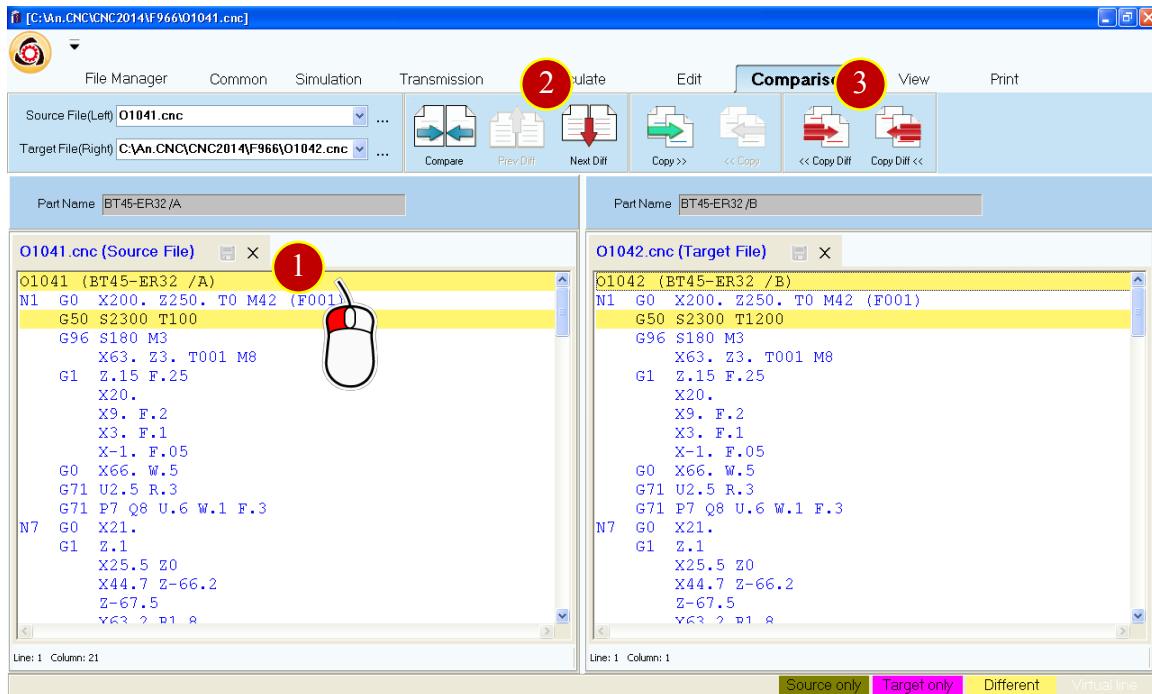
| Button | Name | Instruction |
|--------|--------------|---|
| | Prev Diff | To find the previous different area |
| | Next Diff | To find the next different area |
| | Copy >> | Copy source file to target file |
| | << Copy | Copy target file to source file |
| | Copy Diff >> | Copy different area in source file to target file |
| | << Copy Diff | Copy different area in target file to source file |

(1) Click the file below

e.g. click the first line in 01041.CNC

(2) Click 【Next Diff】 to the third line

(3) Click 【<< Copy Diff】



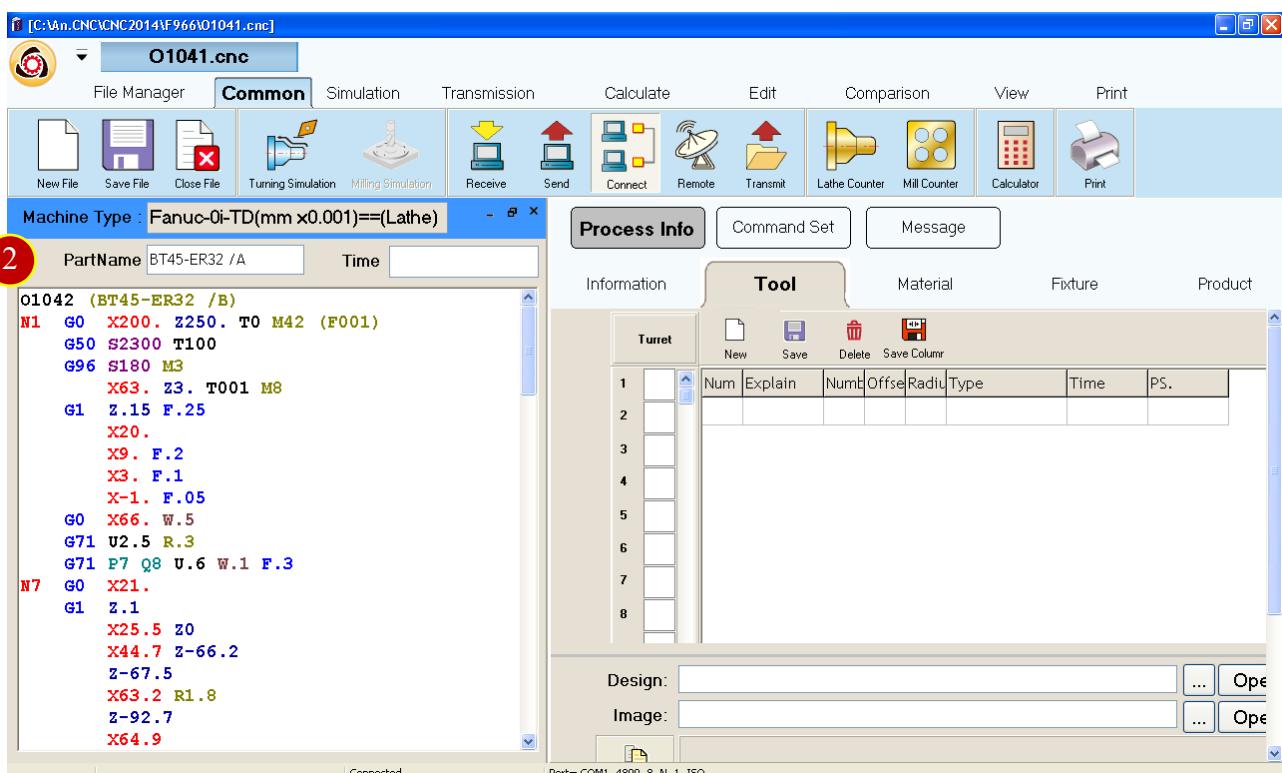
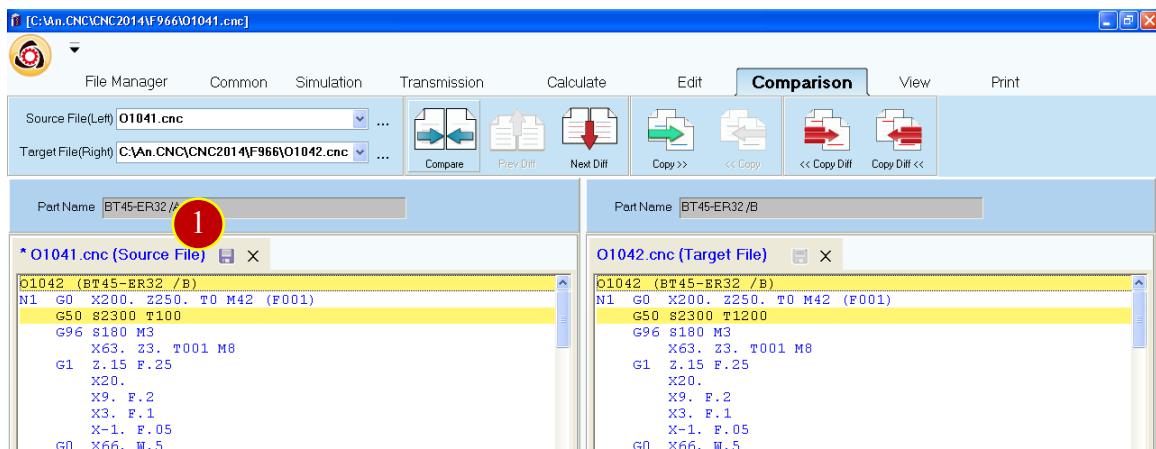
6.1.4 Save the comparison file

When you revise the context in the file, you can save at the same time.

Furthermore, you can synchronize with other file in the editor.

(1) Click **【Save】** in 01041.CNC

(2) Switch to **【Editor】** The file is modified



6.2 Comparison Settings

Change the Comparison options: font and the background colors

6.2.1 Comparison options

Change the comparison options

(1) Click 【Option】

(2) Click 【Set comparison】

(3) Check 【Comparison function】

(a) Space: compare the differences of spaces

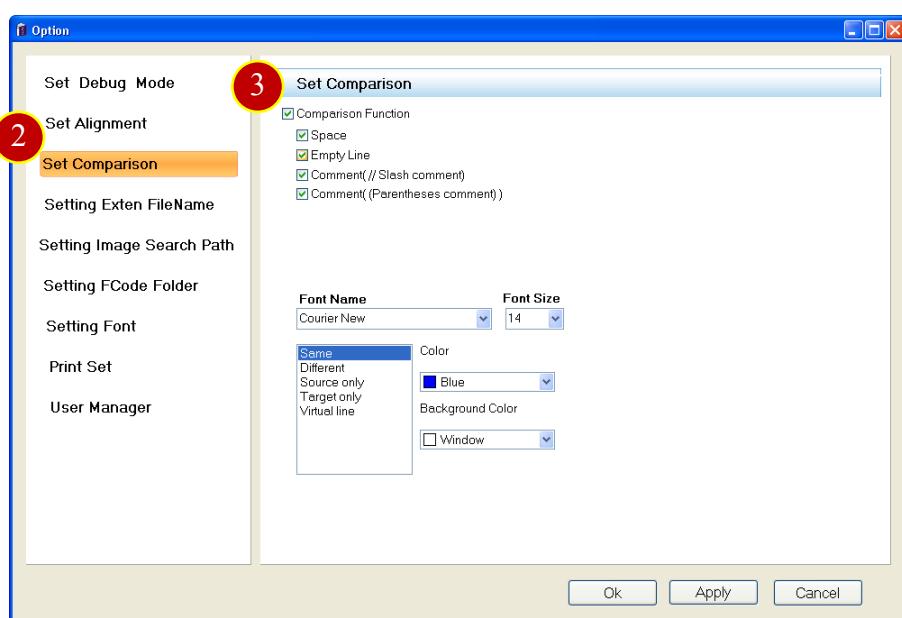
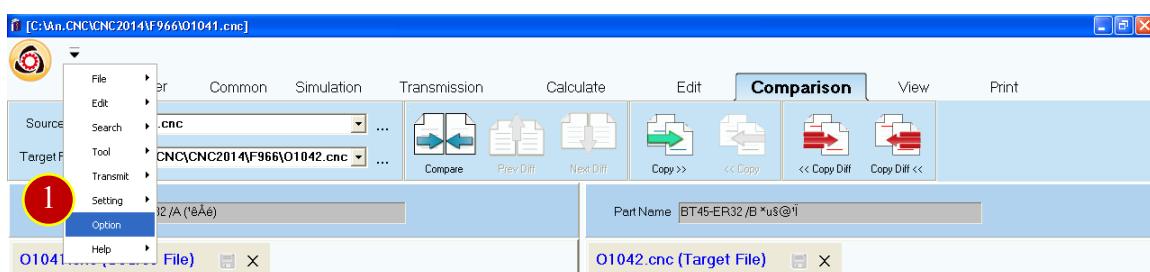
(b) Empty line: compare the differences of blank line

(c) Comment(// Slash comment): when you are comparing, the instruction will also be compared.

e.g. G01 X10. (Slash comment)

(d) Comment(Parentheses comment): when you are comparing, the instruction next to semicolon will be compared.

e.g. G01 X10.; Parentheses comment



6.2.2 Compare fonts and colors

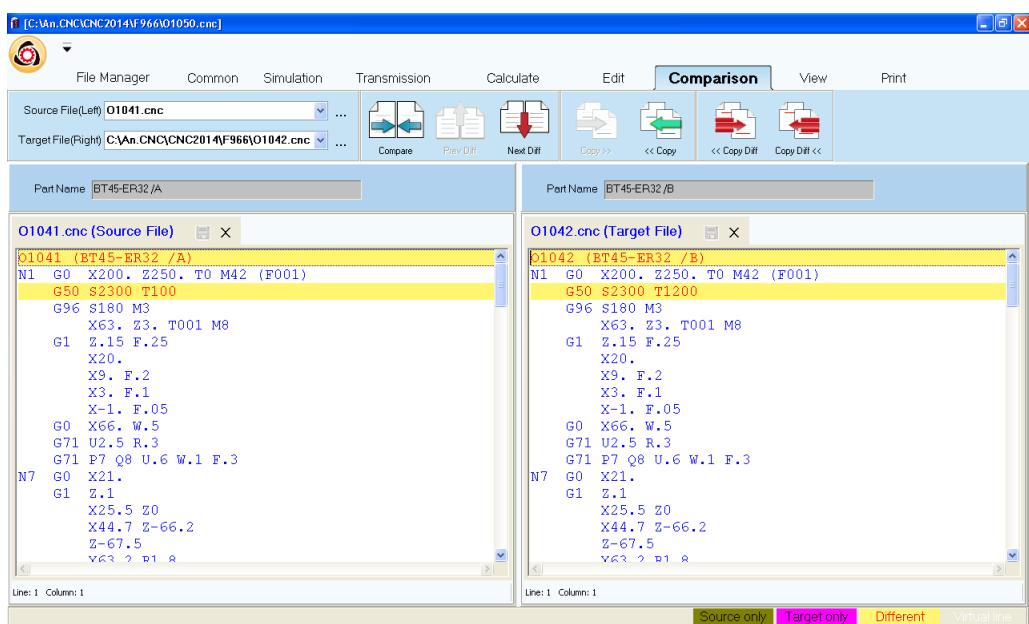
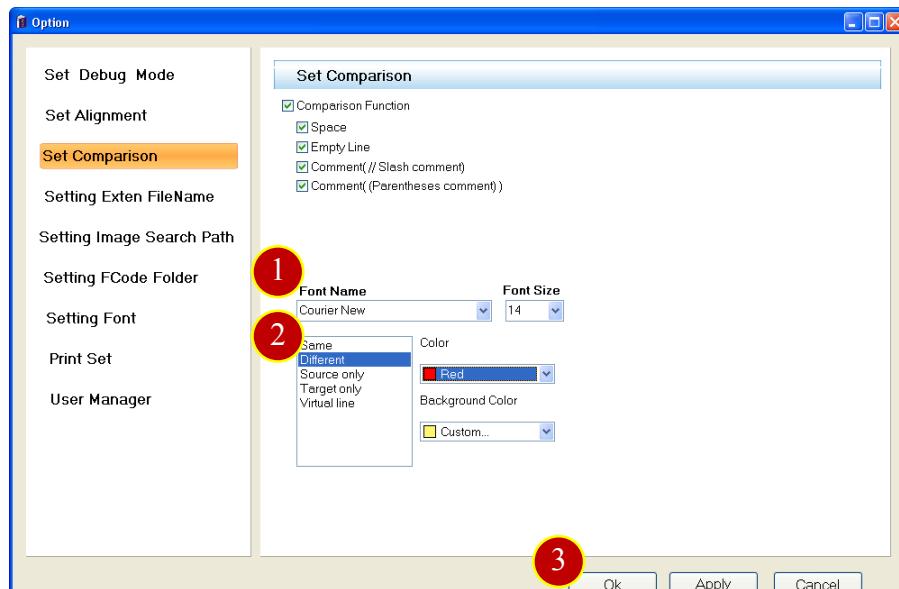
(1) Change the original font in comparison

e.g. font= Courier New, size=14

(2) Change the different color

e.g. select difference, context = Red

(3) Click 【OK】



7. Tools Management

7.1 Use Tool Magazines

Set the turret in the CNC program and click the tool to simulate cutting process.

7.1.1 Select “Turret”

To select the current turrets in CNC program, follow the steps below:

e.g. How to use turret NC38-02

- (1) When you open a new file, click 【Simulation】
- (2) Click 【Turret】 and set turrets



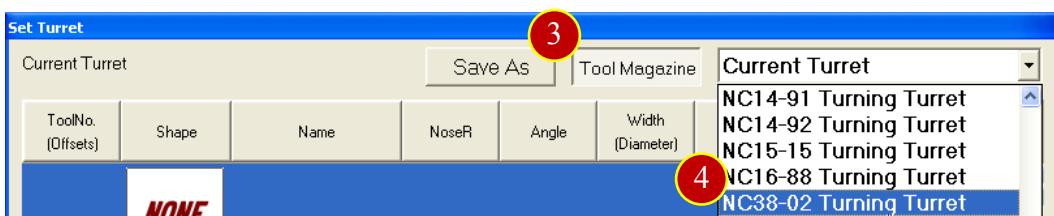
(3) Click 【Tool Magazine】

When you open tool magazine, it will overwrite the already exist tools.

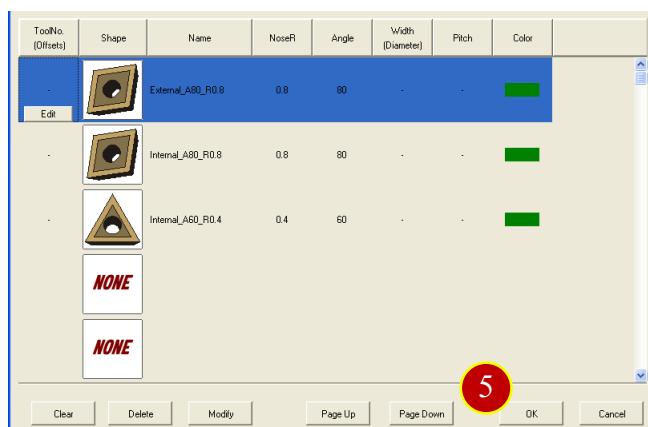
If you need to save the tools, please click 【Save As】 to save the current tools.

(4) Select the current turret

e.g. NC38-02 Turning Turret



(5) Click 【OK】



7.1.2 Type the tools explanation

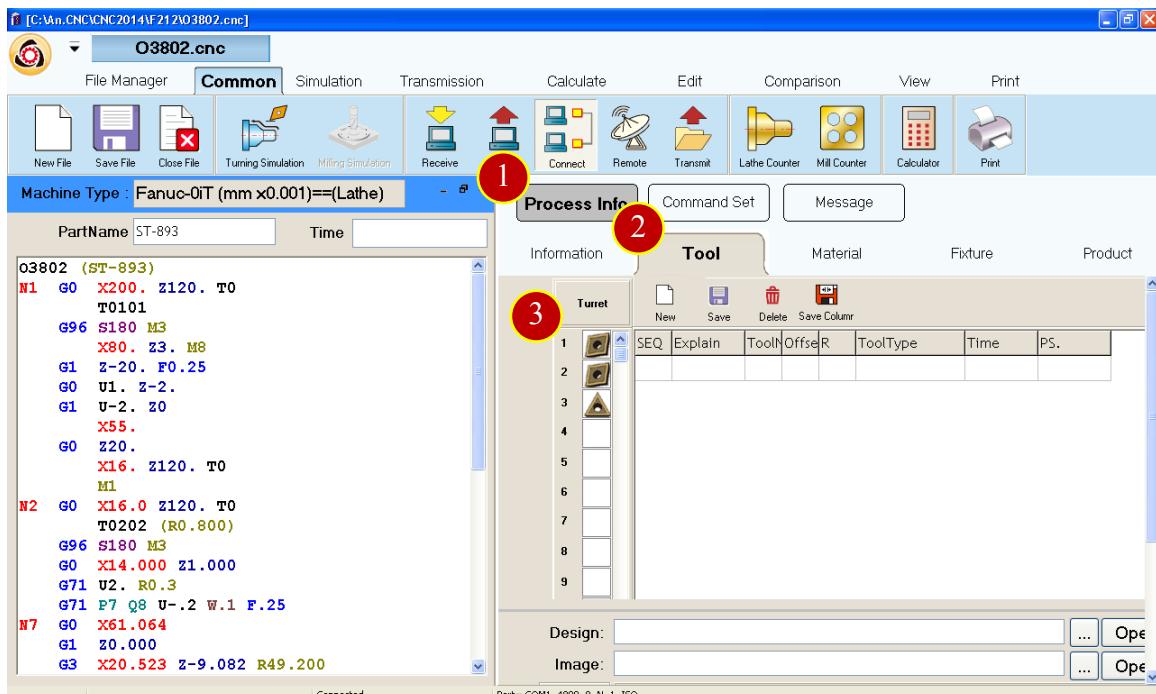
You can type some information to understand the respective process in a short time.

e.g. Write the tool explanation for program 03802

(1) Click 【Process Info】

(2) Click 【Tool】

(3) It will show the current tool for 03802



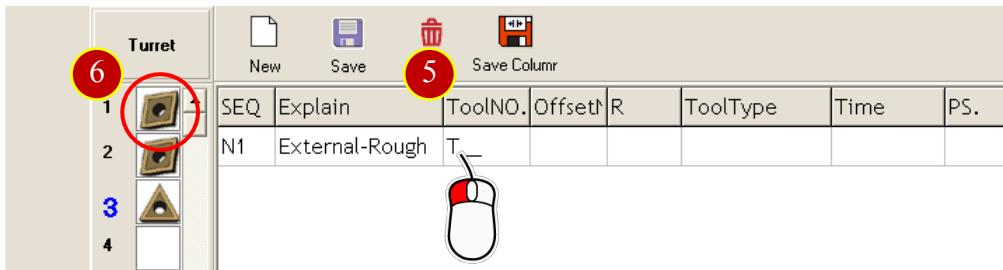
(4) In turret list, key in the tool explanation

e.g. You can click the 【SEQ】column and key in [N1] , and key in [External-Rough] in the [Explain] column

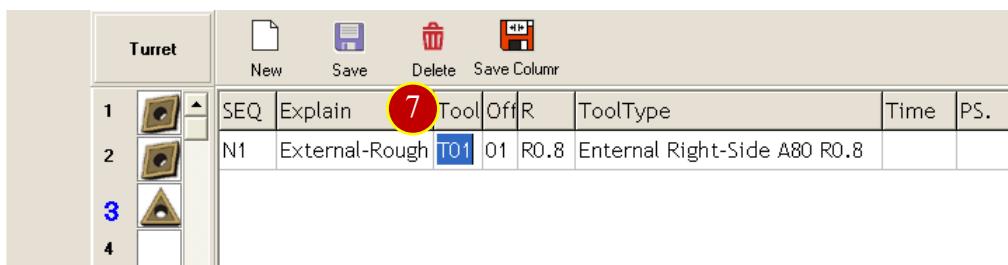
| Turret | SEQ | Explain | ToolNO. | OffsetNO. | R | ToolType | Time | PS. |
|--------|-----|----------------|---------|-----------|---|----------|------|-----|
| 1 | N1 | External-Rough | | | | | | |
| 2 | | | | | | | | |
| 3 | | | | | | | | |
| 4 | | | | | | | | |

Tools Management

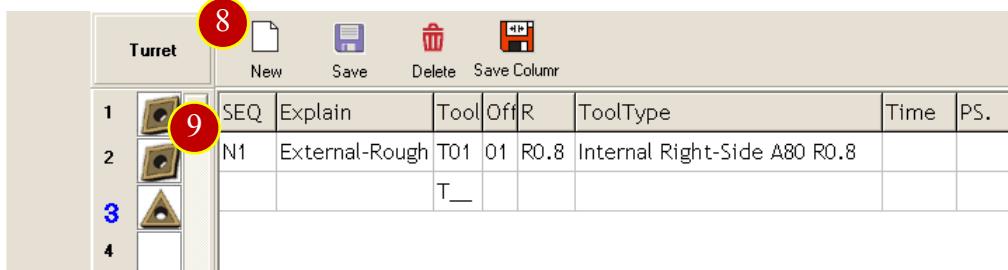
- (5) Click the 【ToolNO.】 column
- (6) Select the tool and click the tool pattern on the left
e.g. NO.1 tool



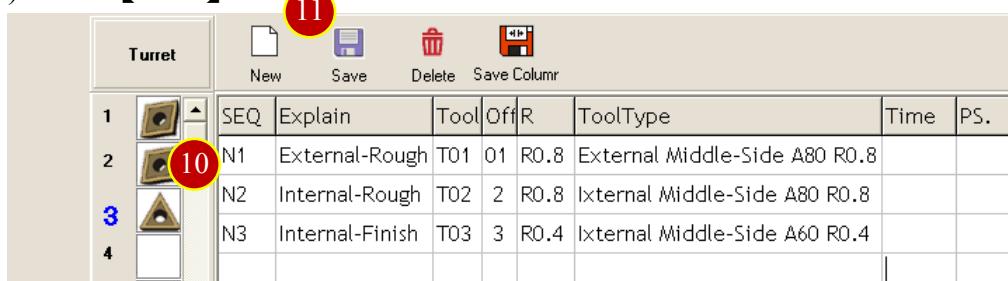
- (7) N0.1 the explanation of tool will copy to the column automatically
* you can also key in [01] by your own



- (8) Click 【New】 to add a new tool explanation
- (9) And the new tool explanation will appear



- (10) Follow the steps from ④ to ⑨ to write all explanations
- (11) Click 【Save】



7.2 Tool Magazines Management

Set the tools used on the turret, and you can manage the tools you are used to choosing.

7.2.1 Current Tool Management

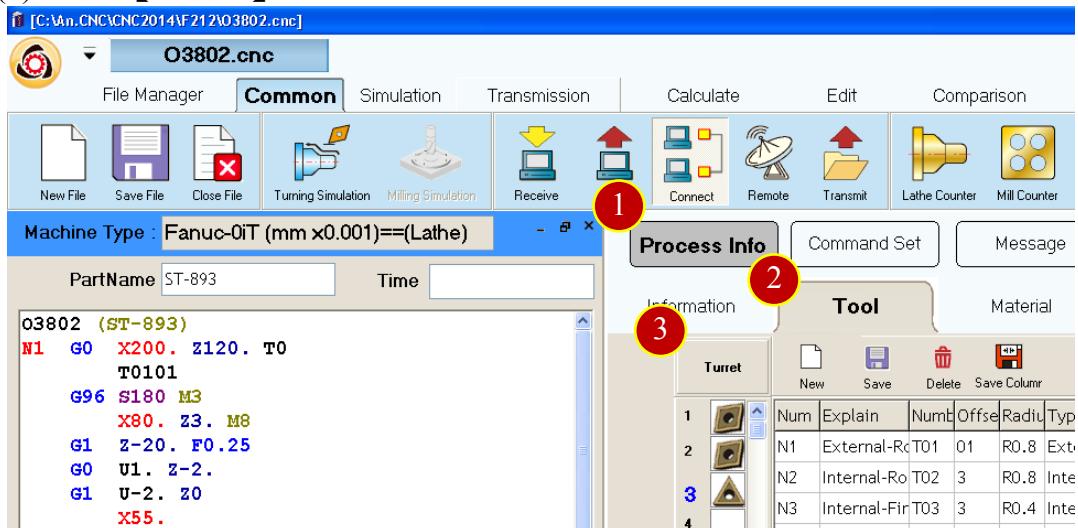
Manage the turret that used in the CNC program

e.g. Add 【External Groove 5MM R0.4】

(1) When you open the new file, click 【Process Info】

(2) Click 【Tool】

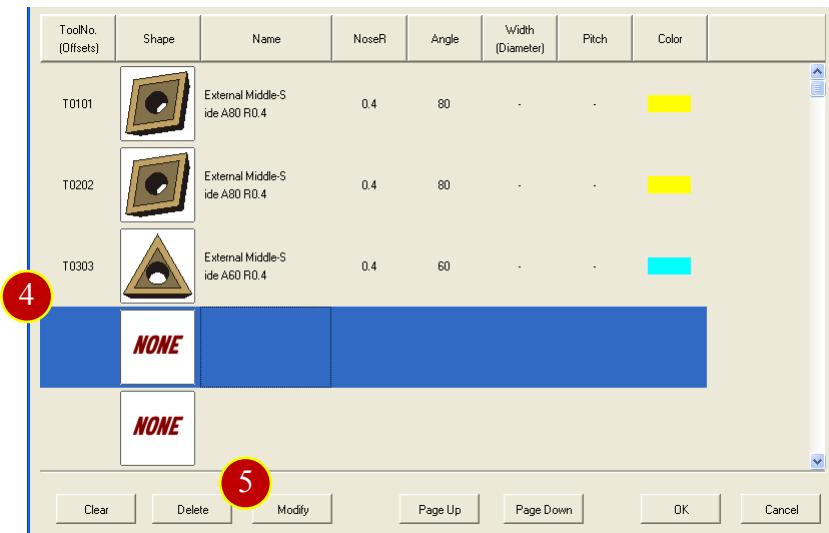
(3) Click 【Turret】 to show the current turret



(4) Click the tool list

e.g. The fourth one

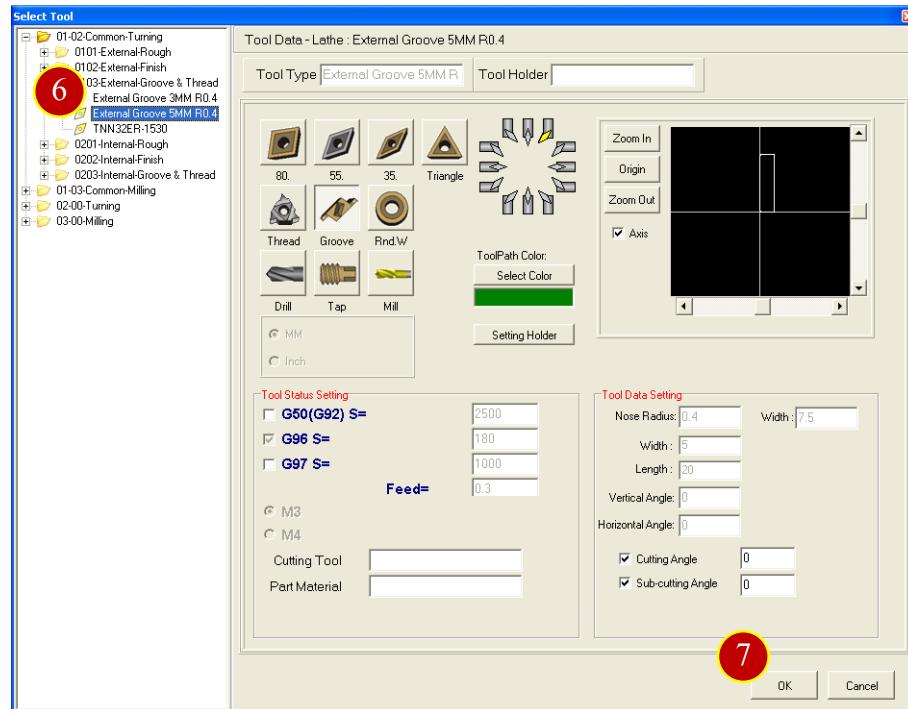
(5) Click 【Modify】



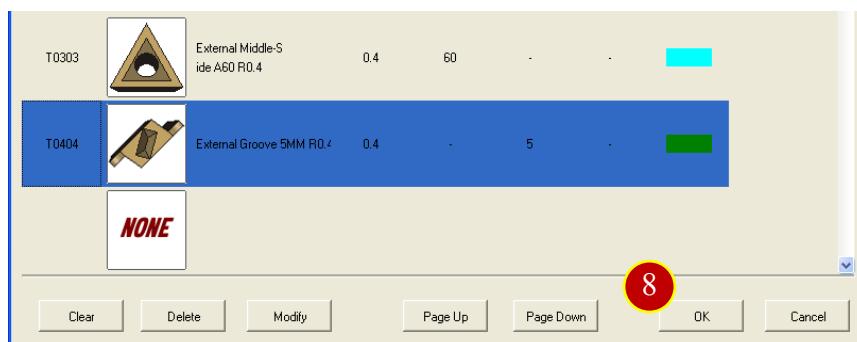
(6) Select the tool in your current file

e.g. 【External Groove 5MM R0.4】

(7) Click 【OK】 to add this new one



(8) Click 【OK】



(9) It will show in the list after you add a new tool

| Turret | | | | | | |
|--------|-----------------|------|------|----------|-------------------------------|-----|
| SEQ | Explain | Tool | OffR | ToolType | Time | PS. |
| N1 | External-Rough | T01 | 01 | R0.8 | External Middle-Side A80 R0.8 | |
| N2 | Internal-Rough | T02 | 2 | R0.8 | Internal Middle-Side A80 R0.8 | |
| N3 | Internal-Finish | T03 | 3 | R0.4 | Internal Middle-Side A60 R0.4 | |
| 4 | | | | | | |
| 5 | | | | | | |

7.2.2 Save the tool as another file

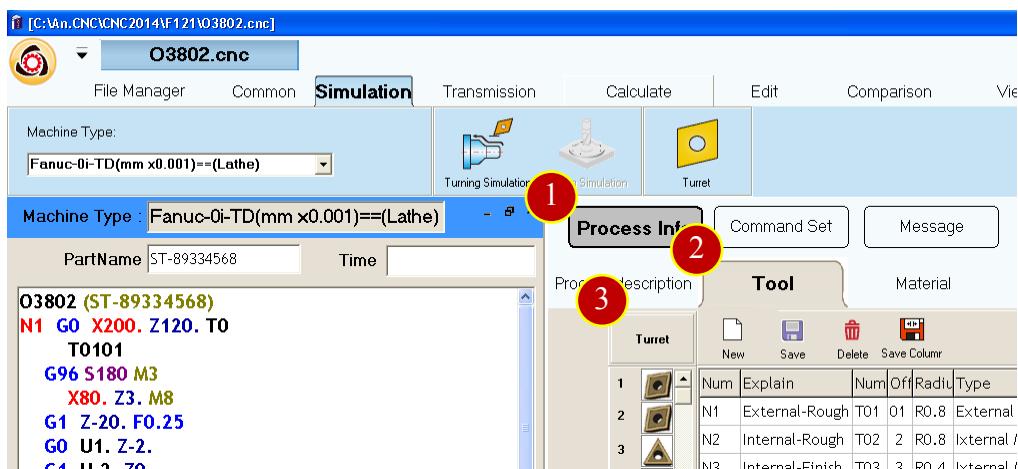
Save the current turret to magazine

e.g. Save 【T38-03 Turning Tool】 to a new one

(1) Open file and click 【Process Info】

(2) Click 【Tool】

(3) Click 【Turret】 to show the current turret



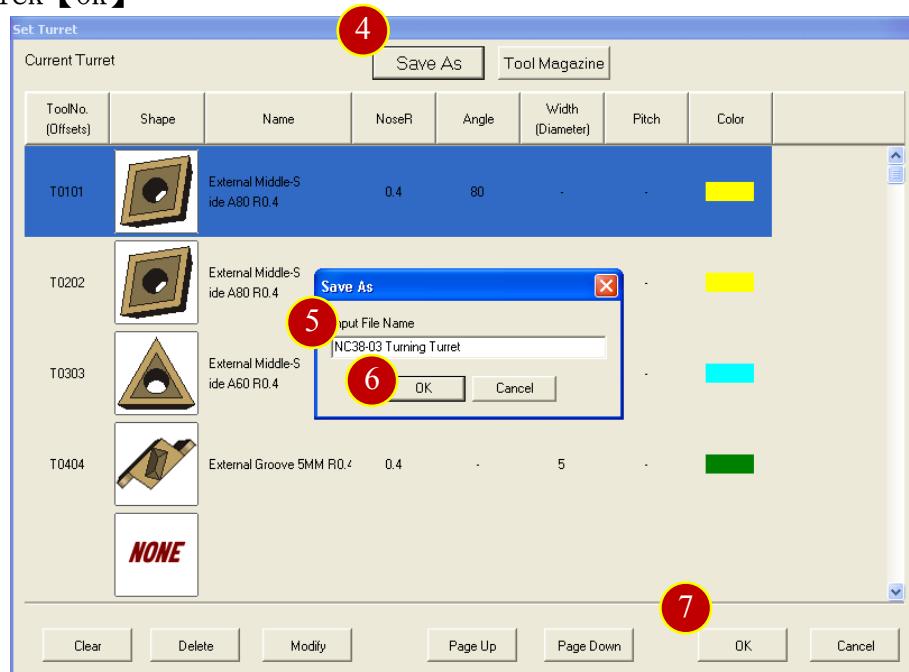
(4) Click 【Save as】

(5) Key in the turret file name

e.g. [NC38-03 Turning Turret]

(6) Click 【OK】

(7) Click 【OK】



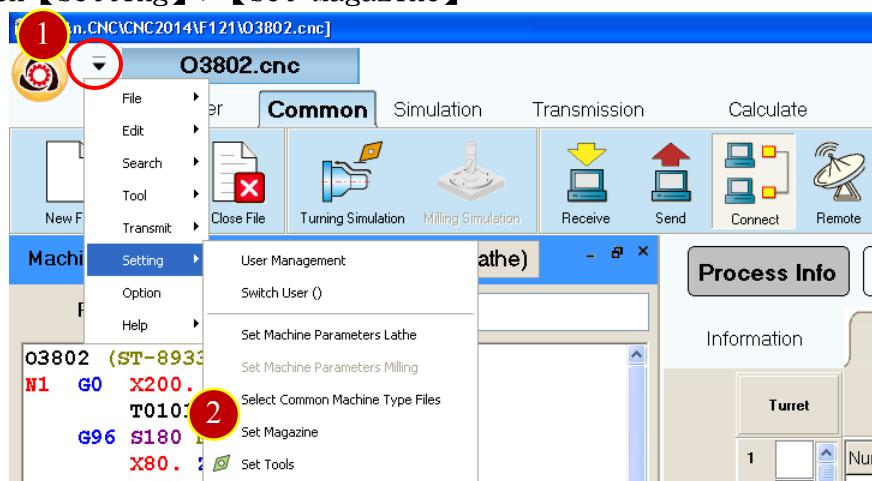
7.2.3 Add new turret

Add turret to make it easy to use.

e.g. Add 【NC01-04 Turning Tool】

(1) Click the pull-down list

(2) Click 【Setting】 > 【Set Magazine】

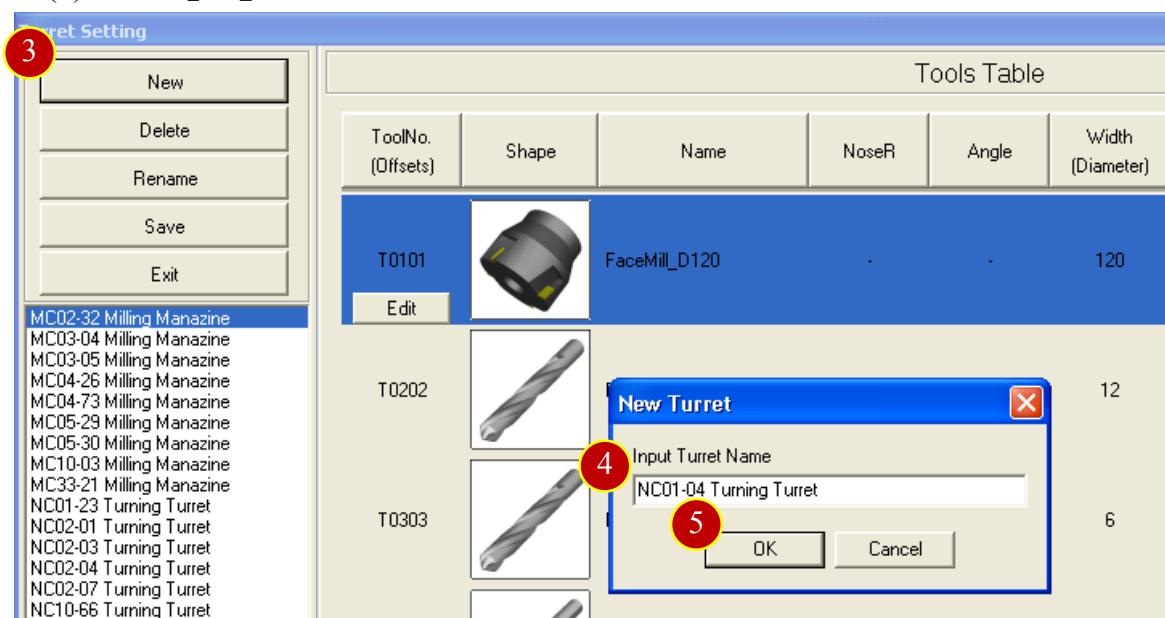


(3) Click 【New】

(4) Key in the turret name

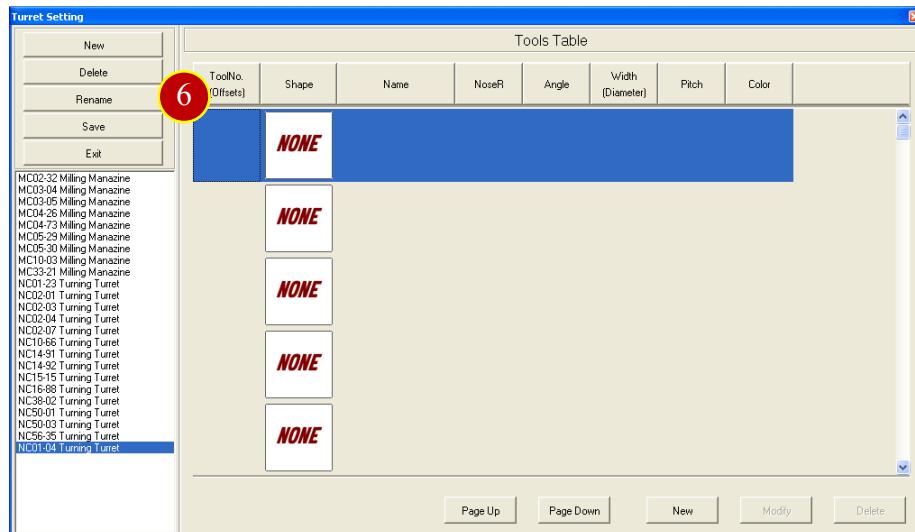
e.g. NC01-04 Turning Turret

(5) Click 【OK】



(6) Double click the tool blank

e.g. First blank

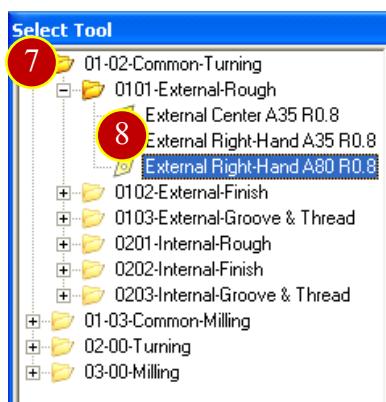


(7) Select the tool data

e.g. [01-02-Common - Turning] > [0101 External-Rough]

(8) Tool choosing

e.g. [External Right-Hand A80 R0.8]



(9) Click 【OK】



NcEditor MANUAL



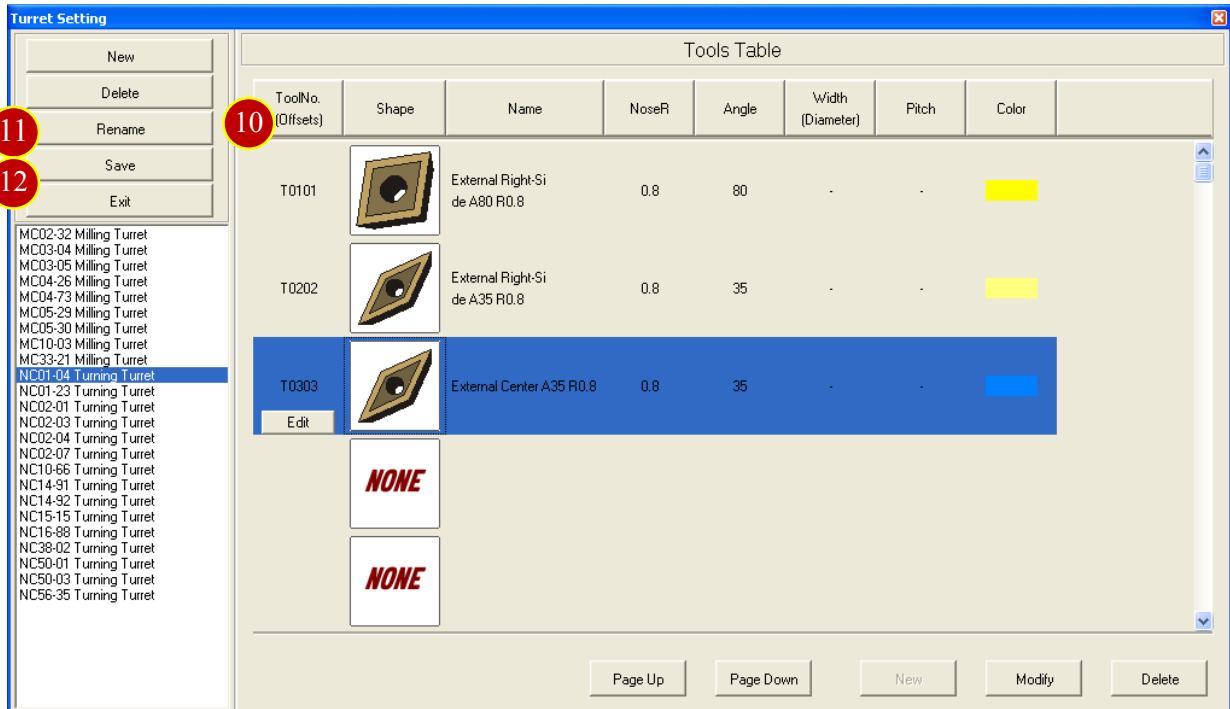
RenAn Information Technology Co., Ltd.
www.RenAn.com.tw

Tools Management

(10) Repeat the steps from ⑥ to ⑨ to finish the tool setting

(11) Click 【Save】

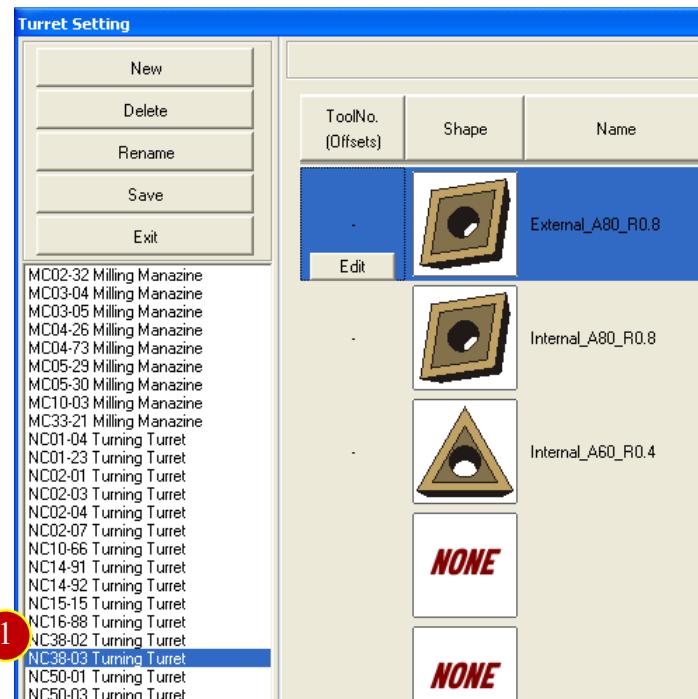
(12) Click 【Exit】



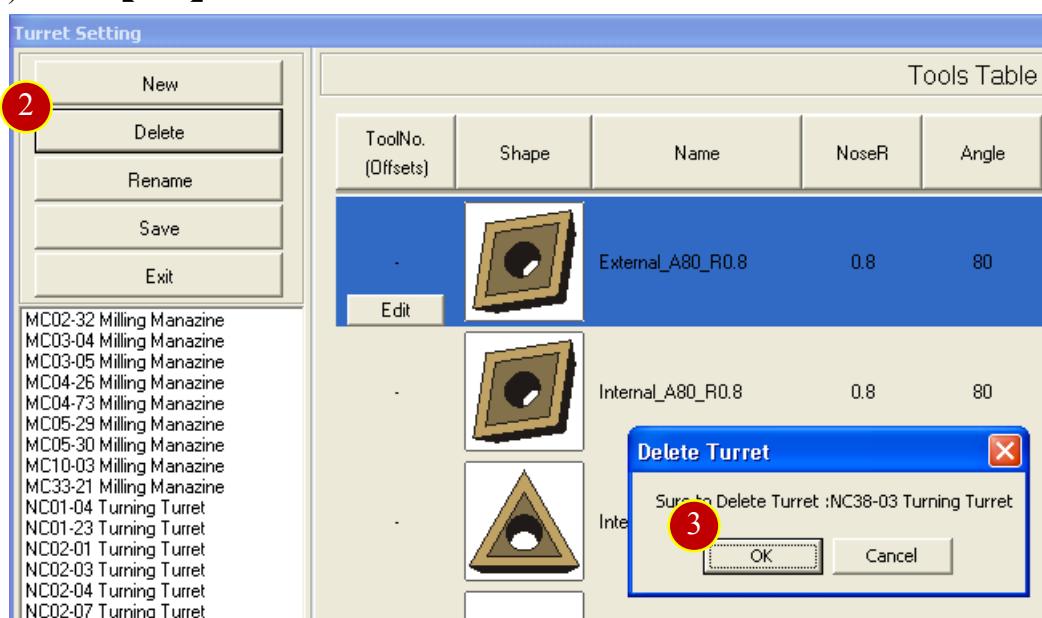
7.2.4 Delete the turret

When the tools are no longer used, you can delete
e.g. Delete 【NC38-03 Turning Turret】

- (1) Select the turret you want to delete
e.g. NC38-03 Turning Turret



- (2) Click 【Delete】
- (3) Click 【Yes】



7.3 Tool Data Management

You can add, delete, change name and modify parameter by managing the tool files

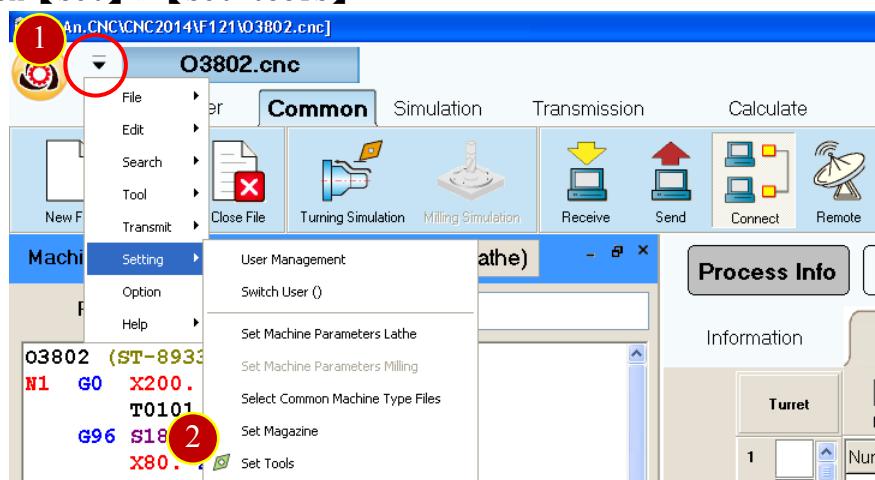
7.3.1 Add new tool

If the current tools do not reach your demand, you can add new tool and save it in the current tool.

e.g. Add tool 【External Middle-Side A80 R0.2】

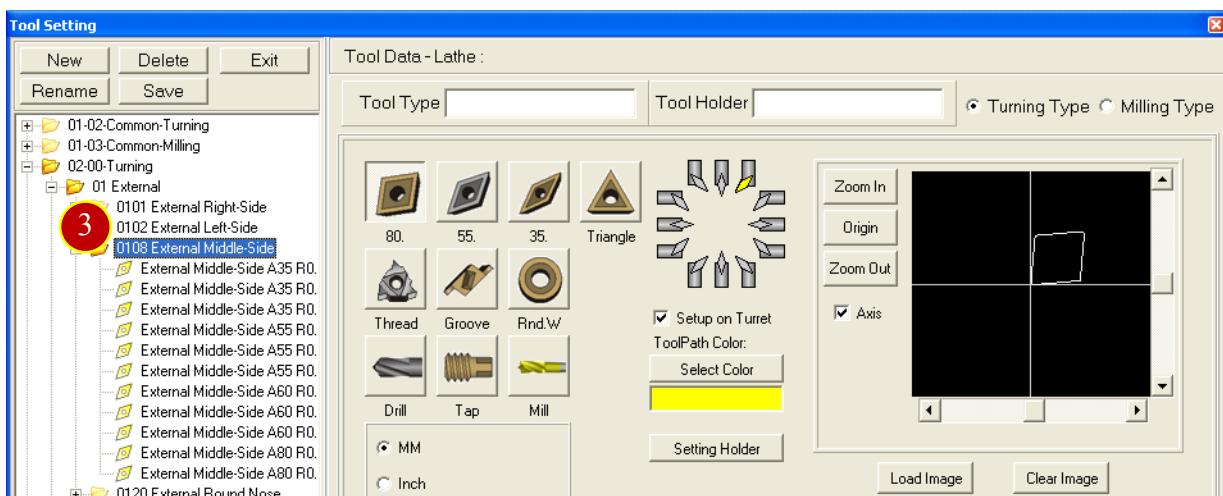
(1) Click pull-down list

(2) Click 【Set】 > 【Set tools】



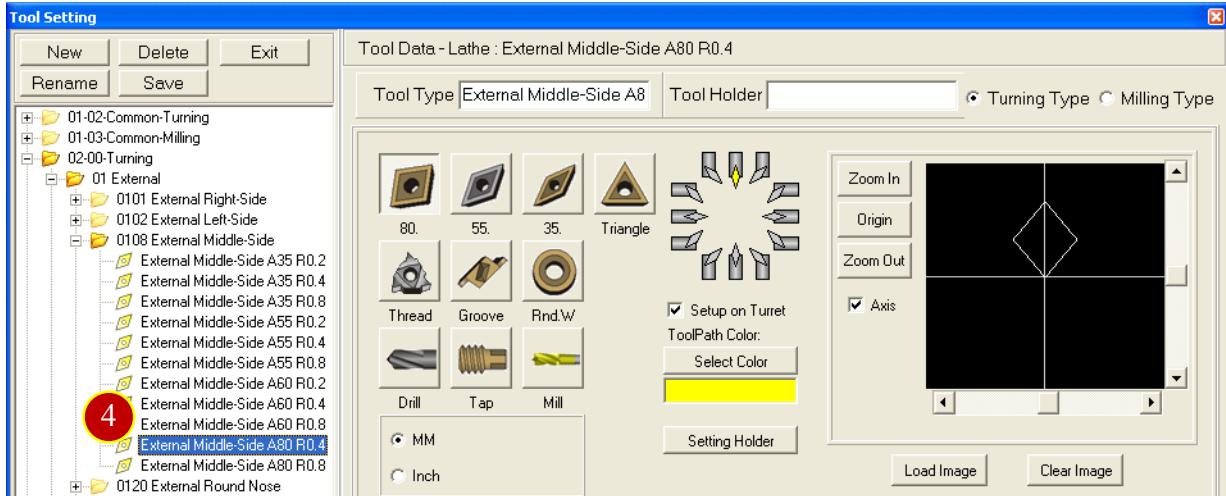
(3) Select the files

e.g. [02-00-Turning]>[External]>[0108 External Middle-Side]



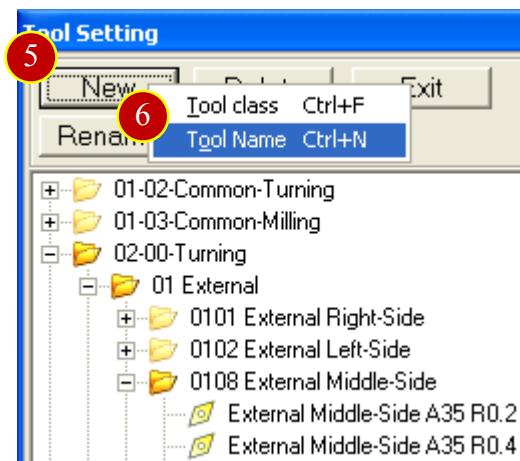
(4) Select the current tool which is similar to the new tool and select the new one. You can keep using part of parameter in a current tool to make it easier to modify the new one.

e.g. To add 【External Middle-Side A80 R0.2】 , you can select 【External Middle-Side A80 R0.4】 first



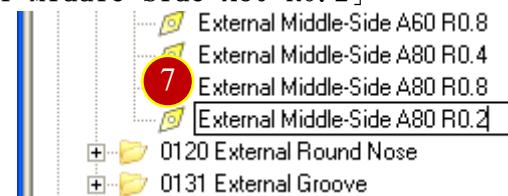
(5) Click 【New】

(6) Click 【Tool Name】



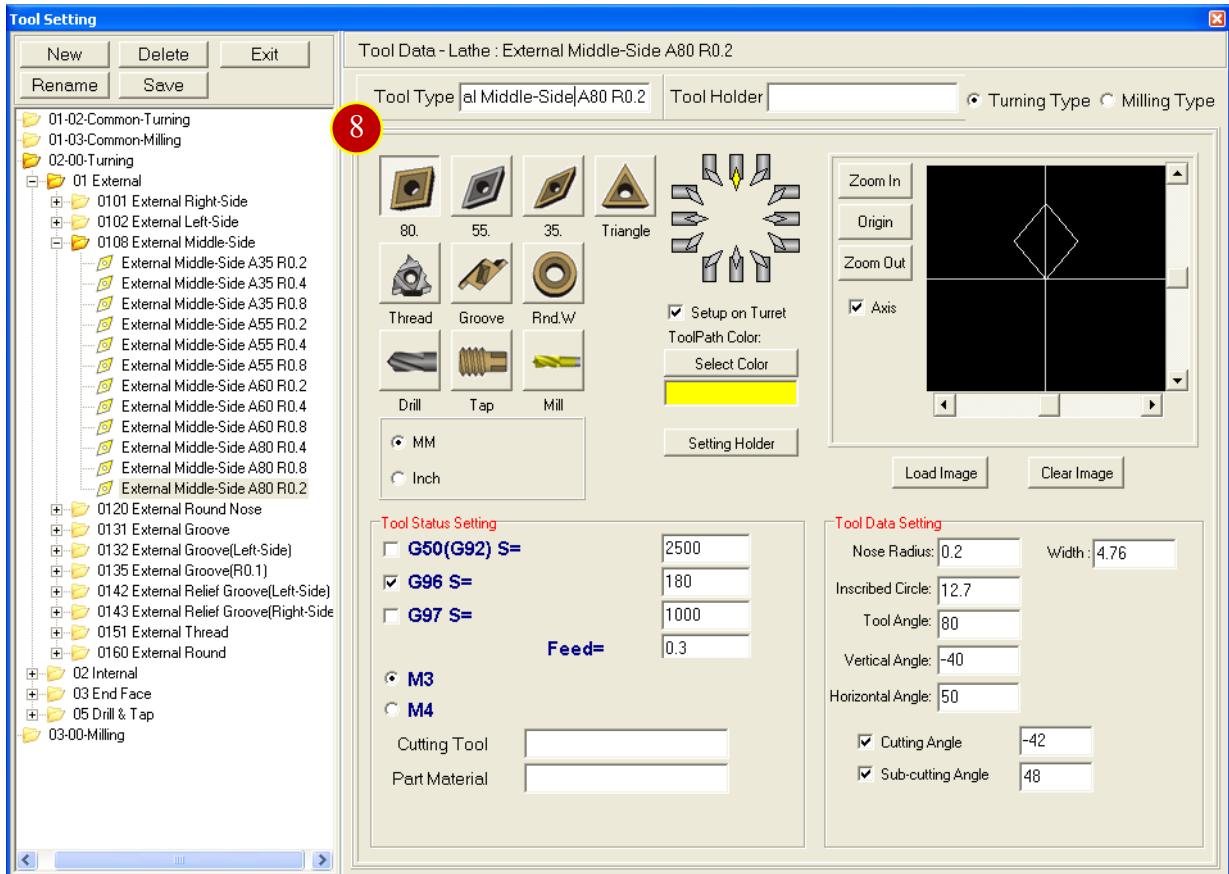
(7) The name of new tool will be set as NEW and you can change it as well.

e.g. [External Middle-Side A80 R0.2]



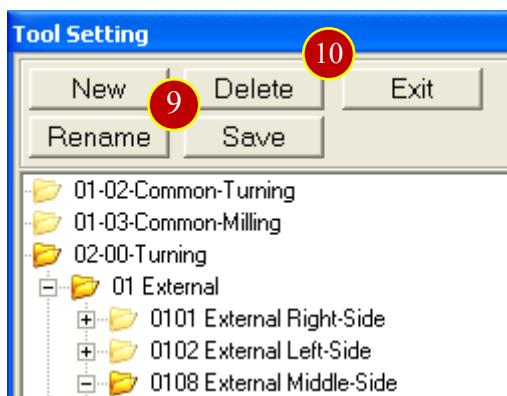
(8) Set the tool and the related parameter

e.g. Nose Radius = 0.2



(9) Click 【Save】

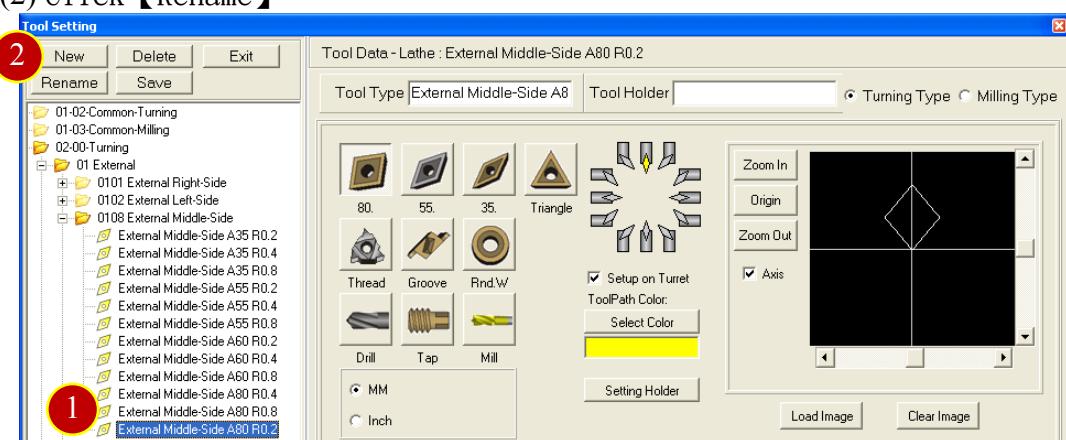
(10) Click 【Exit】



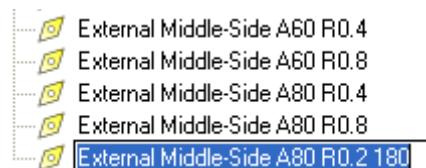
7.3.2 Change tools name

If tool's name or information are wrong, you can use it to correct your fault.
e.g. Change [External Middle-Side A80 R0.2] to [External Middle-Side A80 R0.2 S180]

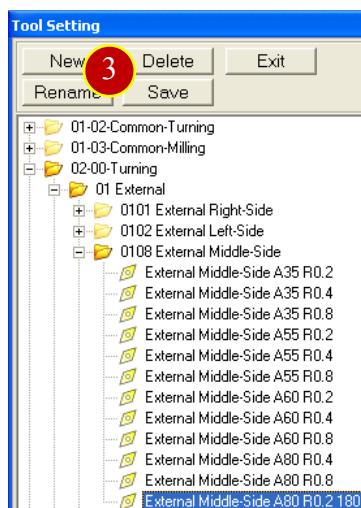
- (1) Select the tool you want to modify
e.g. [External Middle-Side A80 R0.2]
- (2) Click 【Rename】



- (3) Type the new name of the tool
e.g. [External Middle-Side A80 R0.2 180]



- (4) Click 【Save】



8. Machine Type File

Machine setting file store the specification, controller type, G code, M code, etc. It also store the related parameter. According to the setting, it will be much more precise when you debug the program, simulate the cutting path.

8.1 Select Machine Type File

To make the debug function correspond with the real machine, please use the actual machine type and select the correct machine type setting.

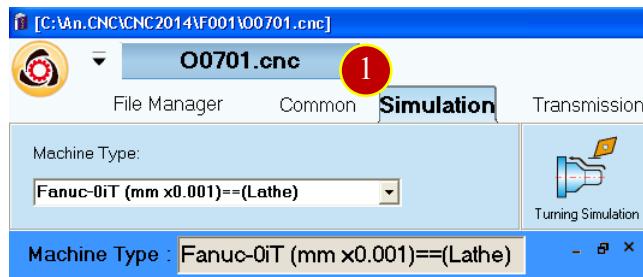
8.1.1 Select the type file

e.g. Machine type Fanuc-0i-TD, so you have to select

【Fanuc-0i-TD(mm x0.001)==(Turning)】

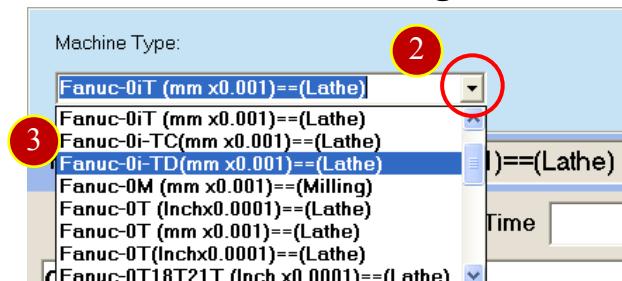
(1) Open file and click **【Simulation】**

e.g. Open **【F001\00701.CNC】**

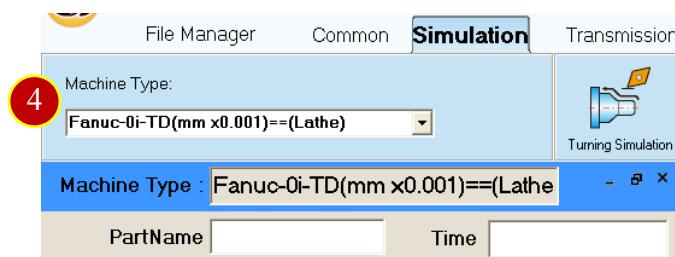


(2) Click **【Machine Type】**

(3) Select **【Fanuc-0i-TD(mm x0.001)==(Turning)】**



(4) Complete



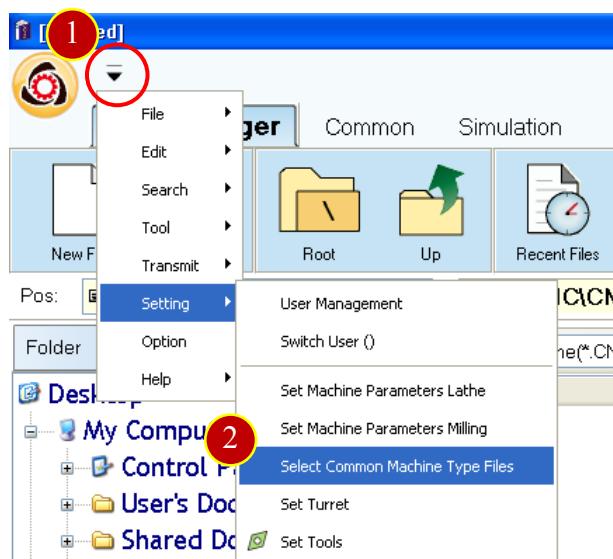
8.2 Common Machine Setting File

To reduce the waste time of choosing the settings, you can add the machine type in common use to the common use list. And, you can delete those files you seldom use.

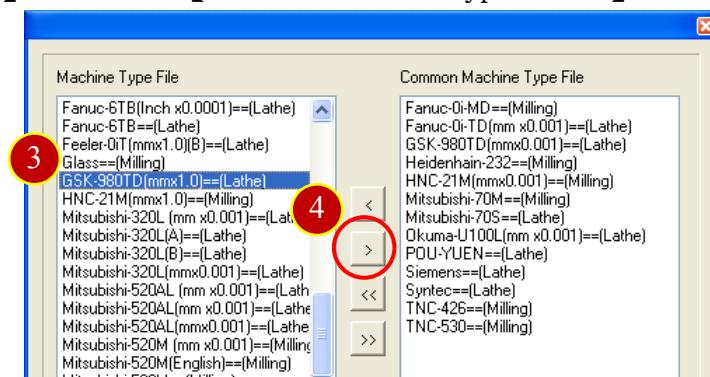
8.2.1 Add new setting to common use

e.g. Add 【GSK-980TD-GSK(mm x1.0)==(Turning)】 into the common list

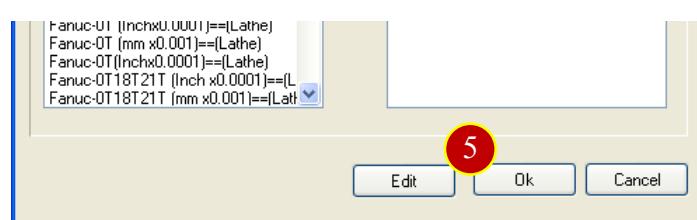
- (1) Click pull-down list
- (2) Click 【Setting】 > 【Select Common Machine Type Files】



- (3) Click 【GSK-980TD-(mm x1.0)==(Turning)】 in 【Machine Type Files】
- (4) Click 【>】 to add to 【Common Machine Type Files】



- (5) Click 【OK】

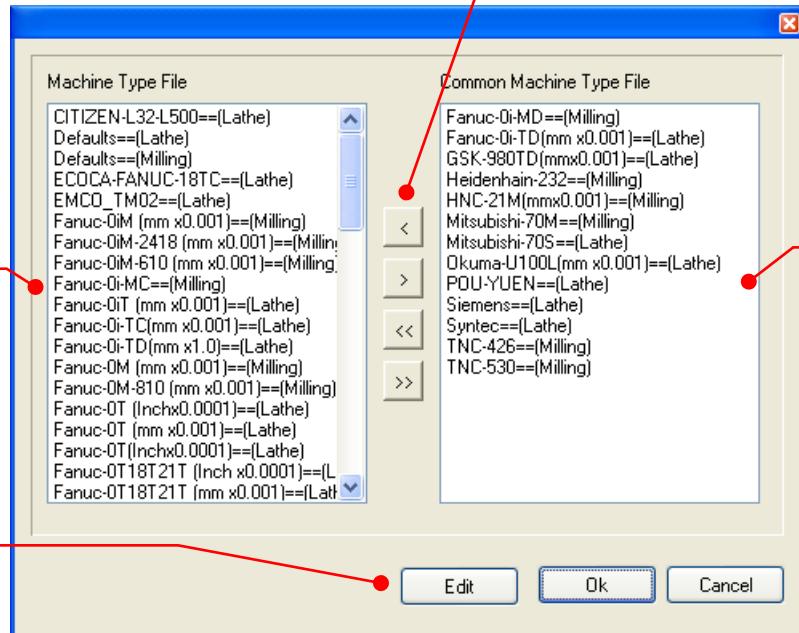


8.2.2 Operation Interface

Common machine setting instruction:

All Machine type list

Edit setting



Common machine type list

Description of setting function:

| Setting button | Description |
|----------------|--|
| < | Delete the chosen setting in common list |
| > | Add the chosen setting to common list |
| << | Delete all settings in common list |
| >> | Add all chosen settings to common list |

8.3 Machine Type Parameters Settings

When the parameter is not correspond with the actual machine, you can modify the parameter.

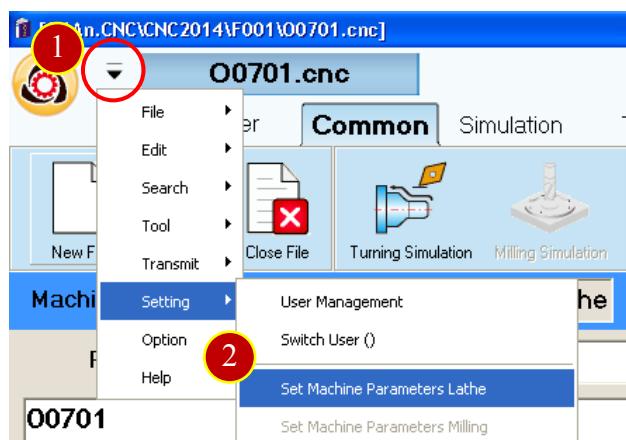
8.3.1 Machine parameter modification

e.g. In the setting of 00701.CNC 【Fanuc-0iT】 , modify the max. spindle speed to 4000

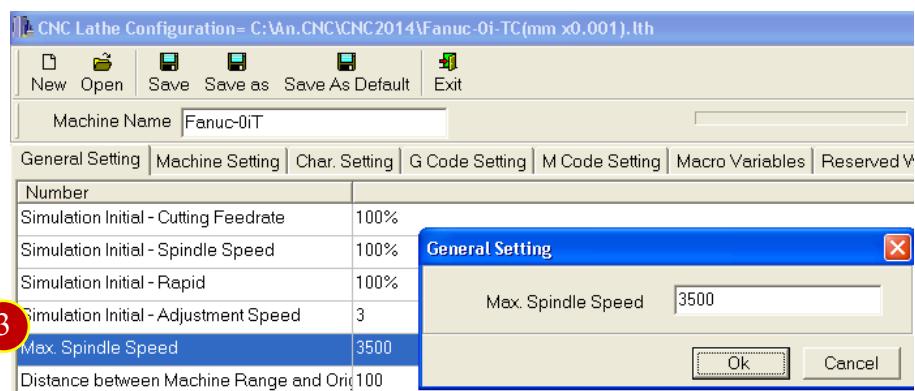
(1) After open the CNC file, click pull-down list

e.g. F001\00701.CNC

(2) Click 【Setting】 > 【Set Machine Parameter Lathe】



(3) Double click Max. Spindle Speed



(4) Key in [4000]

(5) Click 【OK】



NcEditor MANUAL

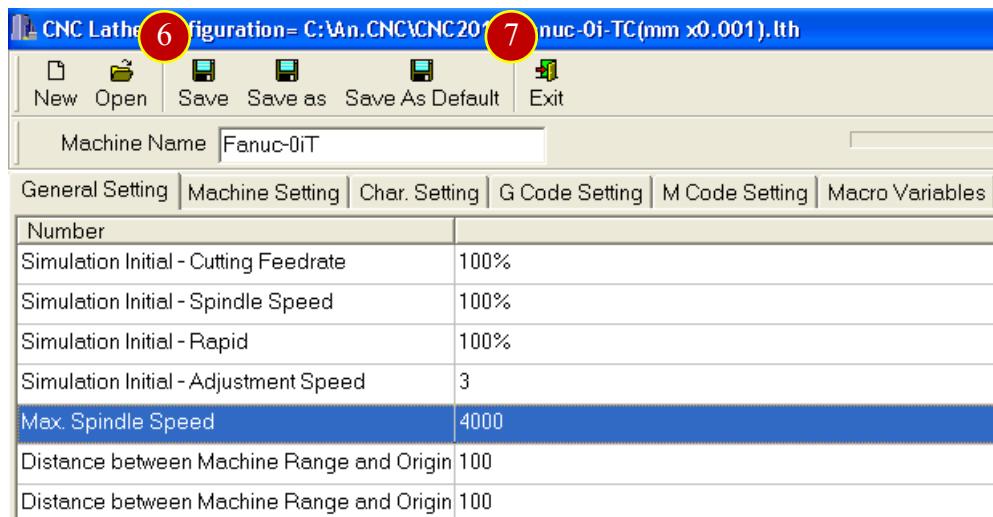


RenAn Information Technology Co., Ltd.
www.RenAn.com.tw

Machine Type File

(6) Click 【Save】

(7) Click 【Exit】



9. CNC Program Simulation (Turning and Milling)

After editing CNC programs, you can check the result and the accuracy of cutting path by using the simulator.

9.1 Material Setting for Turning

To show the cutting process and the products precisely, you should first set Turning Material

9.1.1 Material setting

e.g. Adjust the material length of F121\03802.CNC to 101

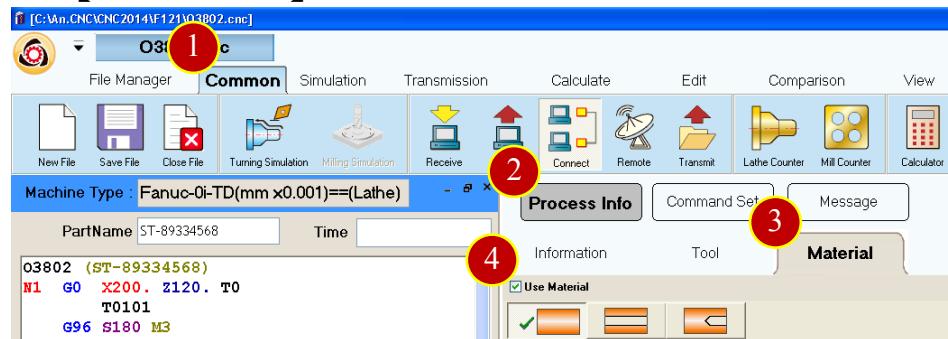
(1) After open the CNC file, click 【Common】

e.g. open F121\03802.CNC

(2) Click 【Process Info】

(3) Click 【Material】

(4) Check 【Use Material】

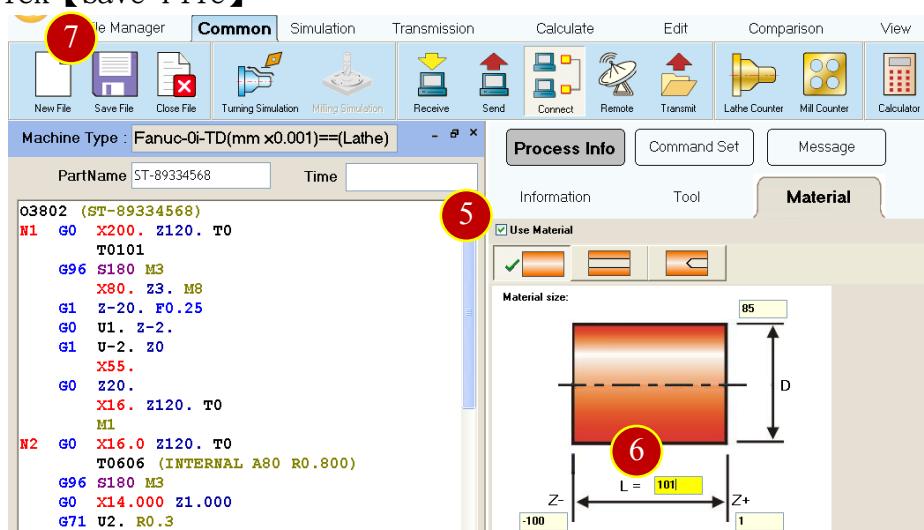


(5) Click 【Solid Material】

(6) Click L= to edit the length

e.g. 101

(7) Click 【Save File】



9.2 Simulation Operation and Settings for Turning

9.2.1 Execute "Turning Simulation"

e.g. Simulate the Turning program F121\03802.CNC

(1) After open the CNC file, click 【Common】

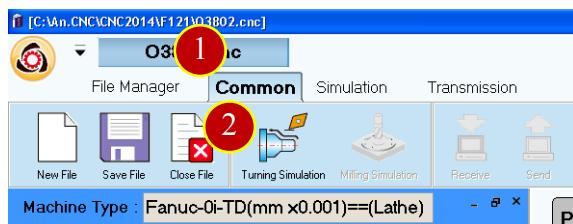
e.g. open F121\03802.CNC

(2) Click 【Turning Simulation】

* When the warning appears, please check some items below:

(a) Machine type set completely

(b) Machine type is Turning Machine

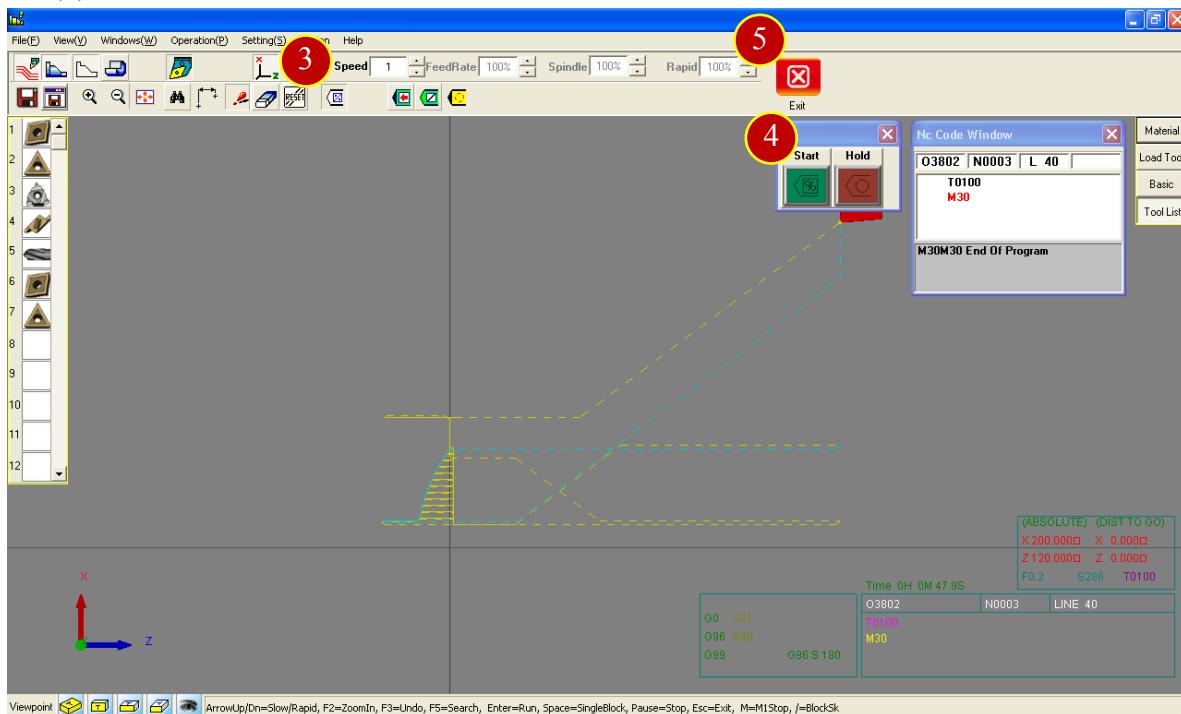


(3) Click 【Cycle Start】 to display control area

(4) After open it, it will run automatically

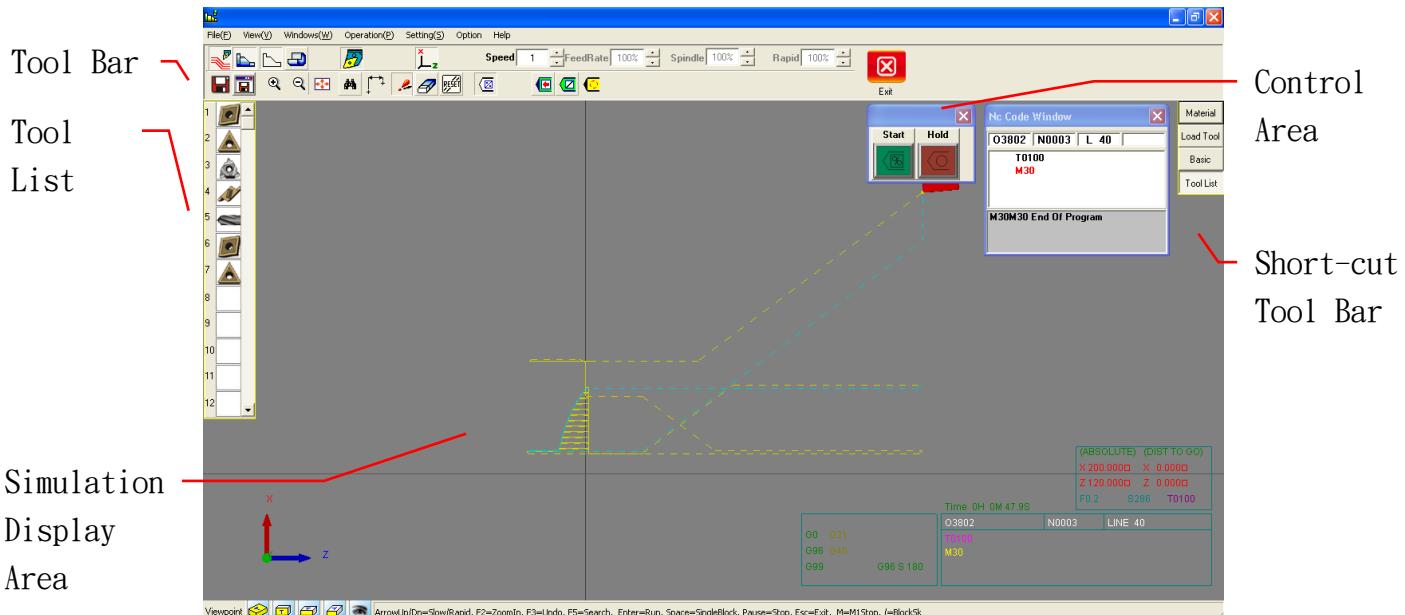
If it is not run, you can click 【Start】

(5) Click 【Exit】 to return to NcEditor



9.2.2 Operation of Turning Simulation

Instructions for the Turning Simulation software



Simulation Display Area

| Mouse Operation | Instruction |
|---------------------------------|----------------------|
| Mouse Wheel, Scrolling Forward | Zoom out the pattern |
| Mouse Wheel, Scrolling Backward | Zoom in the pattern |
| Middle Click & Drag | Move the pattern |
| Click & Drag | Magnify the pattern |

Control Area

| Button | Name | Instruction |
|--------|-------|-------------------|
| | Start | Start the program |
| | Hold | Hold the program |

Convenient Tool List

| Button | Name | Instruction |
|--------|-----------|----------------------------|
| | Material | Show or hide the material |
| | Load Tool | Load the tool information |
| | Basic | Set the basic tools |
| | Tool List | Show or hide the tool list |



Tool bar

| Button | Name | Instruction |
|--------|-----------------|------------------------------------|
| | Path | Show or hide the path |
| | Solid | Show or hide the solid |
| | Part | Show or hide the part |
| | Solid section | Show or hide the solid section |
| | Tool | Show or hide the tool |
| | Axes | Show or hide the XZ axes line |
| | Speed | Adjust the speed of simulation |
| | Feed Rate | Adjust the feed rate |
| | Spindle | Adjust the spindle override |
| | Rapid | Adjust the rapid rate |
| | Save | Save current program setting |
| | Save state | Save current environment state |
| | Zoom in | Zoom in the pattern |
| | Zoom out | Zoom out the pattern |
| | Initial view | Initial size of the pattern |
| | Search sequence | Search sequence in the CNC program |
| | Dimension panel | Show or hide the dimension panel |

CNC Program Simulation (Turning and Milling)

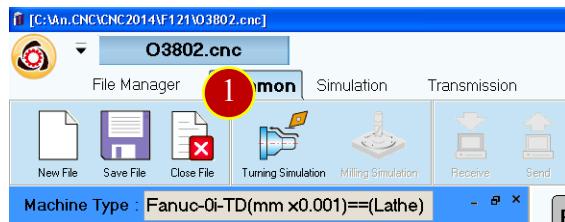
| | | |
|--|----------------|---|
| | Message Switch | Turn on or off message |
| | Clean Screen | Clean path in display area |
| | Reset | Reset the simulation |
| | Cycle Start | Auto-run the program Open Control Area Display 【Cycle Start】 & 【Hold】 Button |
| | S.B.K | Run single block of program and pause |
| | B.D.T | Ignore program after the sign “/” in running |
| | M01 | M01 pause state |
| | Exit | Close simulation |

9.2.3 Turning Simulation - Tool Setting

If you need to adjust, add modify the cutting tools during the simulation, follow the steps below.

e.g. when you are in the simulation of F121\03802. CNC, you can select the drill 【Drill-D05. 2】

(1) Open the program F121\03802. CNC, click 【Turning Simulation】

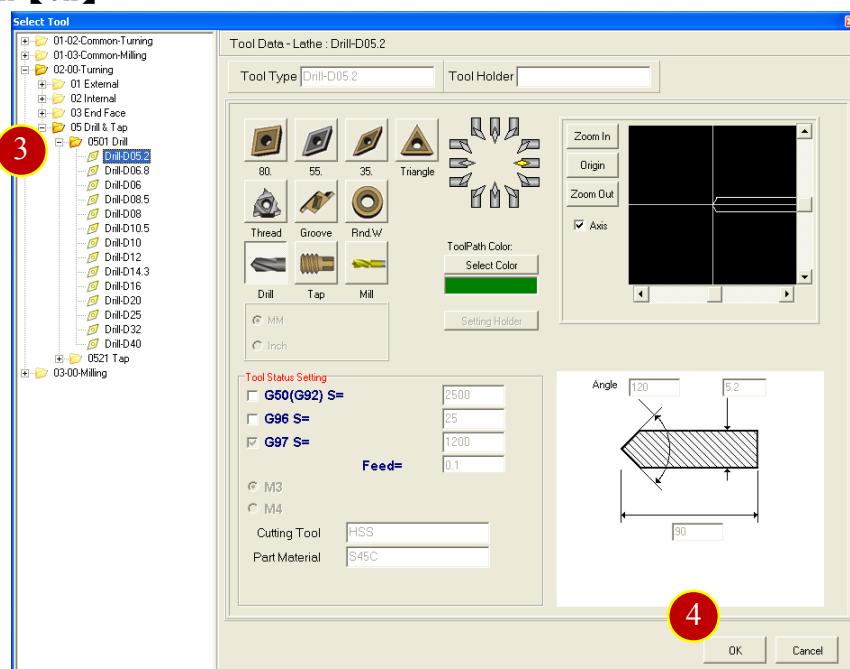


(2) Double click the blank in the cutting tool list

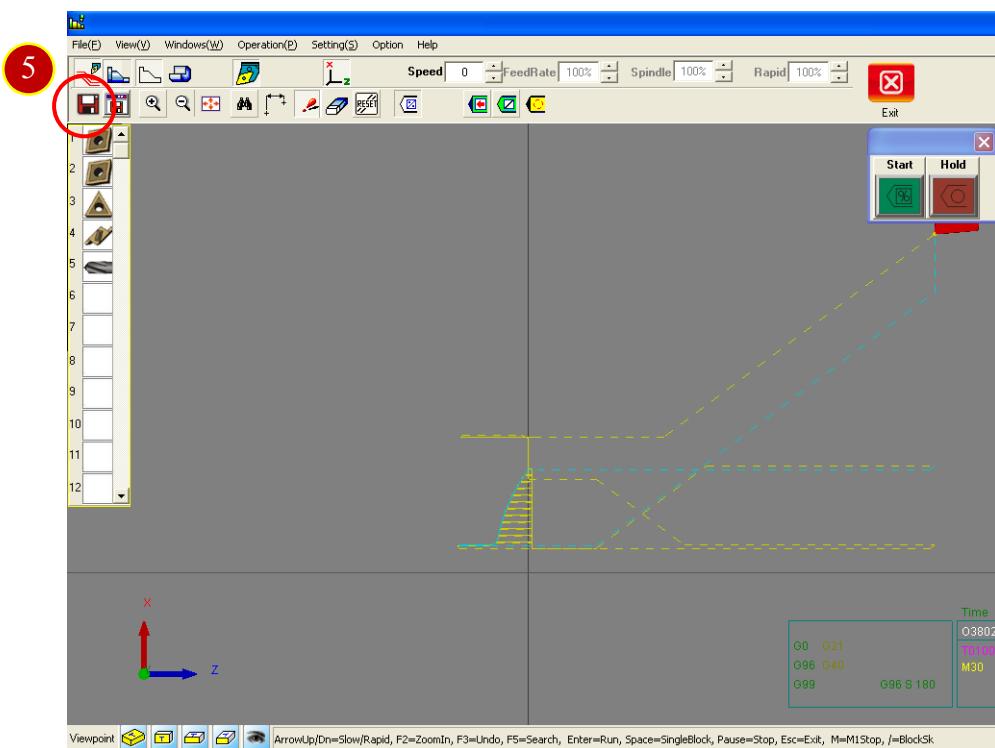


(3) Select the drill 【Drill-D05. 2】

(4) Click 【OK】



(5) Click 【Save】

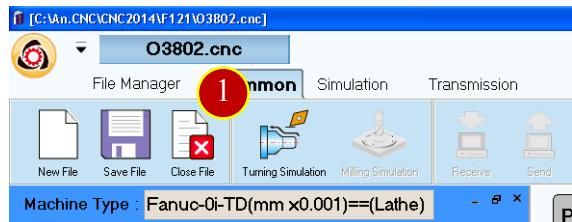


9.2.4 Turning Simulation - Material Setting

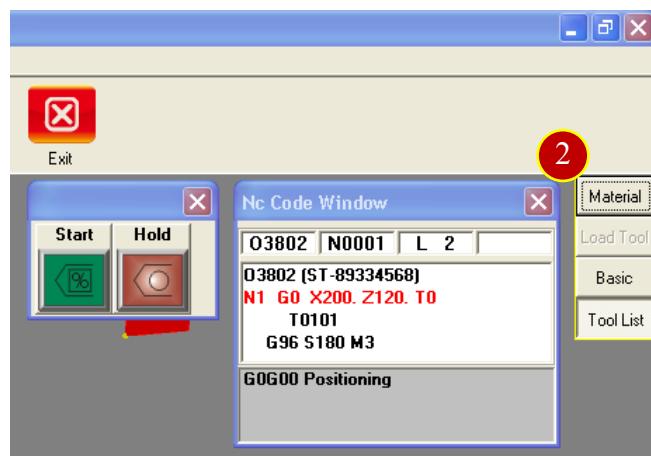
You can adjust or modify the material when you are in the simulation, follow the steps below,

e.g. when you are in the simulation of F121\03802. CNC, you can adjust the material D=85 L=60

(1) Open the program F121\03802. CNC, click 【Turning Simulation】

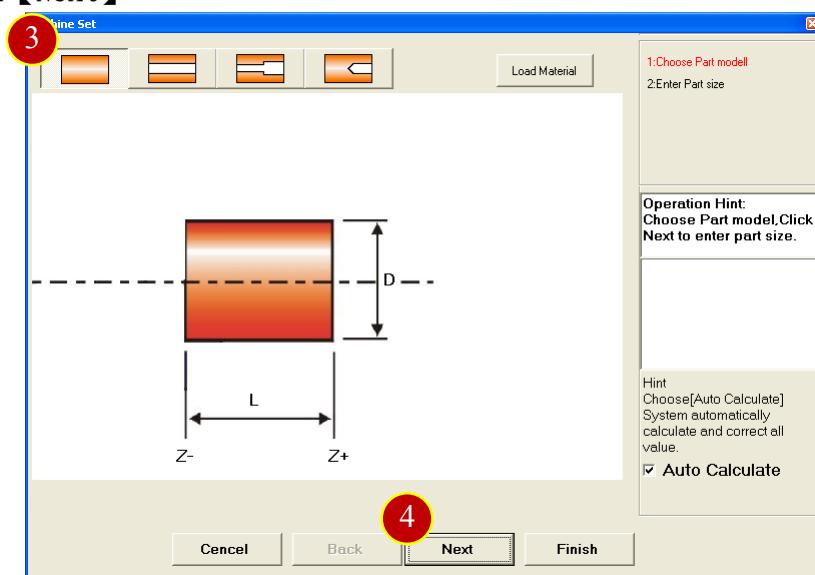


(2) Click 【Material】 in the right tool bar



(3) Select the material type

(4) Click 【Next】

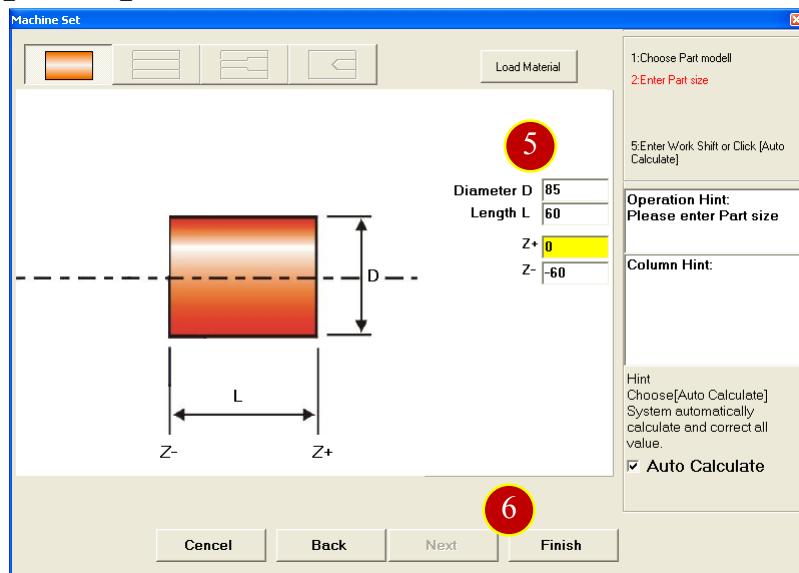


CNC Program Simulation (Turning and Milling)

(5) Key in 85 in diameter D blank and press [Enter]

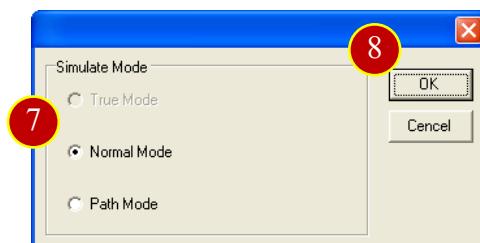
Key in 60 in length L blank and press [Enter]

(6) Click 【Finish】

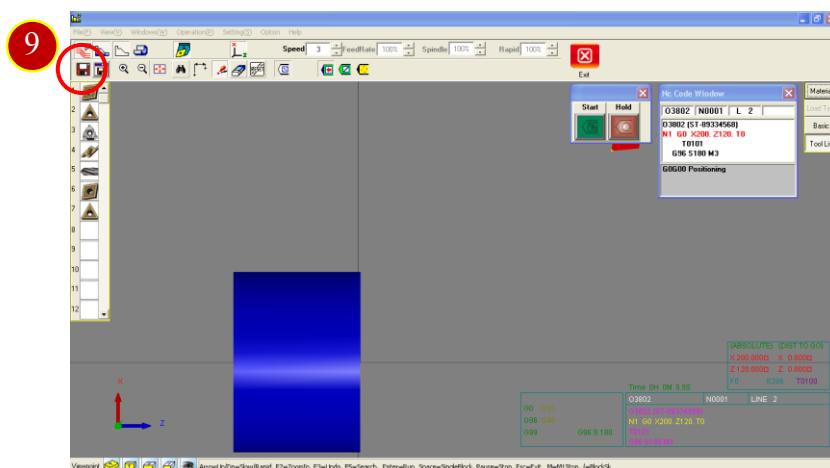


(7) Click 【Normal Mode】

(8) Click 【OK】



(9) Click 【Save】



9.3 Material Setting for Milling

To show the cutting process and the products precisely, you should first set Milling Material.

9.3.1 Material Setting

e.g. Modify the height of the material in F123\01012.CNC to 45

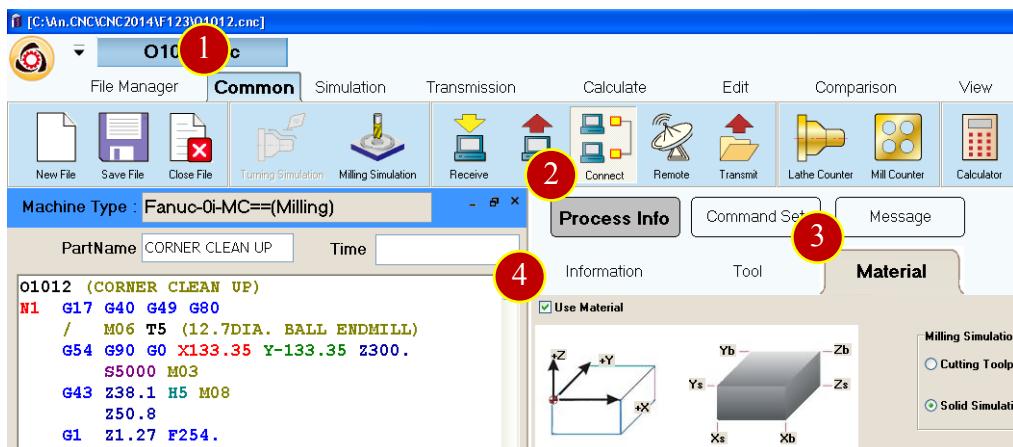
(1) Open the CNC file, click 【Common】

e.g. open F123\01012.CNC

(2) Click 【Process Info】

(3) Click 【Material】

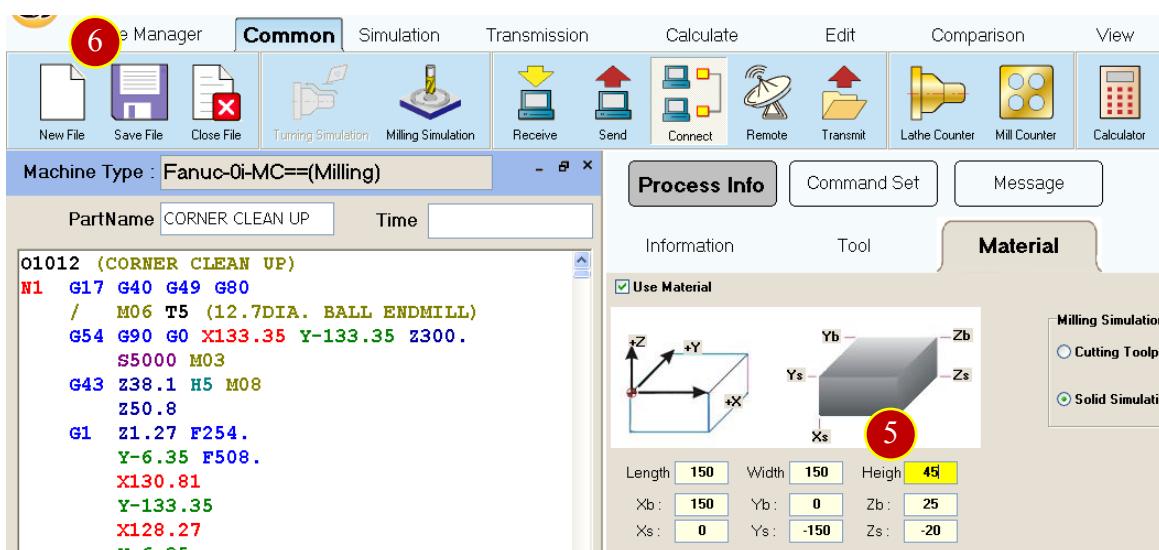
(4) Check 【Use Material】



(5) Click Height to key in data

e.g. 45

(6) Click 【Save File】



9.4 Simulation Operation and Settings for Milling

9.4.1 Execute "Milling Simulation" (Path)

Simulate the cutting path of CNC program

e.g. stimulate the milling path F123\07652.CNC

(1) After open the CNC file, click 【Common】

e.g. open F123\07652.CNC

(2) Click 【Process Info】

(3) Click 【Material】

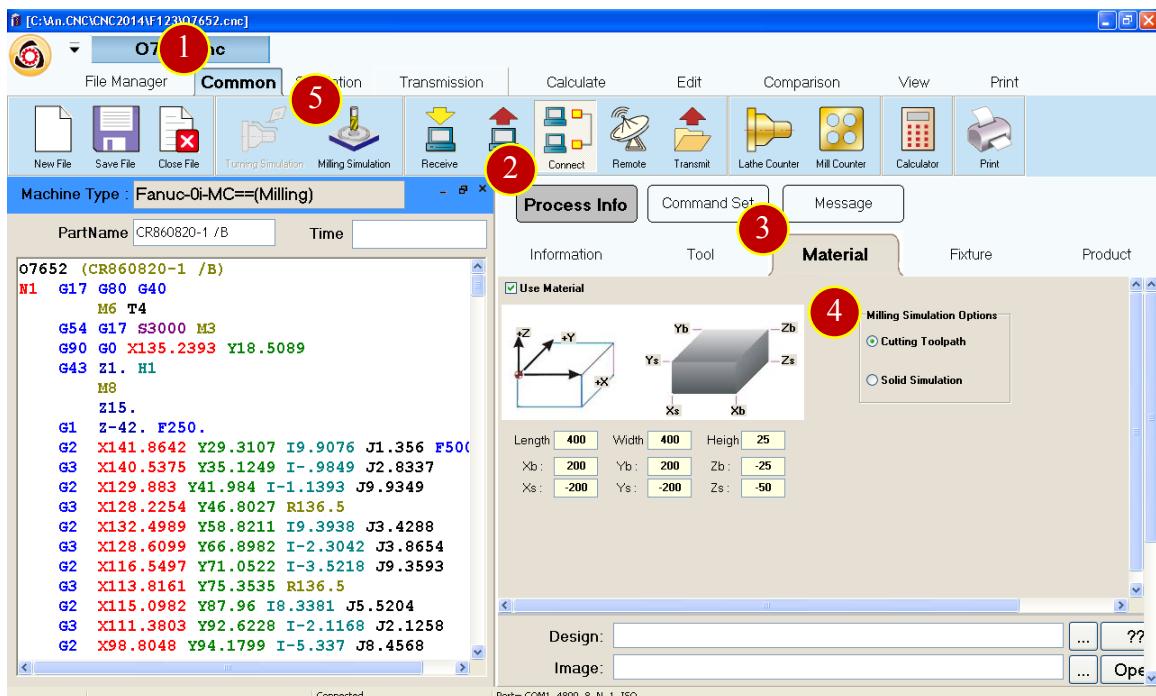
(4) Check 【Cutting Toolpath】

(5) Click 【Milling Simulation】

*If you cannot use it, please check the following items:

(a) Machine type set completely

(b) Machine type is Milling Machine



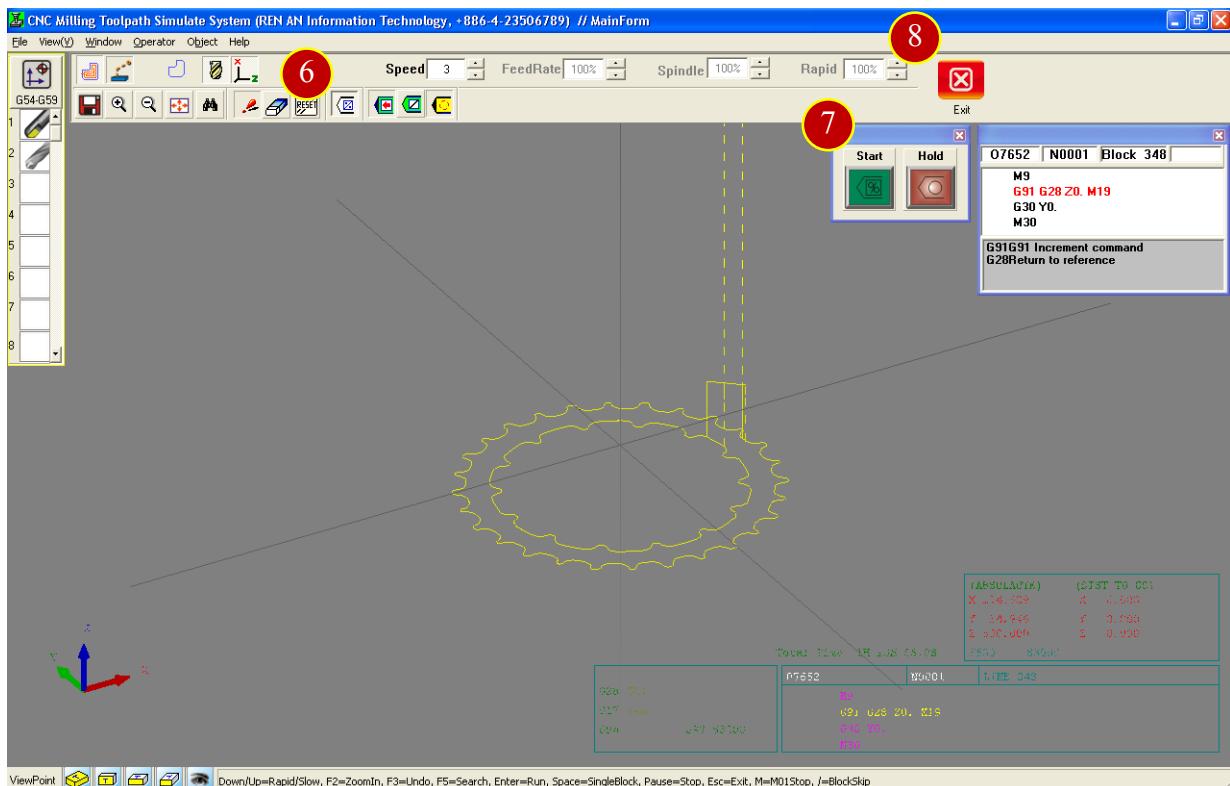
NcEditor MANUAL



RenAn Information Technology Co., Ltd.
www.RenAn.com.tw

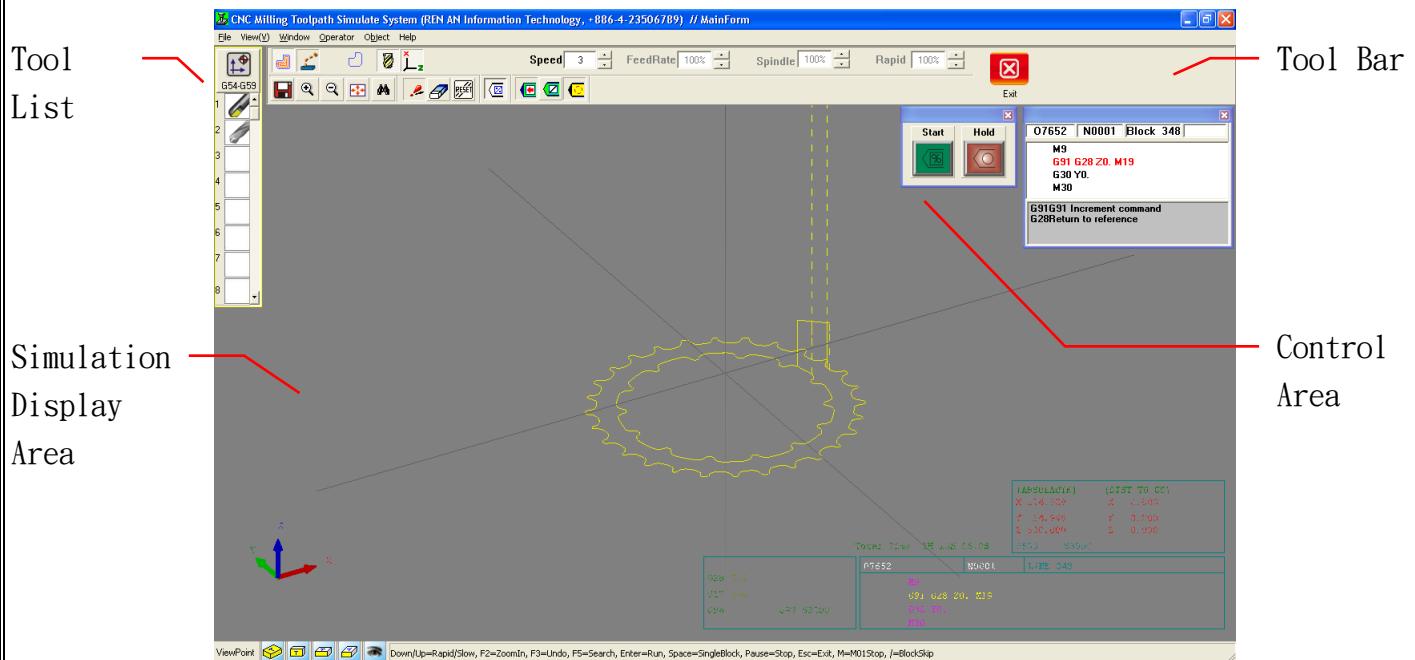
CNC Program Simulation (Turning and Milling)

- (6) Click 【Cycle Start】 to display control area
- (7) It will run automatically after open it
If not, click 【Start】
- (8) Click 【Exit】 to return to NcEditor



9.4.2 Operation of Milling Simulation (Path)

Instructions for the manipulation of Milling Simulation

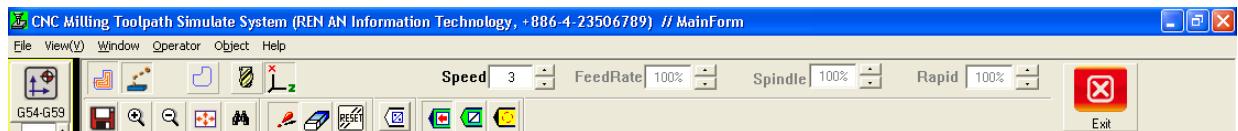


Simulation Display Area

| Mouse Operation | Instruction |
|---------------------------------|----------------------|
| Mouse Wheel, Scrolling Forward | Zoom out the pattern |
| Mouse Wheel, Scrolling Backward | Zoom in the pattern |
| Middle Click & Drag | Move the pattern |
| Click & Drag | Magnify the pattern |

Control Area

| Button | Name | instruction |
|---|-------|-------------------|
|  | Start | Start the program |
|  | Hold | Hold the program |



Tool bar

| Button | Name | Instruction |
|--------|-----------------|------------------------------------|
| | Path | Show or hide the path |
| | Rapid path | Show or hide the rapid path |
| | Tool | Show or hide the tool |
| | Axes | Show or hide the XYZ axes line |
| | Speed | Adjust the speed of simulation |
| | Feed Rate | Adjust the feed rate |
| | Spindle | Adjust the spindle override |
| | Rapid | Adjust the rapid rate |
| | Save | Save current program setting |
| | Zoom in | Zoom in the pattern |
| | Zoom out | Zoom out the pattern |
| | Initial view | Initial size of the pattern |
| | Search sequence | Search sequence in the CNC program |
| | Dimension | Show or hide the dimension panel |
| | Message switch | Turn on or off message |
| | Clean screen | Clean path in display area |
| | Reset | Reset the simulation |

CNC Program Simulation (Turning and Milling)

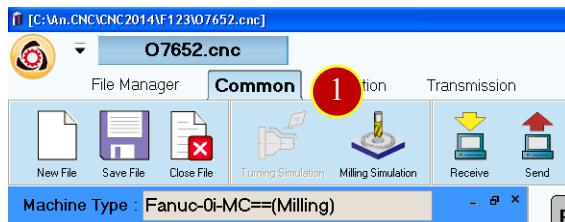
| | | |
|---|-------------|---|
|  | Cycle Start | Auto-run the program Open control area display 【Cycle Start】 & 【Hold】 button |
|  | S.B.K | Run single block of program and pause |
|  | B.D.T | Ignore program after the sign “/” in running |
|  | M01 | M01 pause state |
|  | Exit | Exit simulation |

9.4.3 Milling Simulation - Cutting Tool Setting

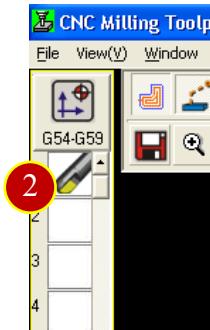
If you need to adjust, add modify the cutting tools during the simulation, follow the steps below.

e.g. when you are in the simulation of F123\07652. CNC, you can select the drill **【Drill-D03】**

(1) Open the program F123\07652.CNC, click **【Milling Simulation】**

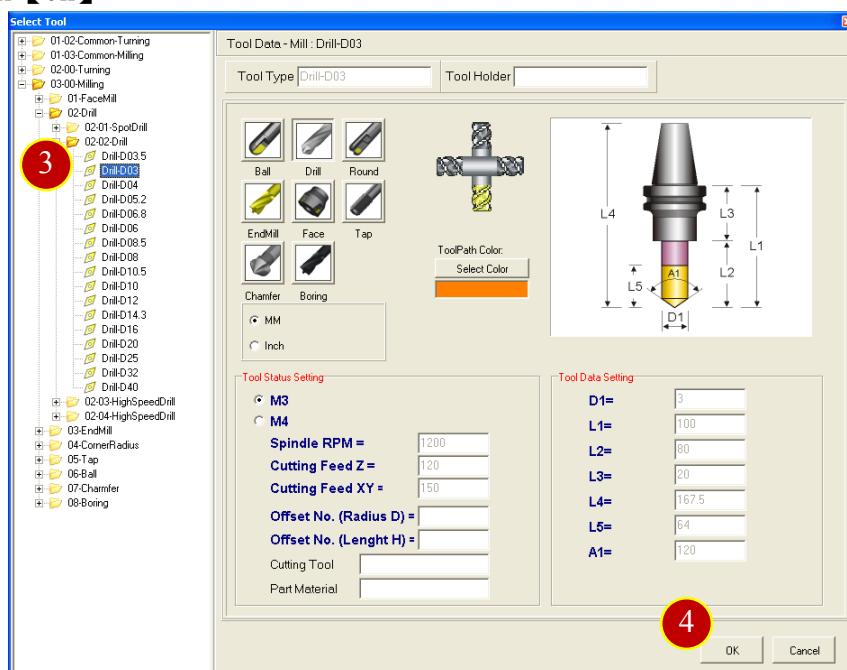


(2) Double click the blank in the tool list

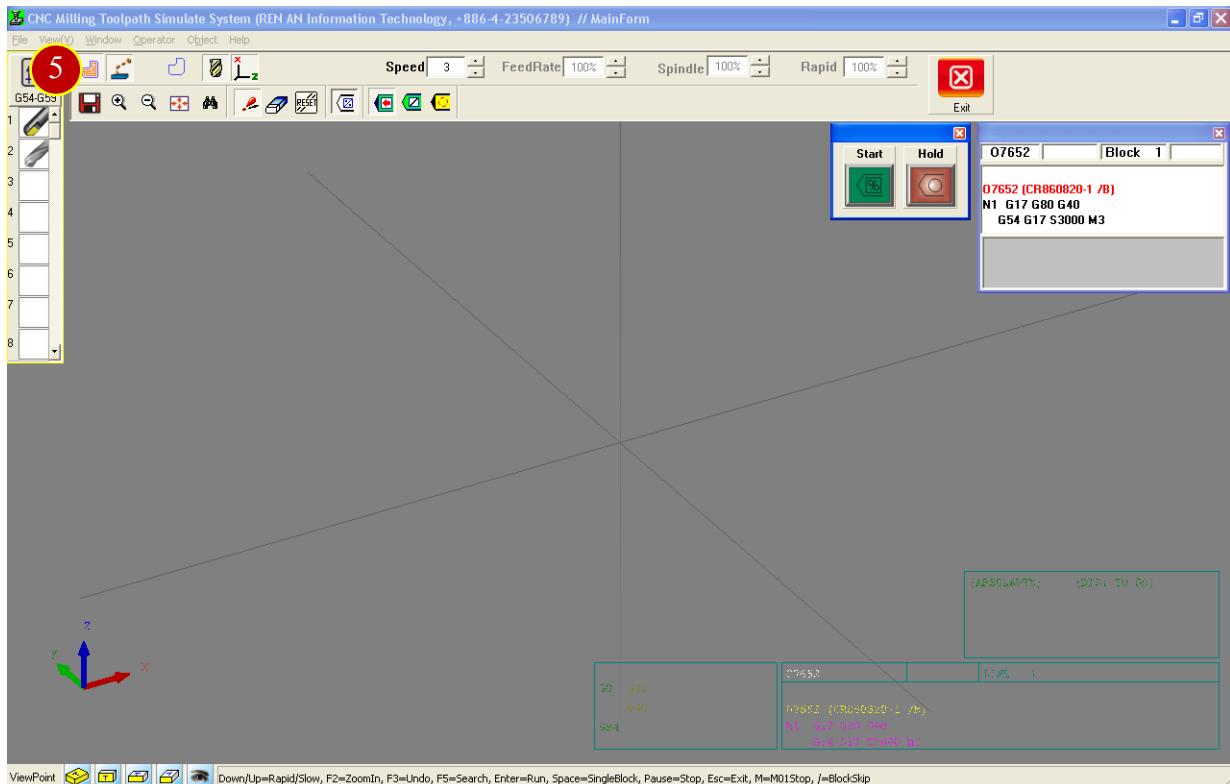


(3) Select the drill **【Drill-D03】**

(4) Click **【OK】**



(5) Click 【Save】



9.4.4 Open the Milling Simulation (Solid)

(1) Simulate the cutting process in reality

e.g. simulate the cutting process in F123\07652.CNC

(2) After open the CNC file, click 【Common】

e.g. open F123\07652.CNC

(3) Click 【Process Info】

(4) Click 【Material】

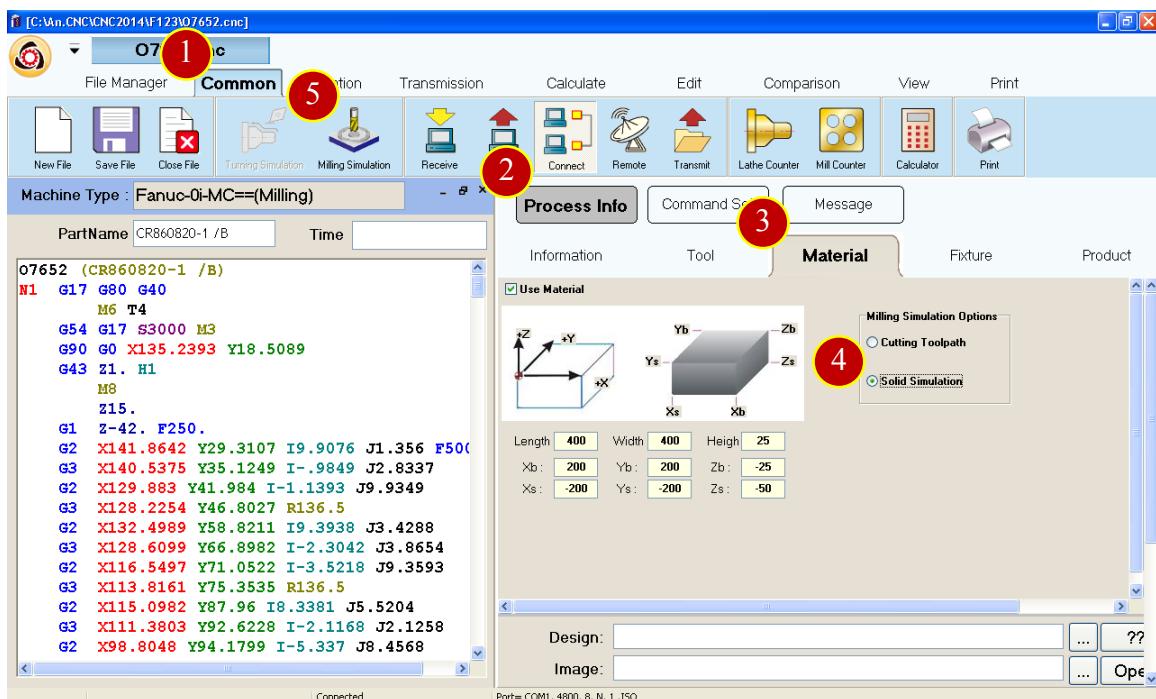
(5) Click 【Solid Simulation】

(6) Click 【Milling Simulation】

* If you cannot use, please check the items below:

(a) Machine type set completely

(b) Machine type is Milling Machine



NcEditor MANUAL

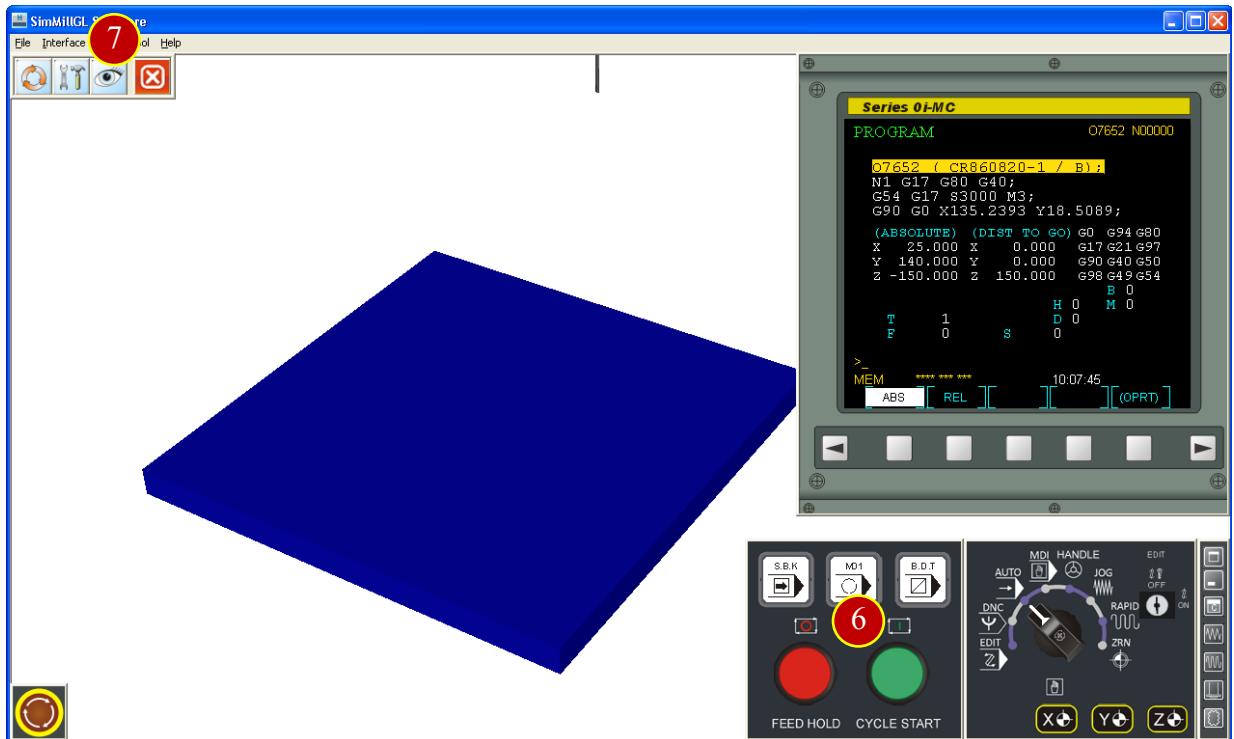


RenAn Information Technology Co., Ltd.
www.RenAn.com.tw

CNC Program Simulation (Turning and Milling)

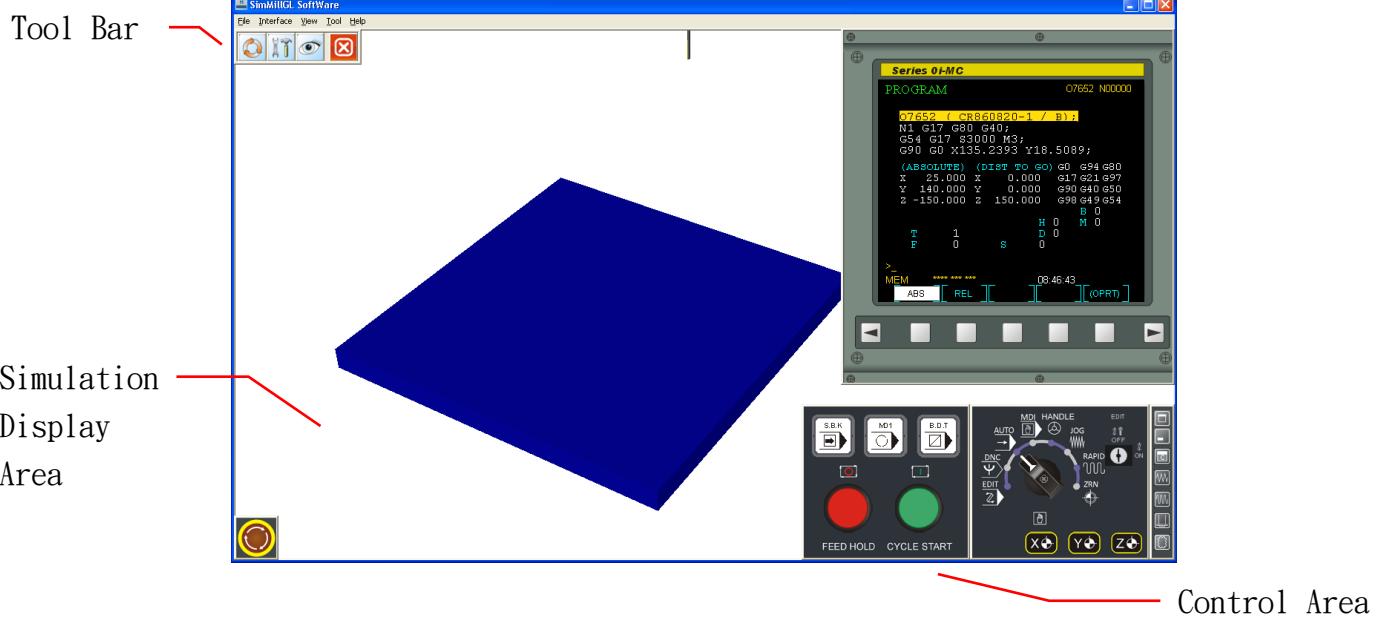
(7) Click 【Cycle Start】

(8) Click 【Exit】 to return to NcEditor



9.4.5 Milling Simulation (Solid)

Instructions for Milling Simulation



Simulation Display Area

| Mouse Operation | Instruction |
|---------------------------------|----------------------|
| Mouse Wheel, Scrolling Forward | Zoom out the pattern |
| Mouse Wheel, Scrolling Backward | Zoom in the pattern |
| Middle Click & Drag | Move the pattern |
| Click & Drag | Rotate the pattern |

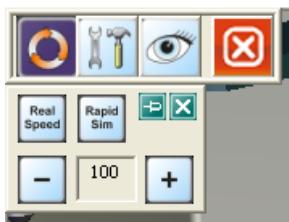
Control Area

| Button | Name | Instruction |
|--------|-------------|--|
| | S.B.K | Run single block of program and pause |
| | M01 | M01 pause state |
| | B.D.T | Ignore program after the sign “/” in running |
| | Feed Hold | Stop the program |
| | Cycle Start | Start the program |



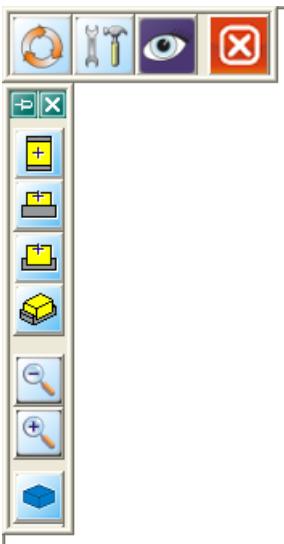
Tool List

| Button | Name | Instruction |
|--------|---------------|--------------------------------|
| | Speed Control | Show or hide the speed control |
| | Function | Show or hide the function bar |
| | View Control | Show or hide the Control |
| | Exit | Close Milling Simulation |



Speed Control

| Button | Name | Instructions |
|--------|-------------|--|
| | Real Speed | Simulate real speed |
| | Rapid Speed | Simulate rapid speed |
| | Speed up | Rapid speed up |
| | Speed | To show the current rapid speed, the fastest:1 the slowest:500 |
| | Slow down | Slow down the rapid speed |



View Control

| Button | Name | Instructions |
|---|---------------|-------------------------------|
|  | Top view | View changes to top view |
|  | Front view | View changes to front view |
|  | Side view | View changes to side view |
|  | ISD view | View changes to ISD view |
|  | Zoom out | Zoom out the pattern |
|  | Zoom in | Zoom in the pattern |
|  | Material view | View changes to material view |

10. NC Files Transmissions

After connect to CNC machine, you can transmit CNC file to make the back-up file.

10.1 Send Program - From PC to CNC

Send the program from NcEditor to CNC machine

10.1.1 Sending program - PC Operation

e.g. Send the file F121\03802.CNC to CNC machine

(1) Open the program and click 【Transmission】

e.g. open F121\03802.CNC

(2) Click 【Connect】

*If you cannot connect, please check the items below

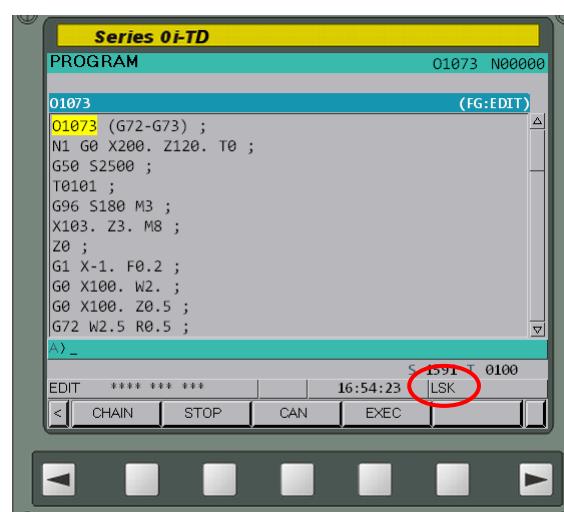
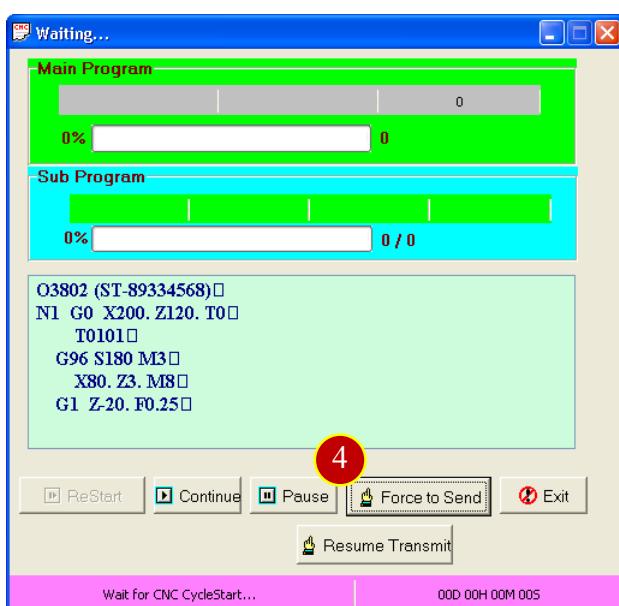
(a) Use common transmission mode, cancel remote mode

(b) Check transmission setting and please read CH10. 6 - Protocol Settings - Basic

(3) Click 【Send】

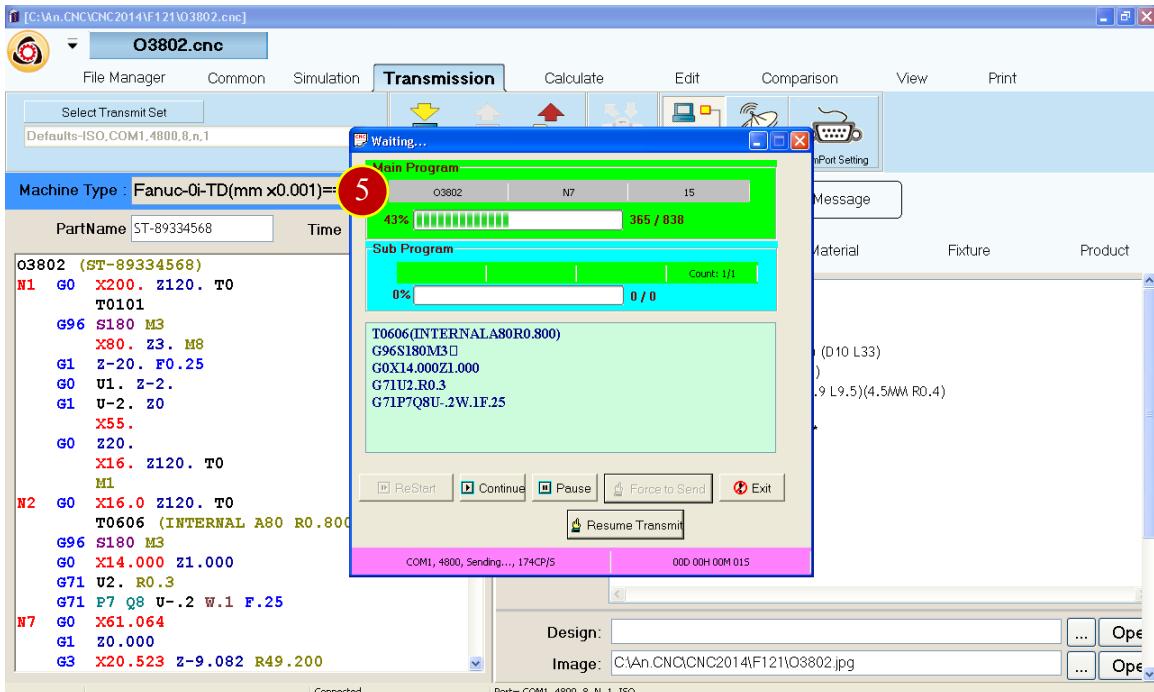


(4) Waiting for CNC receive of the program, please read CH10. 1. 2 or you can click 【Force to send】

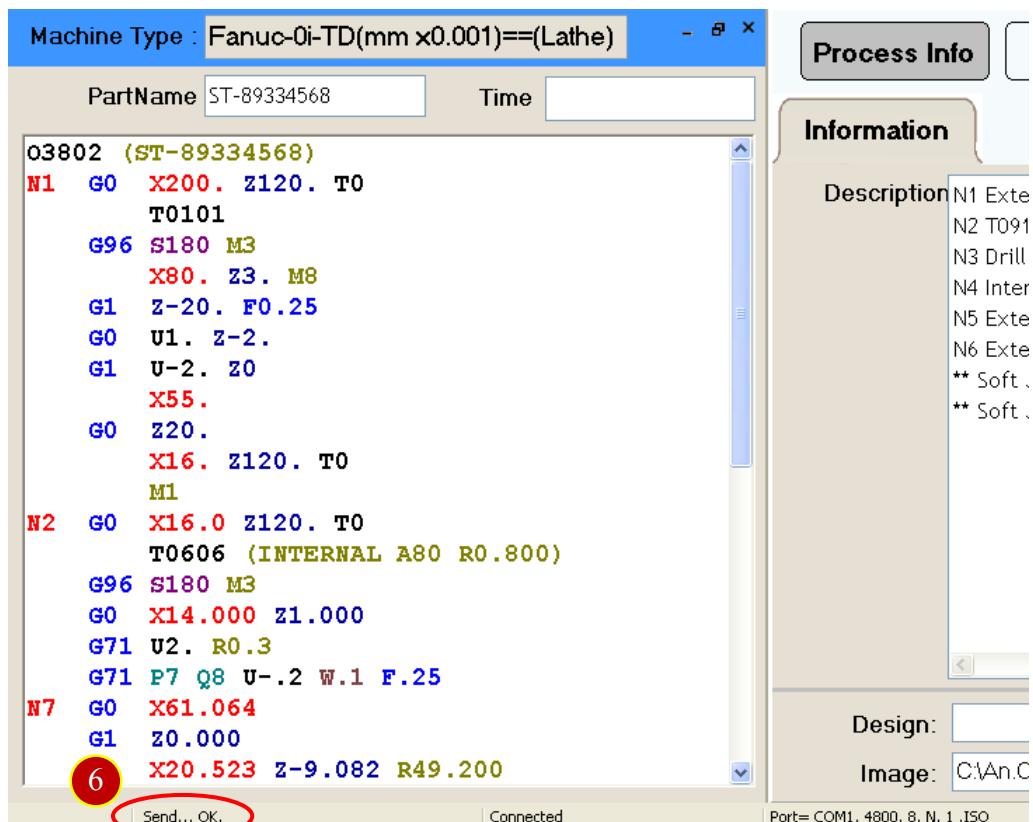


NcEditor MANUAL

(5) When it is sending, the window will display you the current status



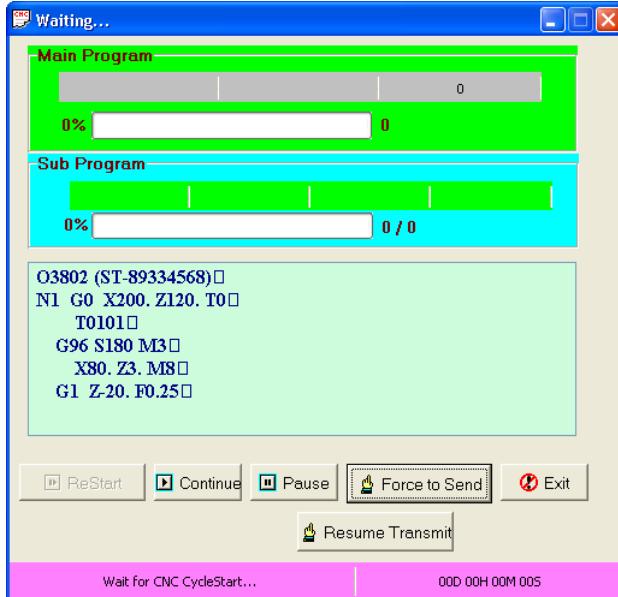
(6) When complete, the status displays 【Send...OK.】



10.1.2 Receiving Program - Operation of CNC machine

This is the operation according to CNC machine type.

Before, you have to open the transmit window in NcEditor.



Take Fanuc 18T, 18M, 21T, 0i-T, 0i-M for example (you can read other types in CH 錯誤! 找不到參照來源)。

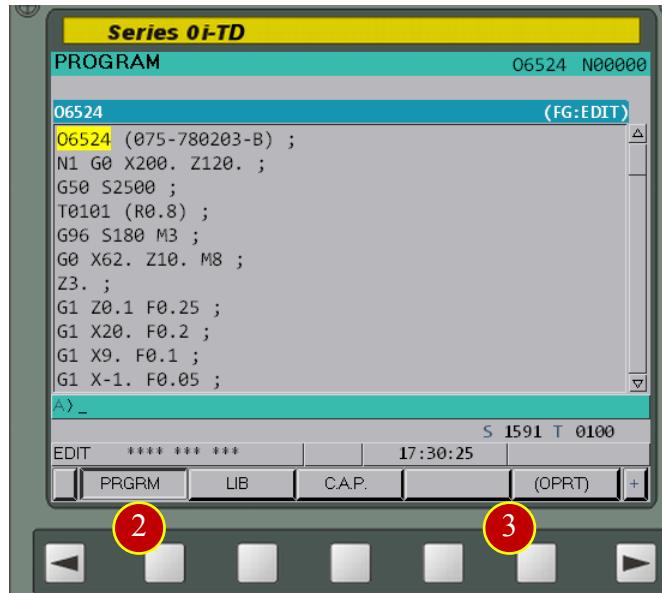
e.g. Receive 03802.CNC from PC

(1) Change to 【EDIT】 mode



(2) Press 【PROGRAM】

(3) Press 【(OPRT)】



(4) Press 【>】 until it displays 【READ】【PUNCH】

(5) Press 【READ】



(6) Press 【EXEC】 to receive the programs and information from PC.

When the program is waiting for receiving, it displays 【LSK】 on the screen.



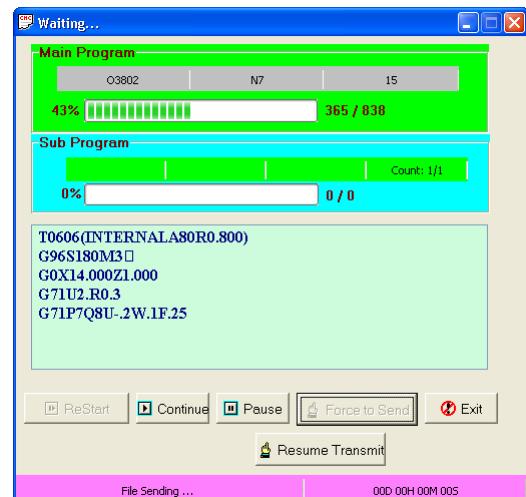
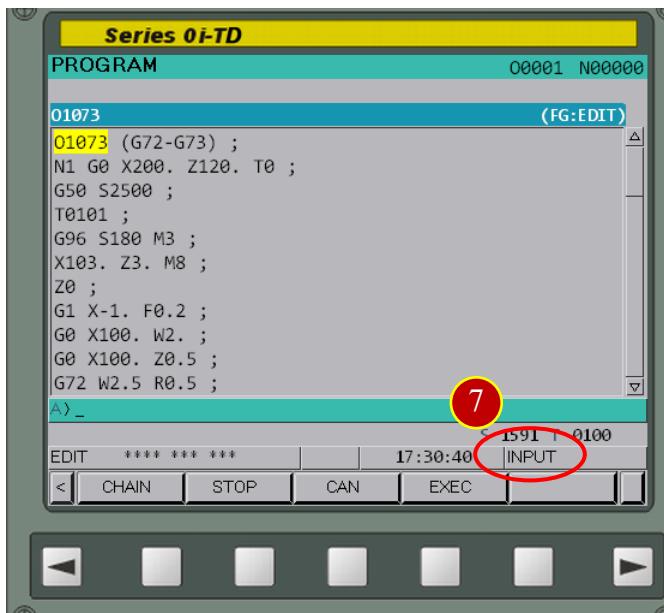
NcEditor MANUAL



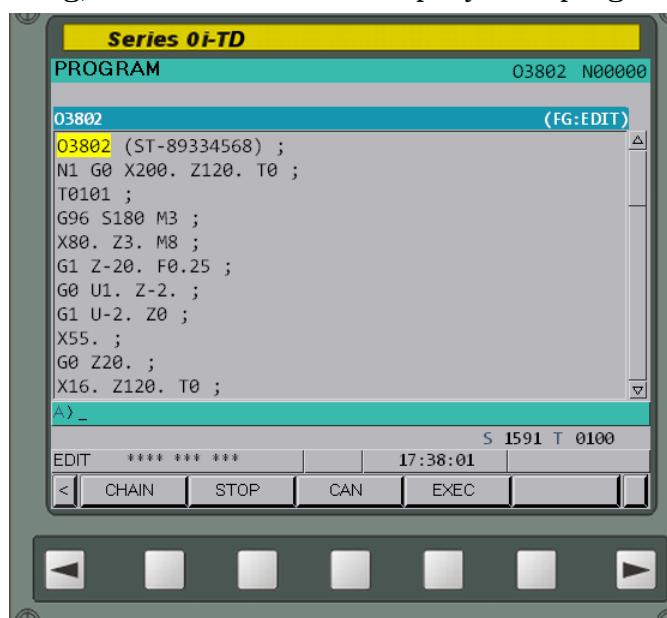
RenAn Information Technology Co., Ltd.
www.RenAn.com.tw

NC Files Transmissions

(7) When the program is being received, it displays 【INPUT】



(8) After receiving, controller will display the program



10.1.3 Program Receiving - CNC Machine Operation List

The follow machine types will be manipulated in the demonstration

(1) FANUC 18T、18M、21T、0i-T、0i-M

e.g. Receive 03802.CNC from PC

| Step | Buttons | Instructions |
|------|---------------------|---|
| 1 | 【EDIT】 | Change to 【EDIT】 |
| 2 | 【PROGRAM】 | Press 【PROGRAM】 to display the program |
| 3 | 【(OPRT)】 【>】 | Press 【(OPRT)】 Press 【>】 until it displays 【READ】【PUNCH】 |
| 4 | 【READ】 | Press 【READ】 |
| 5 | 03802 | <21T, 18M can ignore this step> When you key in the program, you can ignore the program num. |
| 6 | 【EXEC】 or 【STOP】 | Press 【EXEC】 to receive Press 【STOP】 to stop |

(2) FANUC OT、OM

e.g. Receive 03802.CNC from PC

| Step | Buttons | Instructions |
|------|-----------|--|
| 1 | 【EDIT】 | Change to 【EDIT】 |
| 2 | 【PROGRAM】 | Press 【PROGRAM】 to display the program |
| 3 | O3802 | When you key in the program, you can ignore the program num. (Necessity, key in program num e.g.O3802) |
| 4 | 【INPUT】 | Press 【INPUT】 |
| 5 | | When (LSK) show on the screen means wait for input (INPUT) flashing means it is input |

(3) MITSUBISHI

e. g. Receive 03802.CNC from PC

| Step | Buttons | Instructions |
|------|----------------|---|
| 1 | 【Off】 | Program Protection Key 【Off】 |
| 2 | 【IN/OUT】 | Press 【IN/OUT】 at the right side |
| 3 | 【MENU】 | Press 【MENU】 to display 【INPUT/OUTPUT】 |
| 4 | 【INPUT】 | Press 【INPUT】 |
| 5 | #(1) (3802) | Press #(1)DATA(3802) (When you key in the program, you can ignore the program num.) ※ #(1) means input/output CNC program DATA(3802) means program O3802 |
| 6 | 【INPUT】 | Press 【OUTPUT】 |

(4) OKUMA U100L

e. g. Receive 03802.CNC from PC

| Step | Buttons | Instructions |
|------|------------|--|
| 1 | 【F3】【PIP】 | Press 【F3】【PIP】 |
| 2 | 【F1】【READ】 | Press 【F1】【READ】 |
| 3 | 【WRITE】 | Press 【WRITE】 |
| 4 | | After machine's read, it generates file name A.MIN If "file exists, load ok?" displays, press 【Y】 |

(5) Brother TC-S2A

e. g. Receive 03802.CNC from PC

| Step | Buttons | Instructions |
|------|------------------------|--|
| 1 | 【EXTERNAL PROGRAM I/O】 | Function 【EXTERNAL PROGRM I/O】 (Input/ output and send exterior program) |
| 2 | 【INPUT FROM PTR】 | Function 【INPUT FROM PTR】 (Input from PTR) |
| 3 | 【PROGRAM】 【E.STA】 | Function 【PROGRAM】 , PROGRAM _ flashing Press 【E. STA】 directly without key in code |
| 4 | | If the program exists, it displays "Already exist same program NO, Load OK?" Be sure and press 【F0】(YES) |
| 5 | | Chose file editing and check the code |

10.1.4 Direct Sending - PC Operation

e.g. Send file F123\01012.CNC to CNC machine

(1) Click 【Transmission】

(2) Click 【Connect】 to connect with CNC machine

* If the connection is failed, please check items below:

(a) Use the common transmission mode and cancel the remote mode

(b) Check transmission setting, please read CH 錯誤! 找不到參照來源。- 錯

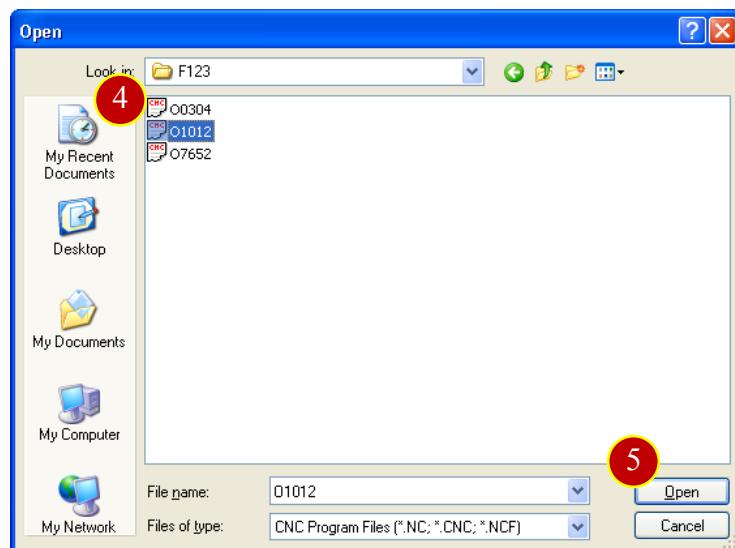
誤! 找不到參照來源。

(3) Click 【Transmit】

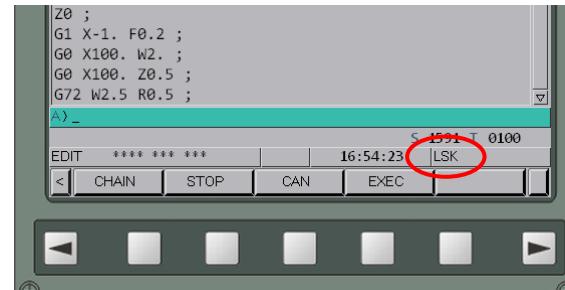


(4) Select file F123\01012.CNC

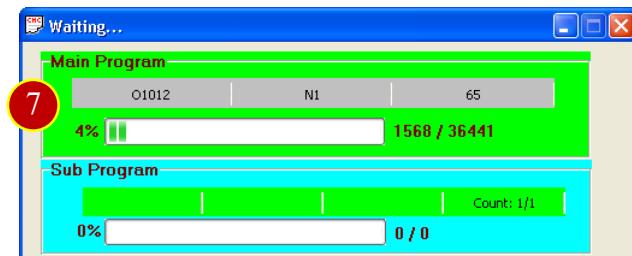
(5) Click 【Open】



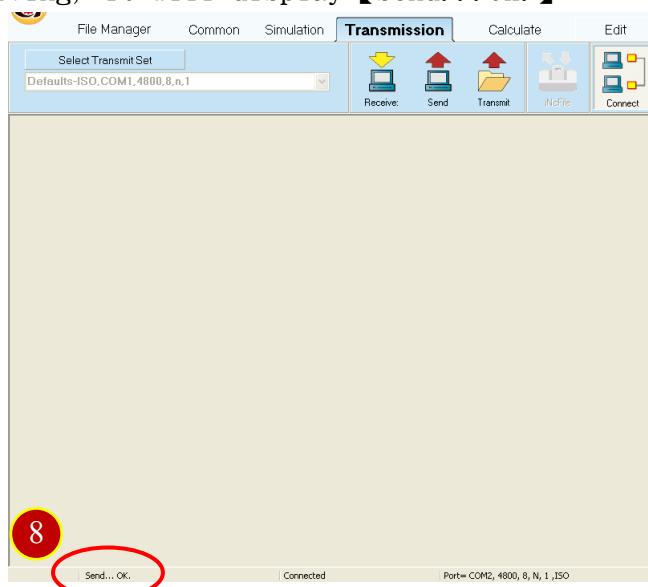
(6) Waiting for CNC receive of the program, please read CH10.1.2
or you can click 【Force to Send】



(7) When it is sending, the window will display you the current status



(8) After receiving, it will display 【Send...OK.】。



10.2 Receive Program - From CNC to PC

Receive NcEditor program from CNC

10.2.1 Receiving Program - PC Operation

e.g. Receive the program 06523.CNC from CNC machine

(1) Click 【Transmission】

(2) Click 【Connect】 to connect with CNC machine

* If the connection is failed, please check items below:

(a) Use the common transmission mode and cancel the remote mode

(b) Check transmission setting, please read CH 錯誤! 找不到參照來源。- 錯

誤! 找不到參照來源。

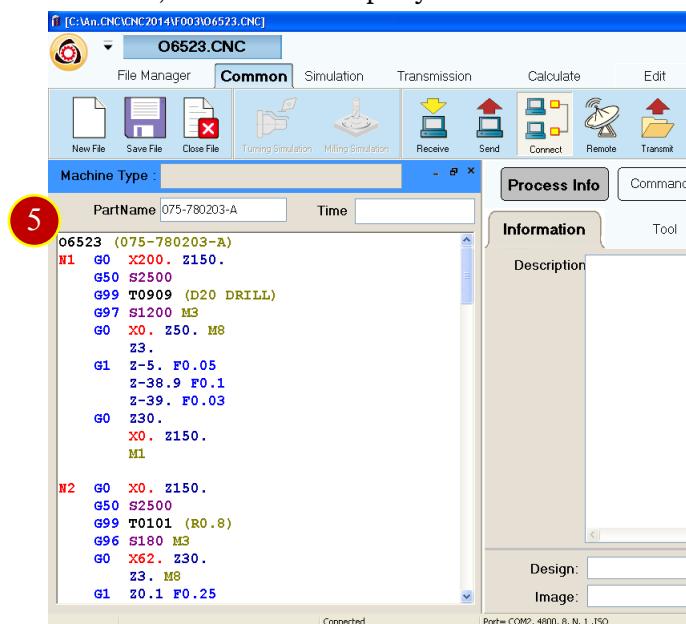
(3) Click 【Receive】



(4) When it is receiving, it will display the received program



(5) After transmission, it will display the context of 06523.CNC



10.2.2 Program Transmission - CNC Machine Operation

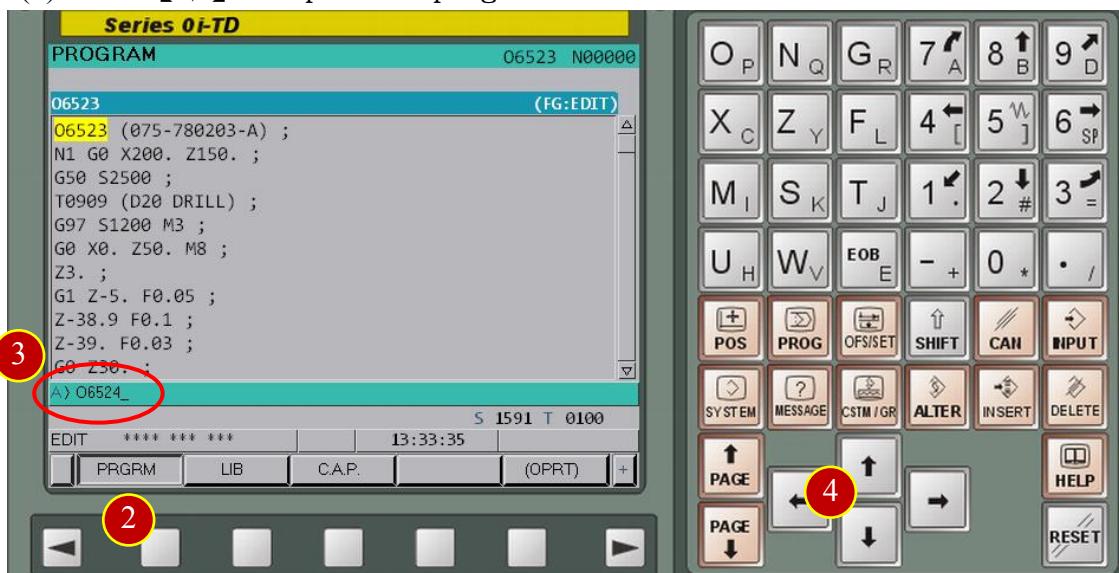
Take Fanuc 0i-T, 0i-M as an example (you can read other types in CH 錯誤!
找不到參照來源。)

(1) Change to 【EDIT】



(2) Press 【PROGRAM】

- (3) Key in [06524] as a program number
- (4) Press 【↓】 to open the program



(5) Press 【(OPRT)】



(6) Press 【>】 until it displays 【READ】【PUNCH】

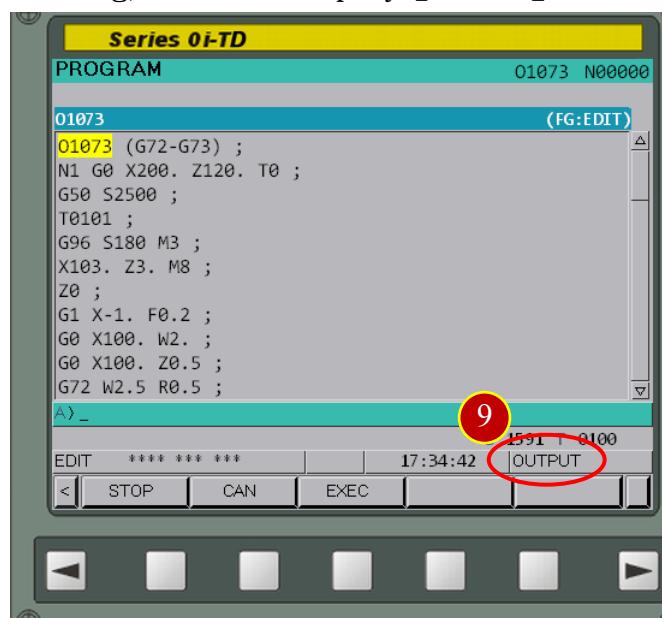
(7) Press 【PUNCH】



(8) Press 【EXEC】 to save from CNC program to PC



(9) When it is sending, it will display 【OUTPUT】



10.2.3 Program Transmission – CNC Machine Operation

The following machine types will be operated in the demonstration.

(1) FANUC 0i-M、0i-T

e. g. Send 06524.CNC to PC

| Step | Buttons | Instructions |
|------|-----------------|---|
| 1 | 【EDIT】 | Change to mode 【EDIT】 |
| 2 | 【PROGRAM】 | Press 【PROGRAM】 to display the program |
| 3 | O6524 | Key in program number e.g. O6524 |
| 4 | 【↓】 | Press 【↓】 to open O6524 Window displays program number and its content |
| 5 | 【(OPRT)】 【>】 | Press 【(OPRT)】 at the right corner Press 【>】 until it displays 【READ】【PUNCH】 |
| 6 | 【PUNCH】 | Press 【PUNCH】 |
| 7 | 【EXEC】 | Press 【EXEC】 to send program from CNC to PC Or press 【Stop】 |

(2) FANUC 18T、18M、21T

e. g. send 06524.CNC to PC

| Step | Buttons | Instructions |
|------|-----------------|---|
| 1 | 【EDIT】 | Change to mode 【EDIT】 |
| 2 | 【PROGRAM】 | Press 【PROGRAM】 to display the program |
| 3 | 【(OPRT)】 【>】 | Press 【(OPRT)】 at the right corner Press 【>】 until it displays 【READ】【PUNCH】 |
| 4 | 【PUNCH】 | Press 【PUNCH】 |
| 5 | O6524 | Key in program number e.g. O6524 |
| 6 | 【EXEC】 | Press 【EXEC】 to send program from CNC to PC Or press 【STOP】 |

(3) FANUC OT、OM

e.g. send O6524.CNC to PC

| Step | Buttons | Instructions |
|------|-----------|---|
| 1 | 【EDIT】 | Change to mode 【EDIT】 |
| 2 | 【PROGRAM】 | Press 【PROGRAM】 to display the program |
| 3 | O6524 | Key in program number e.g. O6524 |
| 4 | 【OUTPUT】 | Press 【OUTPUT】 |
| 5 | | When (OUTPUT) flashing in window means it is output |

(4) MITSUBISHI

e.g. send O6524.CNC to PC

| Step | Buttons | Instructions |
|------|----------------|---|
| 1 | 【Off】 | Program Protection Key |
| 2 | 【IN/OUT】 | Press 【IN/OUT】 at the right side |
| 3 | 【MENU】 | Press 【MENU】 until it displays 【INPUT/OUTPUT】 |
| 4 | 【ONPUT】 | Press 【OUTPUT】 at the bottom |
| 5 | #(1) (6524) | Press #(1) DATA(6524) ※ #1 means the program DATA(6524) means the program O6524 |
| 6 | 【INPUT】 | Press 【INPUT】 at the right side |

(5) OKUMA U100L

e.g. send O6524.CNC to PC

| Step | Buttons | Instructions |
|------|------------------------------|---|
| 1 | 【F3】【PIP】 | Press 【F3】【PIP】 |
| 2 | 【F2】【PUNCH】 (MD1 : INDEX) | Press 【F2】【PUNCH】 Press (MD1:INDEX) |
| 3 | Move the curser 【WRITE】 | Move your curser to the program number e.g. O6524.MIN Press 【WRITE】 |
| 4 | 【WRITE】 | Press 【WRITE】 |

(6) Brother TC-S2A

e. g. send 06524.CNC to PC

| Step | Buttons | Instructions |
|------|--|---|
| 1 | O6524 (DEMO TEST) N1 G50 S2000 F123 G96 S200 M3 T0101 M8 | Functions PROGRAM EDIT, number 6524 Content: Edit program(demo) Folder number F123 Key in folder number F123 in the second line of the program Command 06524 to save in the file F123 |
| 2 | 【EXTERNAL PROGRAM I/O】 | Function 【EXTERNAL, PROGRAM I/O】 |
| 3 | 【OUTPUT TO PTP】 【PROGRAM】 | Function 【OUTPUT TO PTP】 Function 【PROGRAM】 |
| 4 | 6524 【E.STA】 | Key in the program number 6524 at the PROGRAM_ flashing spot Press 【E.STA】 |
| 5 | 【F0】 【F0】 | When it complete sending, double press 【F0】 (Return Menu) |

10.3 DNC Mode

Use DNC mode in CNC Machine and send CNC program from the PC. So, the machine will cutting after the receive of the program.

10.3.1 DNC mode - PC Operation

e.g. Use DNC mode to execute the file F121\03802.CNC

(1) Open the CNC program and click 【Transmission】

(2) Click 【Connect】 to connect with CNC machine

* Please check the items below if the connection is not available

(a) Use the common transmission mode and cancel the remote one

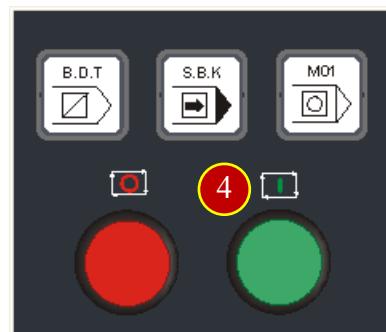
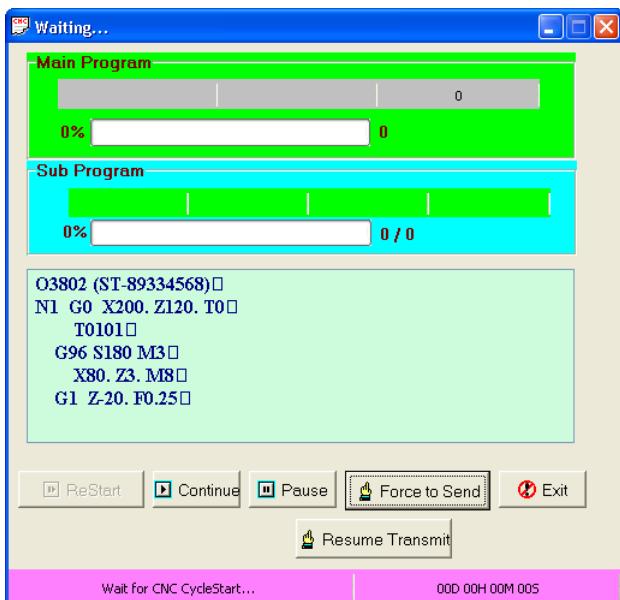
(b) Check the transmission setting in DNC and please read CH 錯誤! 找不到

參照來源。- 錯誤! 找不到參照來源。

(3) Click 【Send】

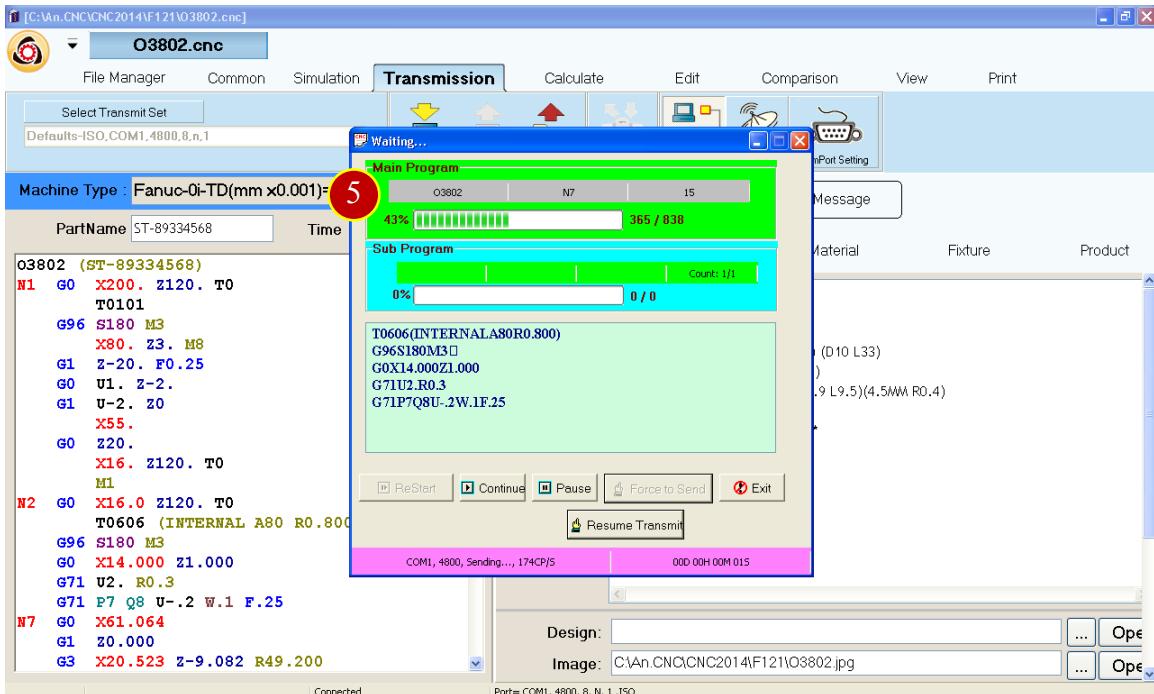


(4) Press 【CYCLE START】 to start



NcEditor MANUAL

(5) It will show the current progress when it is sending



10.3.2 DNC Mode - Machine Operation

This is a demonstration of DNC mode in CNC Machine

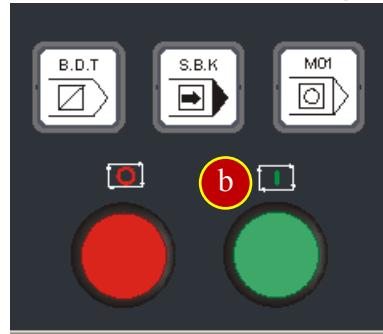
(1) FANUC 0i-M、0i-T

| Step | Buttons | Instructions |
|------|---------------|--|
| 1 | 【DNC】 | Change to 【DNC】 mode (or 【TAPE】 mode) |
| 2 | 【CYCLE START】 | Press 【CYCLE START】 to receive |

(a) Change to 【DNC】 mode



(b) Press 【CYCLE START】 to start the program



(c) The controller will show the code of CNC program and execute it



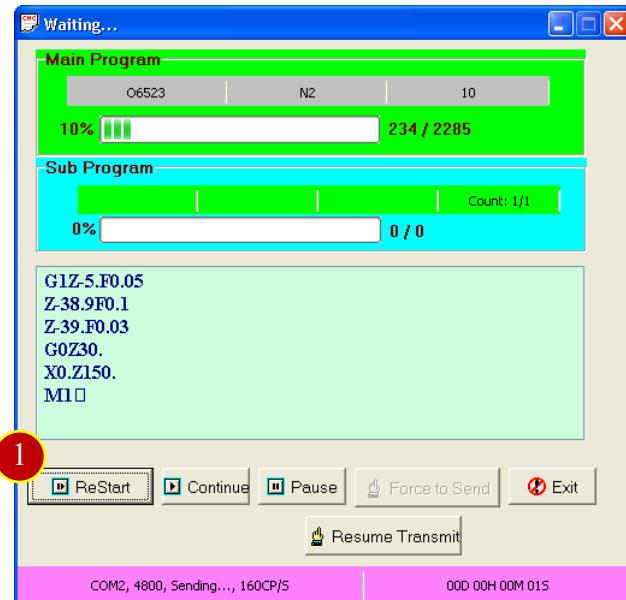
10.4 DNC Mode - Restart

When DNC is on, sending will be ceased if the tool breaks or the machine caused.

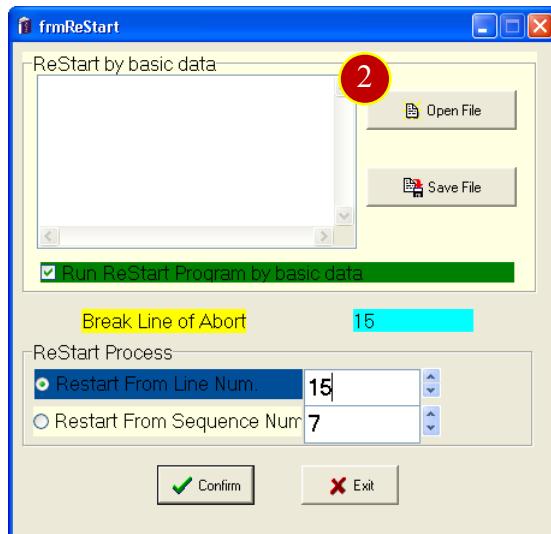
10.4.1 Tool Break and Restart

(1) When the sending is ceased, the button 【Reset】will show up.

Click 【Reset】and it will jump out a new window. Then, start the program
※DNC operation – please read CH10.3 – DNC Mode



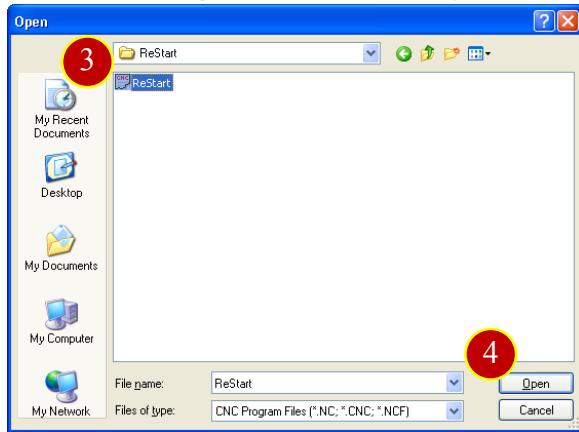
(2) Click 【Open File】to chose the file program that is going to be inserted



(3) Select the program to start basic data

e.g. ReStart.cnc

(4) Click 【Open】and the program will display in 【Restart by basic data】

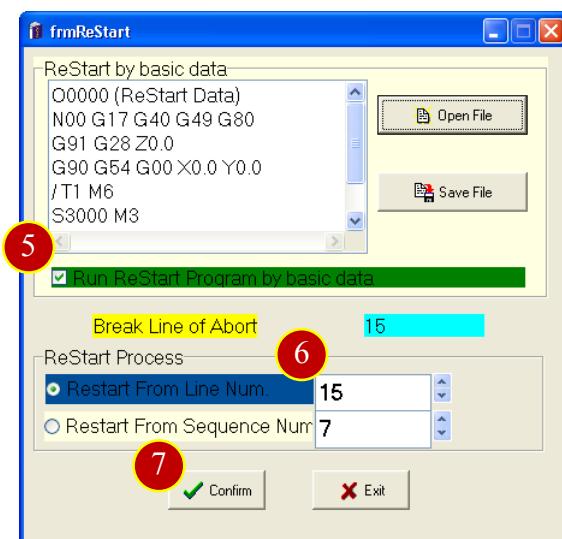


(5) Check 【Restart Program by basic data】

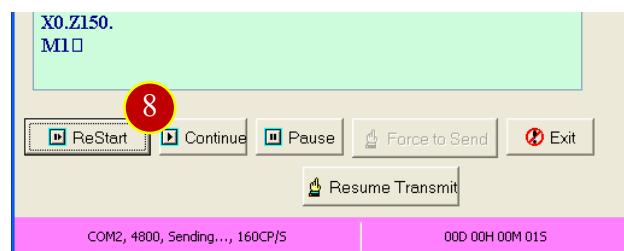
(6) Set the line number of the restart of the program

e.g. 15

(7) Click 【OK】



(8) Click 【Continue】to send the program to start basic data and send the main program from the eleventh line



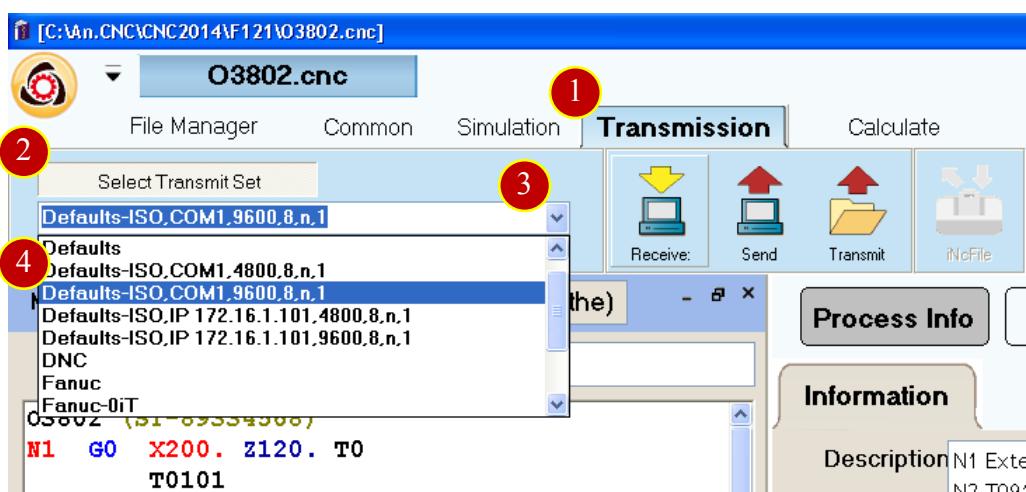
10.5 Common Transmission Term

Quickly switch to different machine transmission, preset the common transmission protocol

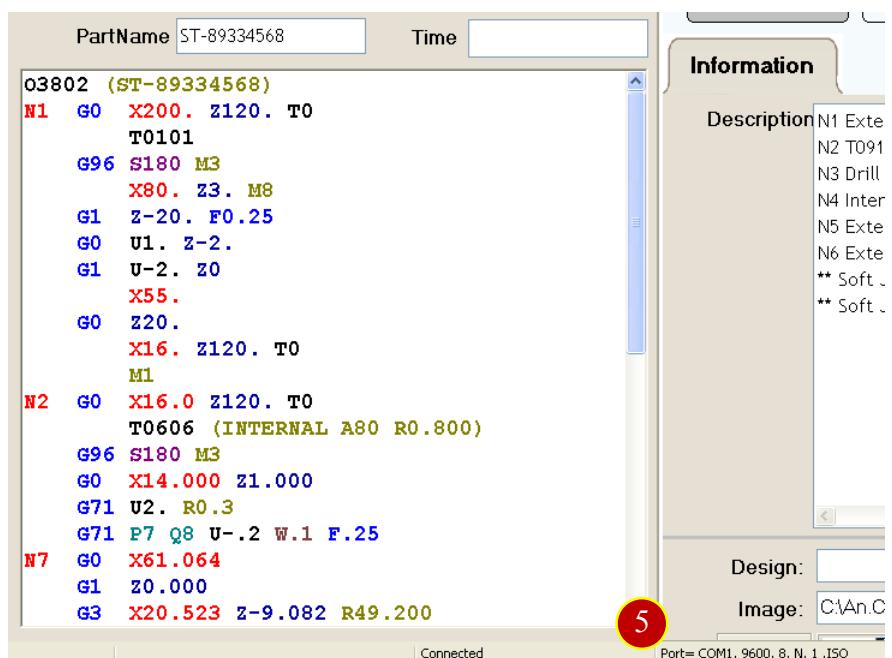
10.5.1 Change the Transmission Protocol

e.g. Change the transmission protocol to [Defaults-ISO, COM1, 9600, 8, n, 1]

- (1) Click 【Transmission】
- (2) Click 【Select Transmit Set】
- (3) Click the pull-down list
- (4) Select 【Defaults-ISO, COM1, 9600, 8, n, 1】



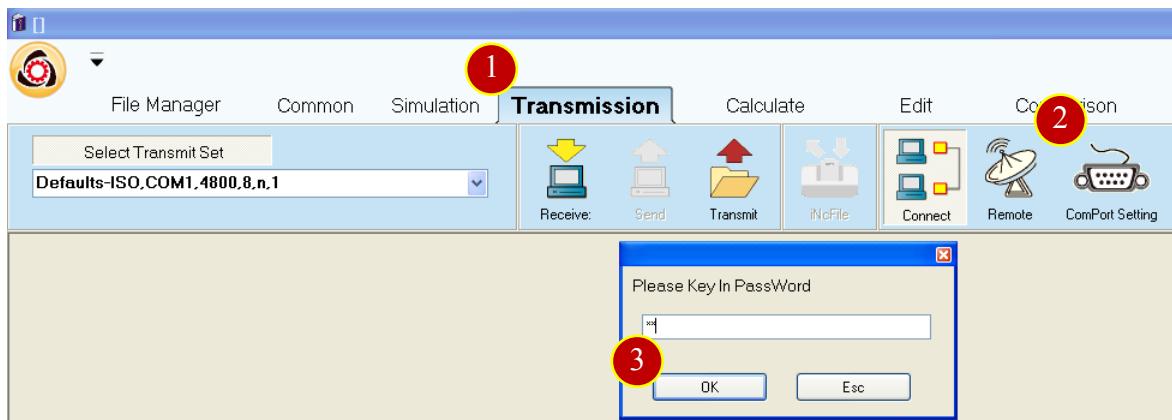
- (5) Show the transmission set



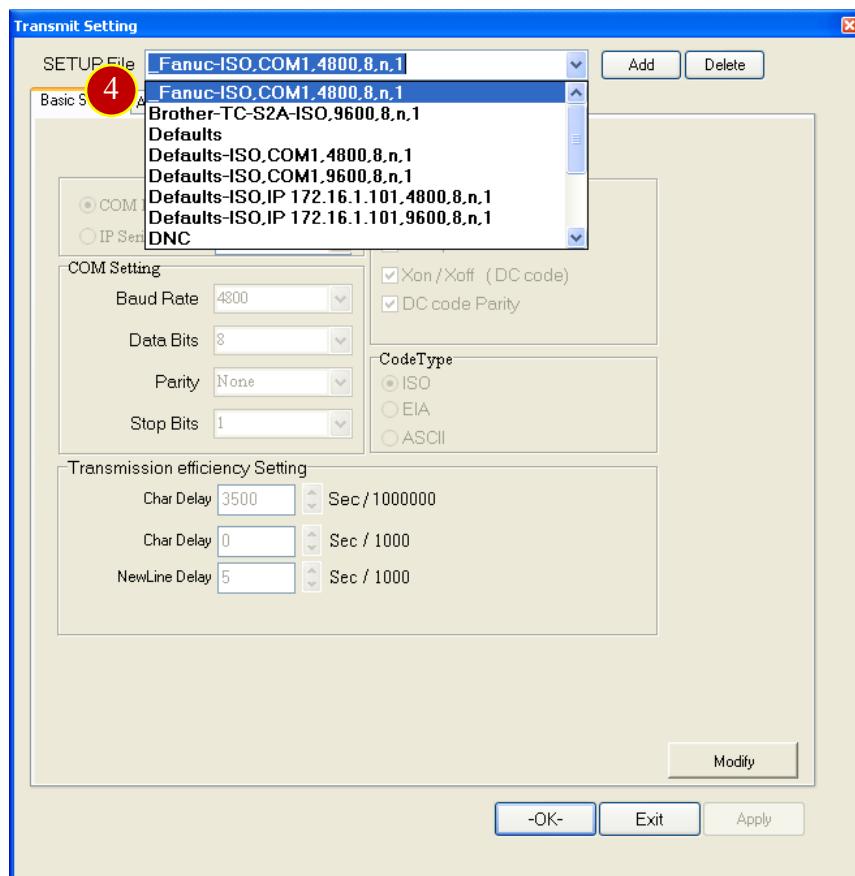
10.5.2 Modify the Transmission Protocol

e.g. Modify 【Fanuc-ISO, COM1, 4800, 8, n, 1】 and open 【DNC Mode】

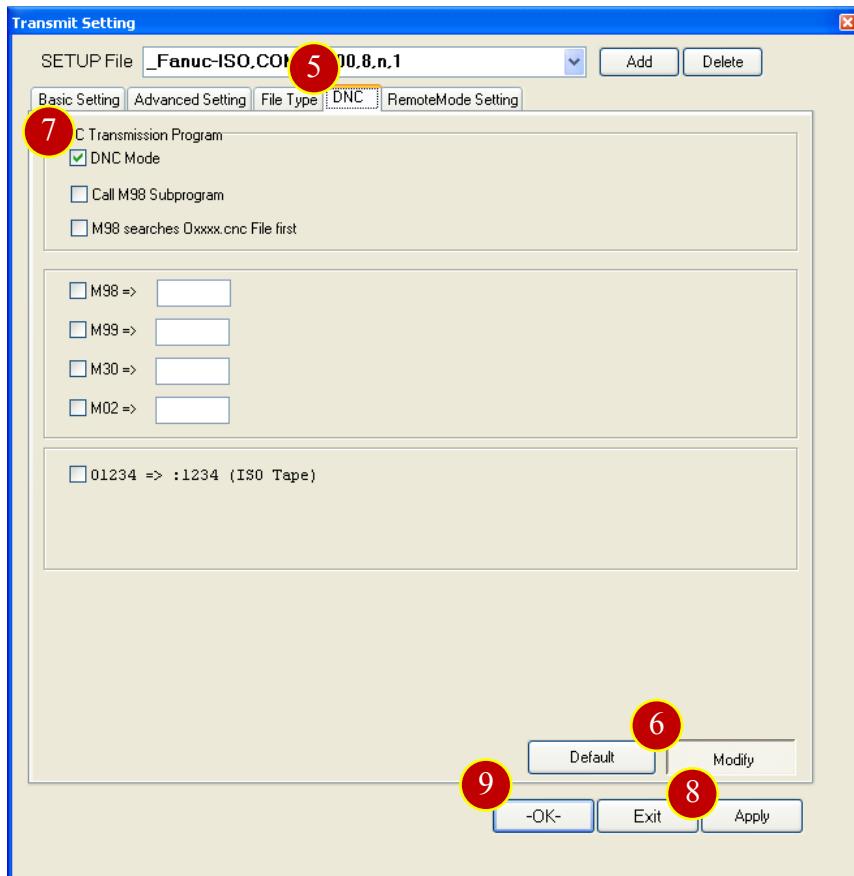
- (1) Click 【Transmission】
- (2) Click 【Protocol setting】
- (3) Key in the password and click 【OK】
the default password is “aa”



- (4) Select 【_Fanuc-ISO, COM1, 4800, 8, n, 1】



- (5) Click 【DNC】
- (6) Click 【Modify】
- (7) Check 【DNC Mode】
- (8) Click 【Apply】
- (9) Click 【OK】



10.5.3 Add New Transmission Protocol

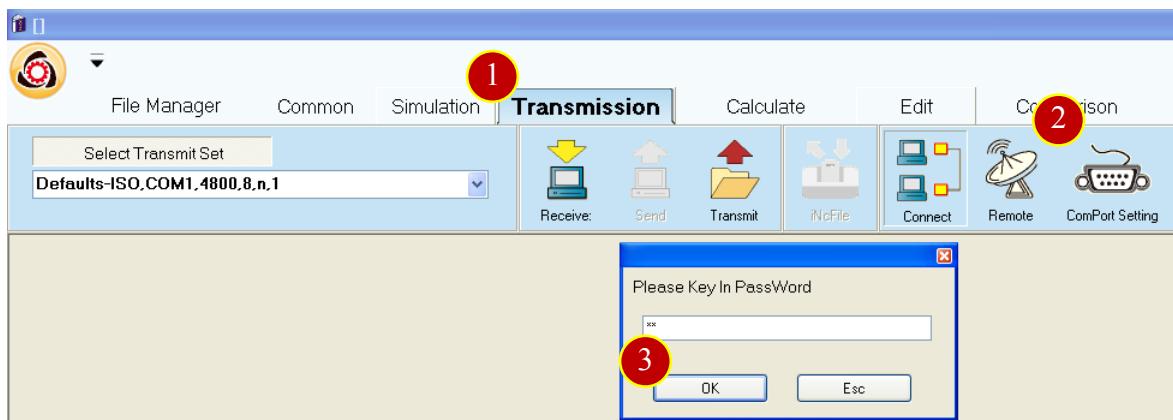
e.g. Add [Defaults-ISO, COM2, 4800, 8, n, 1]

(1) Click 【Transmission】

(2) Click 【Com Port Setting】

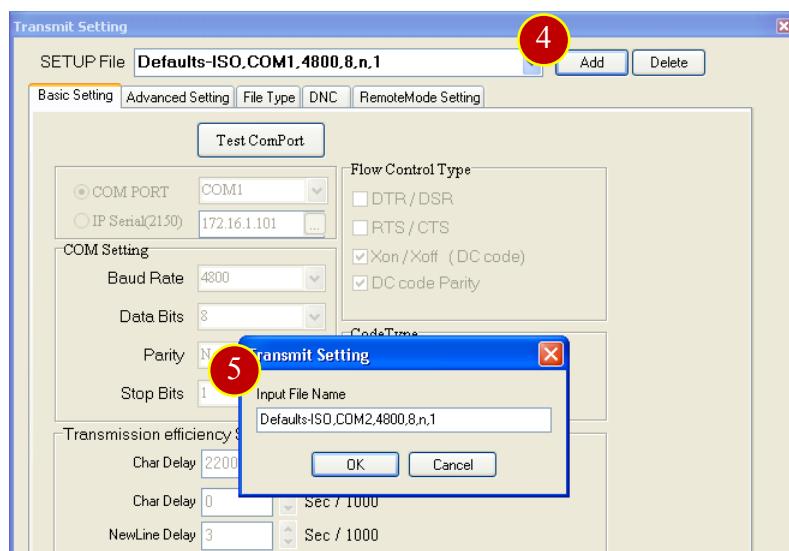
(3) Key in the password and click 【OK】

the default password is “aa”



(4) Click 【Add】

(5) Key in the file name as [Defaults-ISO, COM2, 4800, 8, n, 1], and click 【OK】

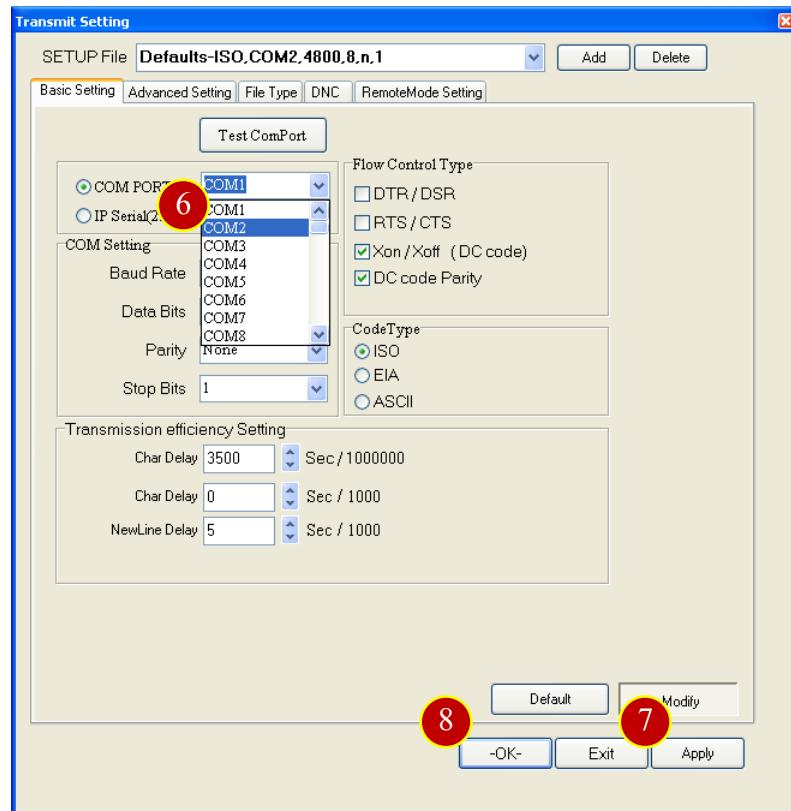


NcEditor MANUAL

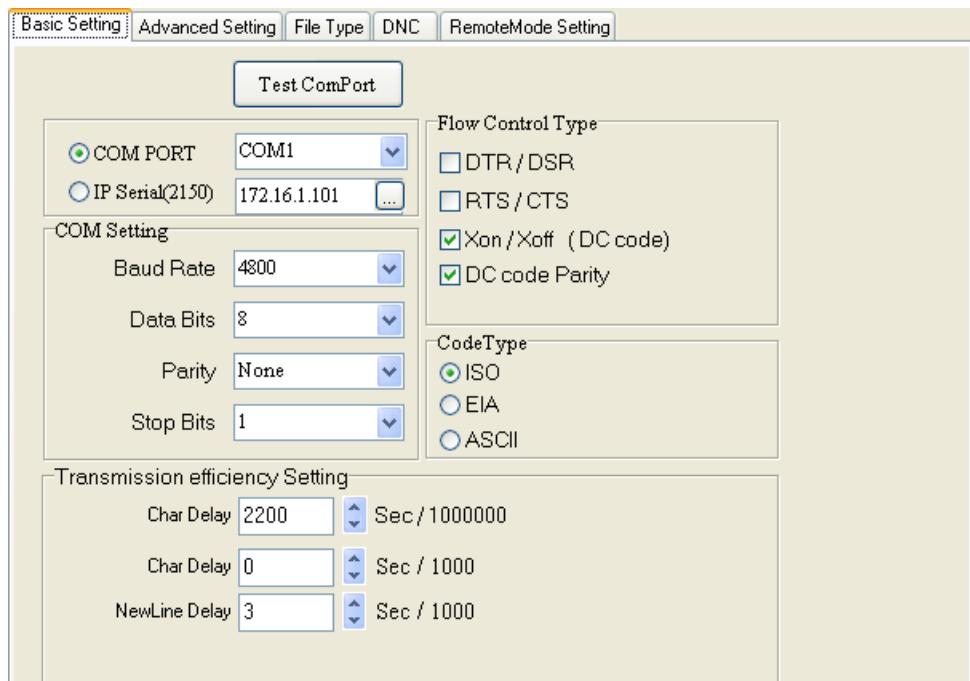
(6) COM PORT select 【COM2】

(7) Click 【Apply】

(8) Click 【OK】



10.6 Protocol Settings – Basic



| Item | Instructions |
|--|--|
| Test ComPort | Test the current condition of the connect port, please check CH 錯誤! 找不到參照來源。 |
| COM PORT | Use COM1 to transmit |
| IP Serial (2150) | Use IP Serial as transmission Details please Check CH10.6.2 |
| COM Setting | The way to transmit the format |
| Flow Control Type | The way to control the flow rate |
| Code Type | Code type |
| Character Time Interval Sec/1000000 | Time Interval between two characters (ns as a unit) |
| Character Time Interval Sec/1000 | Time Interval between two characters (ms as a unit) |
| Line Return Time Interval Sec/1000 | Time Interval of Line Return (ms as a unit) |

10.6.1 Test Com Port

Test the condition of COM Port

Instructions:

COM1(V) = Usable but unconnected

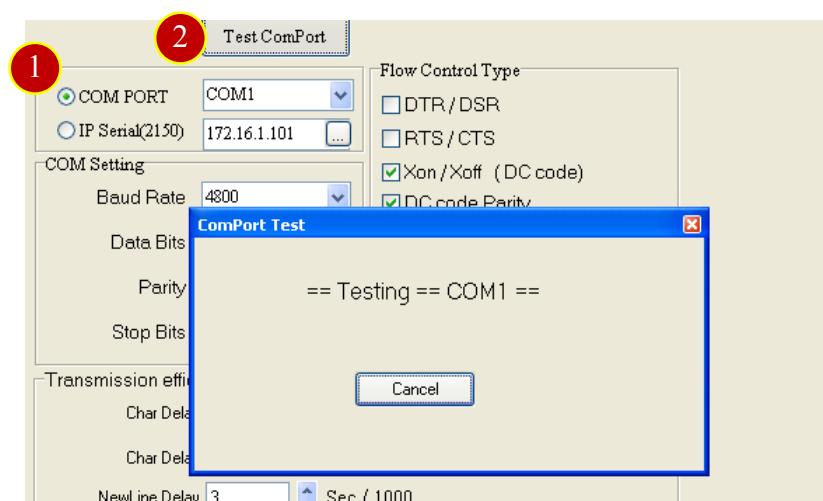
COM2 = Unusable(or is used by other devices)

COM3(#)= COM3 is connected and usable

e.g. Test the condition of COM PORT

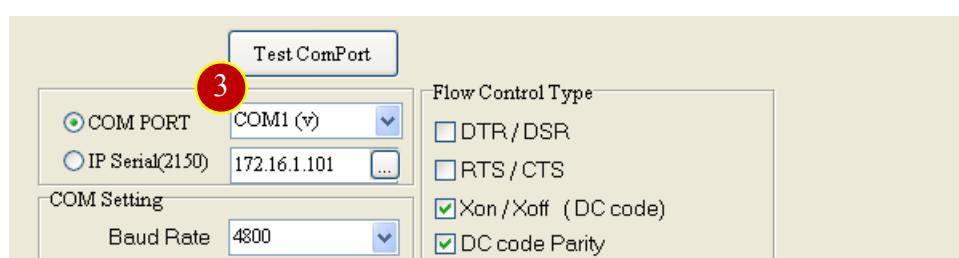
(1) Click 【COM PORT】

(2) Click 【Test ComPort】



(3) After it finishes testing, it will show the condition beside

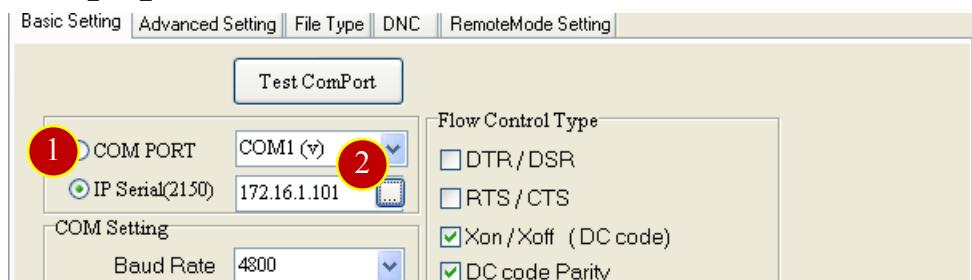
e.g. 【COM1 (V)】 means it is usable now



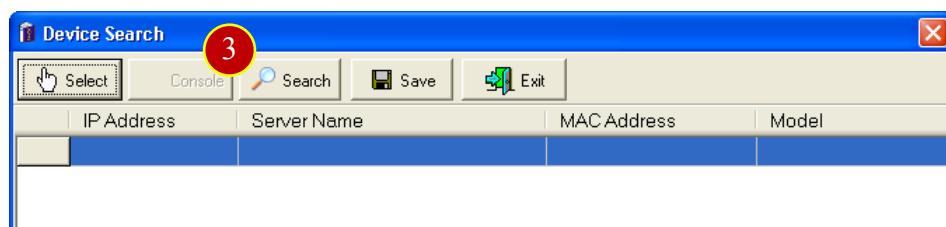
10.6.2 IP Serial

By IP Serial setting, you can transmit with the machine you can use the control interface to set and search IP Serial setting e.g. Search and use IP Serial setting

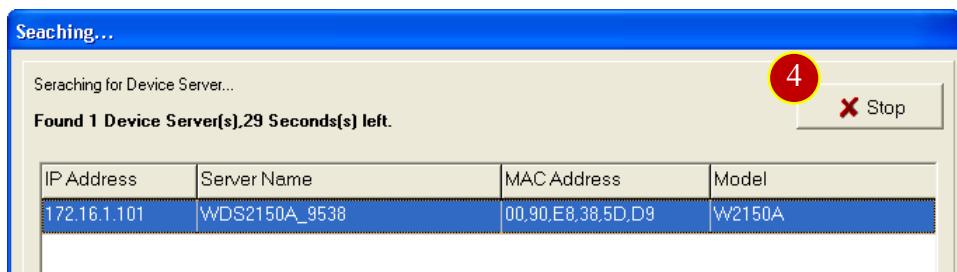
- (1) Select [IP Serial (2150)]
- (2) Click [...] to show the control window of IP Serial



- (3) Click [Search]



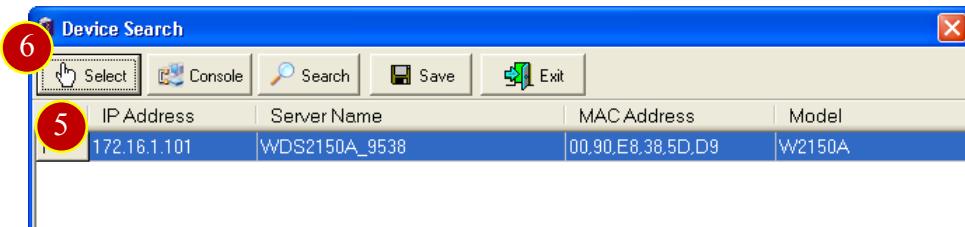
- (4) When IP Serial is searched, window will show the devices found
Wait for 30 sec. or click [Stop]



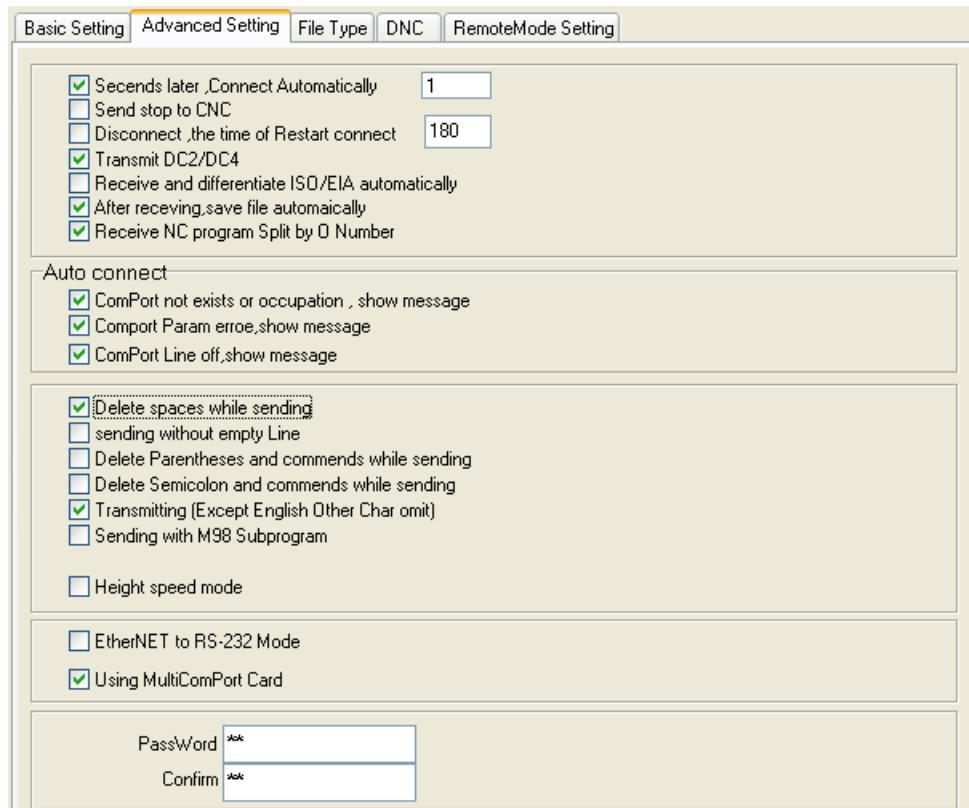
- (5) Click [IP] in IP Serial

e.g. 172.16.1.101

- (6) Click [Select] to finish setting

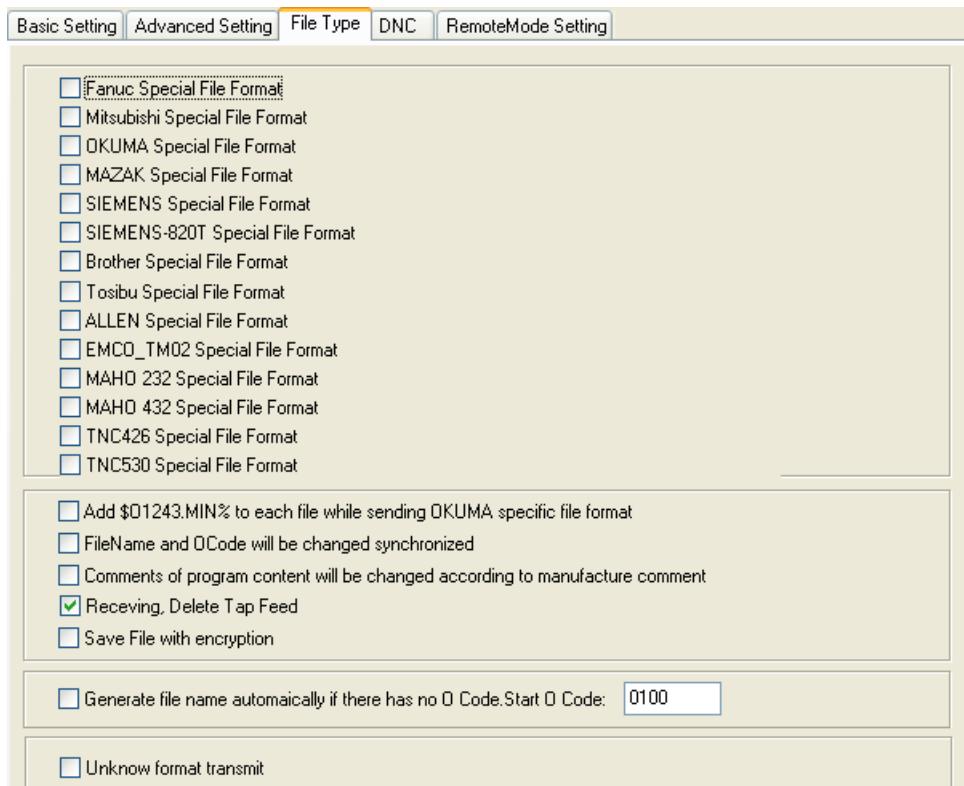


10.7 Protocol Settings – Advance



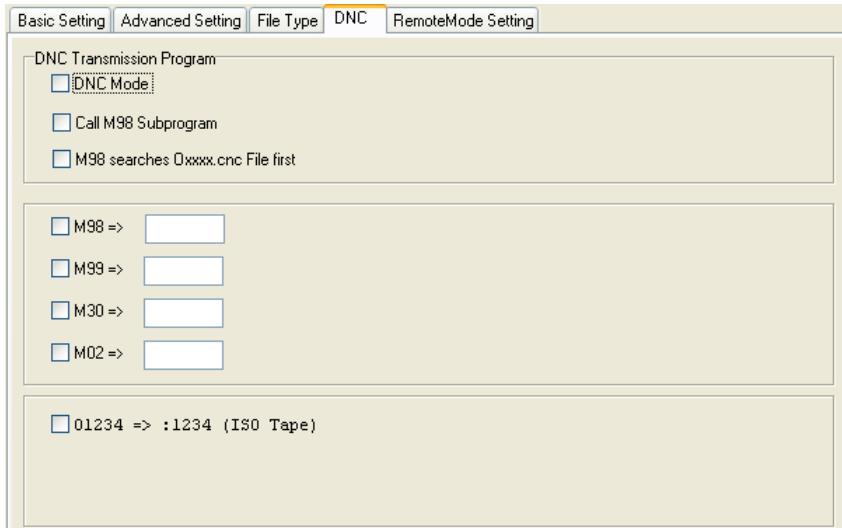
| Items | Instructions |
|---|--|
| Seconds Later- Connect Automatically | Detect COM Port automatically, and delay few seconds when it is connecting |
| Disconnect the time of Restart connect | Unconnected, detect the time interval of COM Port automatically |
| Transmit DC2/DC4 | Support the sending of DC2/DC4 mode and send DC2 at the beginning. Send DC4 in the end |
| Receive and differentiate ISO/EIA automatically | Receive and differentiate Code Type(ISO/EIA) automatically |
| After receiving, save file automatically | After receiving, save file automatically |
| Receive NC Program Split by O Number | Receive several programs from machine, save the program numbers respectively and automatically |
| Password | Set password. Without the password, you cannot log in the setting. Please, do not change it too often. If the revision is done, please remember it or you will not allow to log in the setting |

10.8 Protocol Settings – File Type



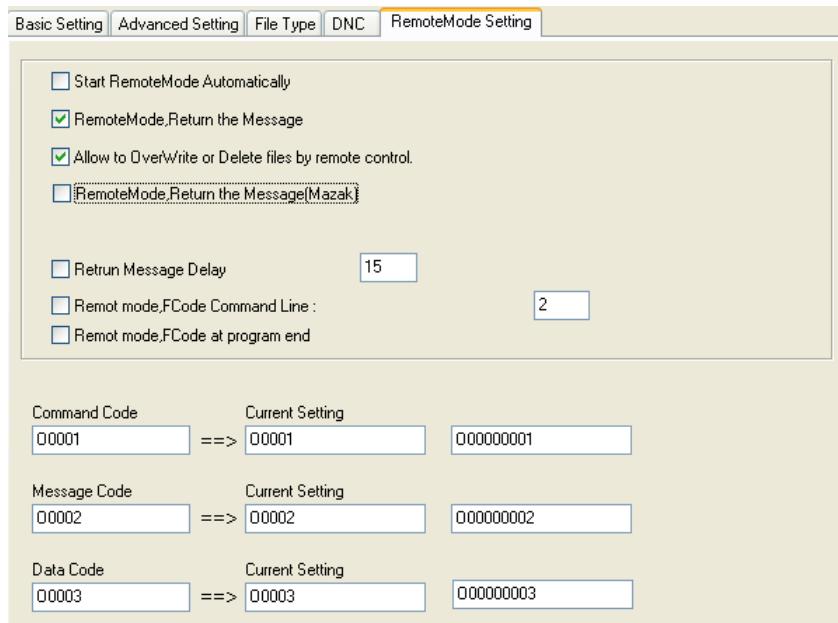
| Item | Instructions |
|--|---|
| Add \$01243.MIN% to each file while sending OKUMA specific file format | When you send the NC program to OKUMA machine, it will add URI automatically |
| File Name and O Code will be changed synchronized | When you receive the save of NC program, O code will change |
| Comments of program content will be changed according to manufacture comment | When you receive the save of NC program |
| Save file with encryption | File will be saved as a encryption file. Only NcEditor can open the file or it will show as scrambled codes |
| Generate file name automatically if there is no O Code. Start O Code | When you are saving the file, system will name the file with numbers and numbers. |

10.9 Protocol Settings – DNC Mode



| Item | Instructions |
|-----------------------------------|---|
| DNC Mode | Use DNC mode. Do one line and send one line simultaneously |
| Call M98 Subprogram | M98 in the program will call the subprogram and send the subprogram. However, M98 doesn't send automatically. |
| M98 searches 0xxxx.cnc File first | M98P program will search the outer program file first |
| M98 M99 M30 M02 | Change M98(M99,M30,M02) to the font setting. ※ You need to check 【<input checked="" type="checkbox"/> DNC mode】 【<input checked="" type="checkbox"/> EXEC M98】 ※ If you keep blank, M98(M99,M30,M02) will be ignored when the transmission is on |
| 01234 => :1234 (ISO Tape) | Change 01234 to 1234(ISO Tape) (For the old type machine) |

10.10 Protocol Settings – Remote Mode



| Item | Instructions |
|--|--|
| Start Remote Mode Automatically | Open NcEditor and it will start 【Remote Mode】 automatically |
| Remote Mode, Return the message | PC returns the message to the machine |
| Allow to Overwrite or Delete files by remote control | When the same program code appears, machine allows to overwrite the files |
| Remote Mode, Return the Message(Mazak) | Only used for Mazak Machine to return the message |
| Return Message Delay | When using the wireless remote transmission mode, it has to delay several seconds before returning the message |
| Remote Mode, FCode Command Line | You can set the location of F Code and you can designate the files |
| Remote Mode, FCode at program end | Set the location of F code in the last line of the program |
| Command Code | Command Code: O Code |
| Message Code | Message Code: O Code |
| Data Code | Data Code: O Code |

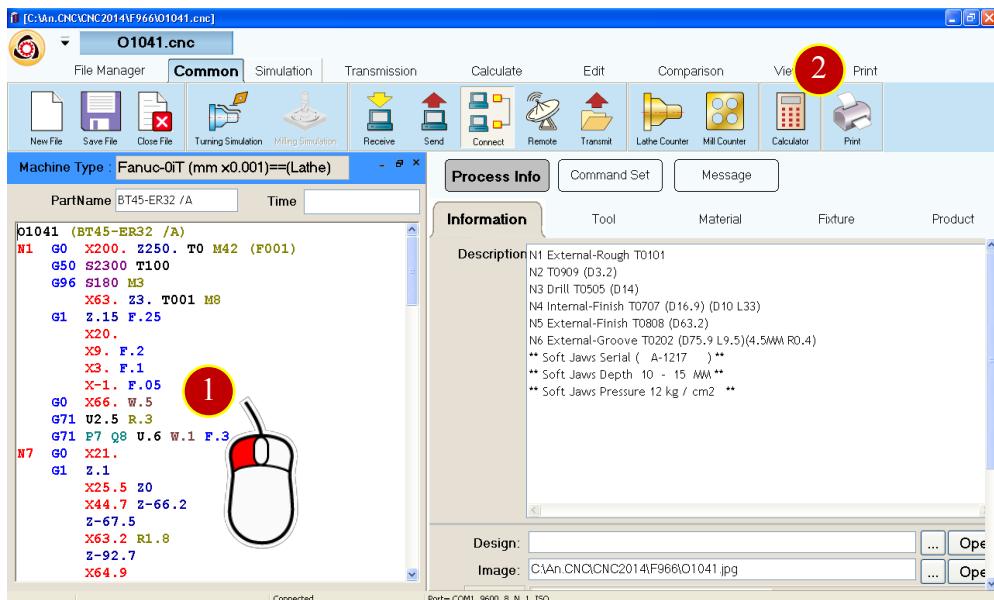
11. Print Programs/Graphics

You can print the program or the related process information from the file you opened.

11.1 Print Programs

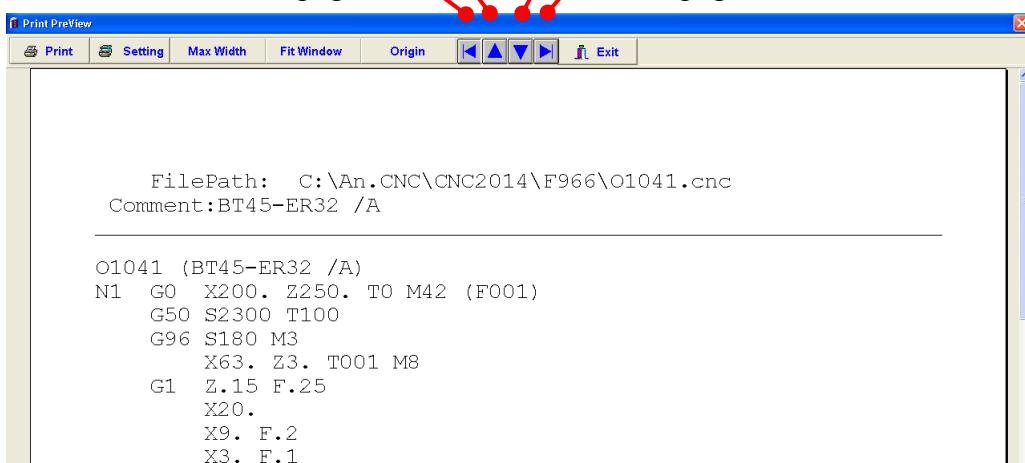
11.1.1 Operation steps of Program Print

- (1) After opening the program, click the program description to ensure the cursor shows on the left side of the program area. If the cursor shows on the tool area, it will print the tool information.
- (2) Click 【Print】



Instructions: use zoom-in, zoom-out buttons to preview the text of the printing. Click 【Print】 after checking your files

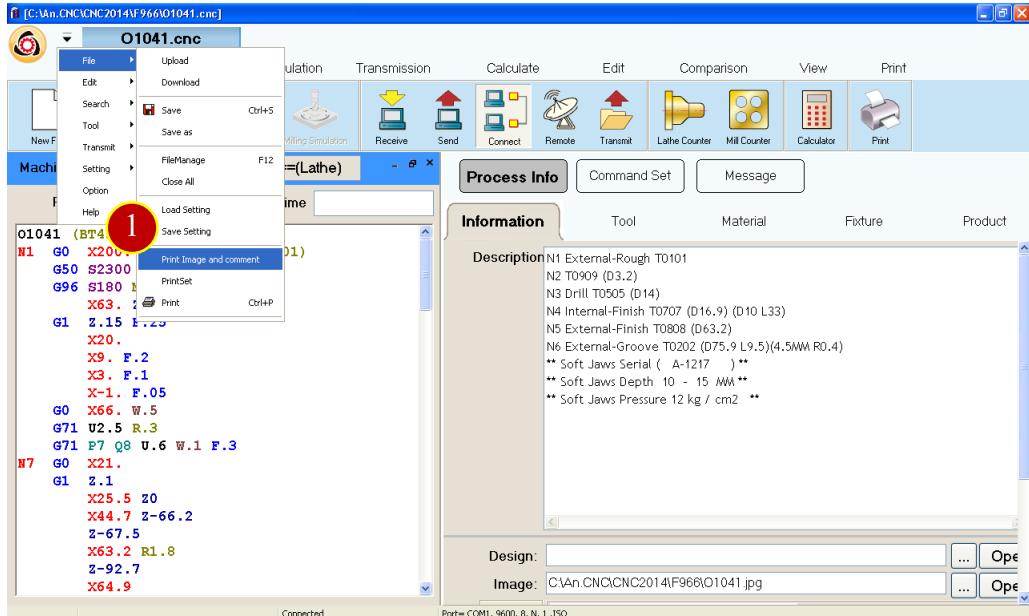
Previous page Next page
First page Last page



11.2 Print Programs/Graphics

11.2.1 Steps of Image Printing and Program Simultaneously

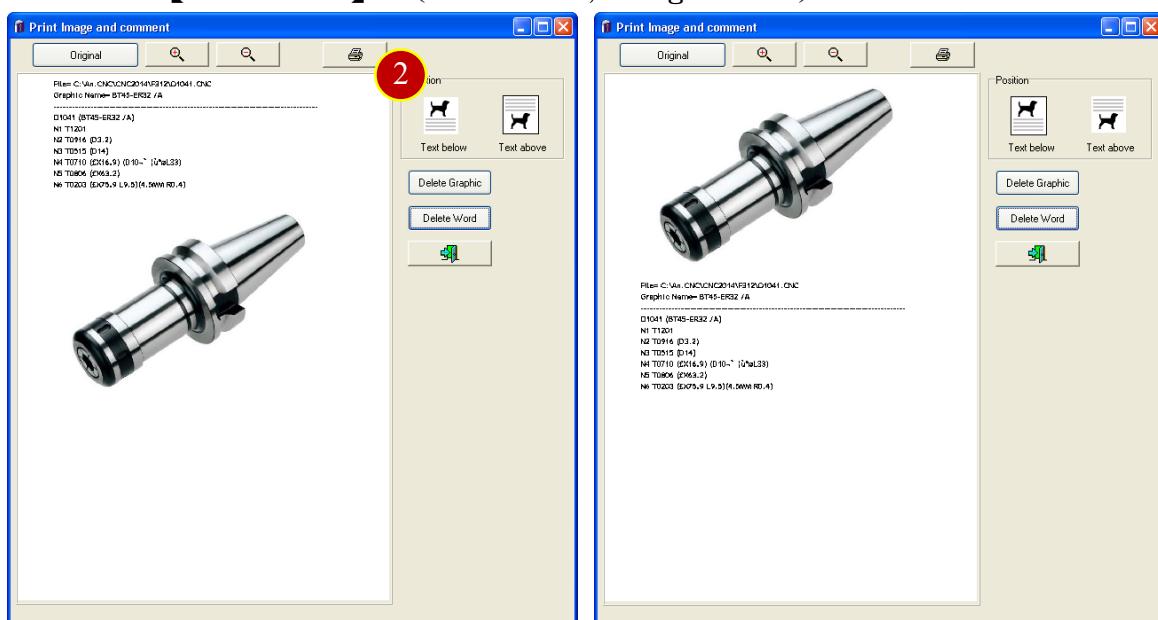
(1) Pull down the list and click 【File】 > 【Print Image and Comment】



(2) In this interface, you can choose the type in accordance with your requirement

【Text below】 = (Text below, Image above)

【Text above】 = (Text above, Image below)

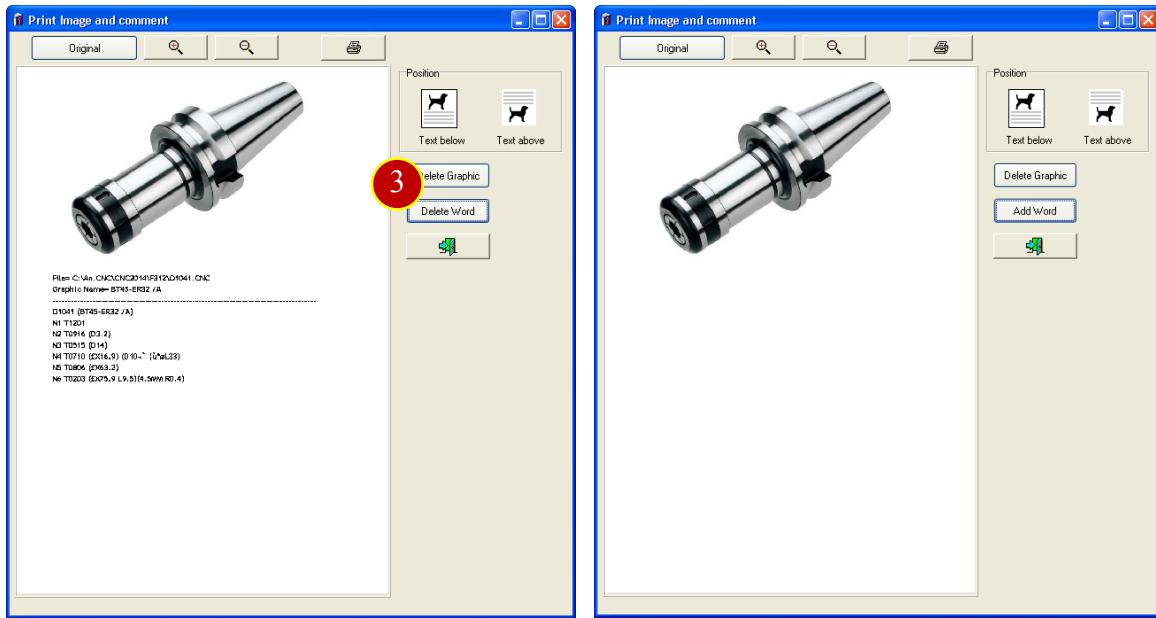


(3) In 【Image and Comment】

Click 【Delete Word】 , means not to print the words

Click 【Add Word】 to add the words

【Delete Graphics】 and 【Add Graphics】 will be the same way as described above

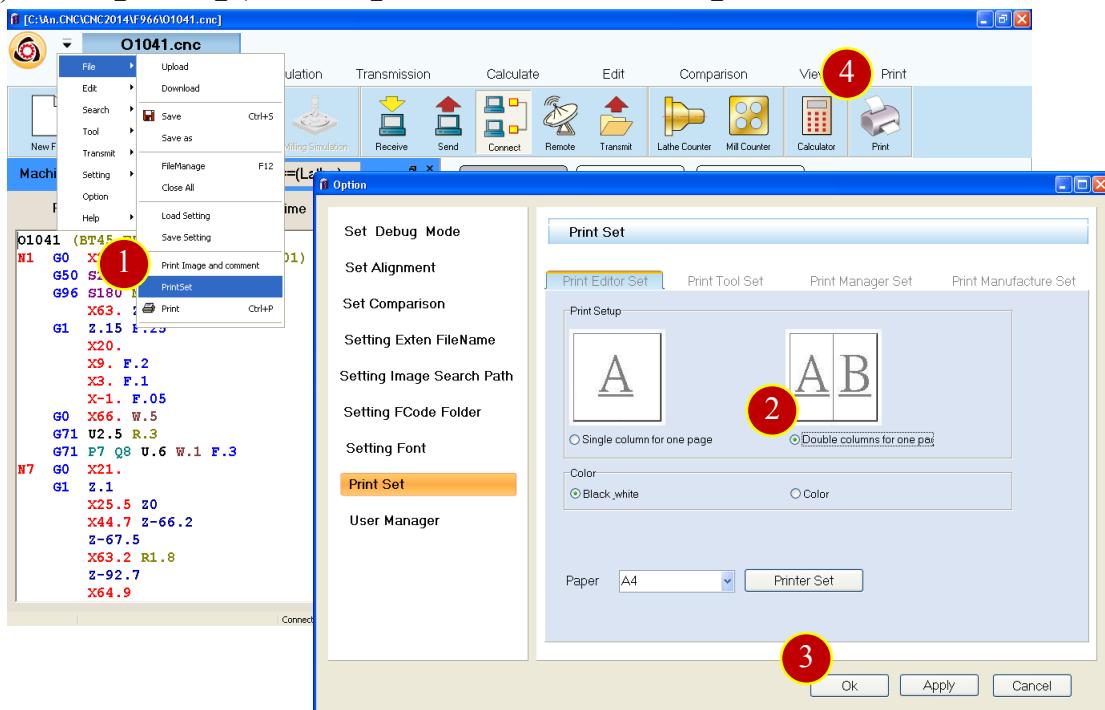


11.3 Double Columns for One Page

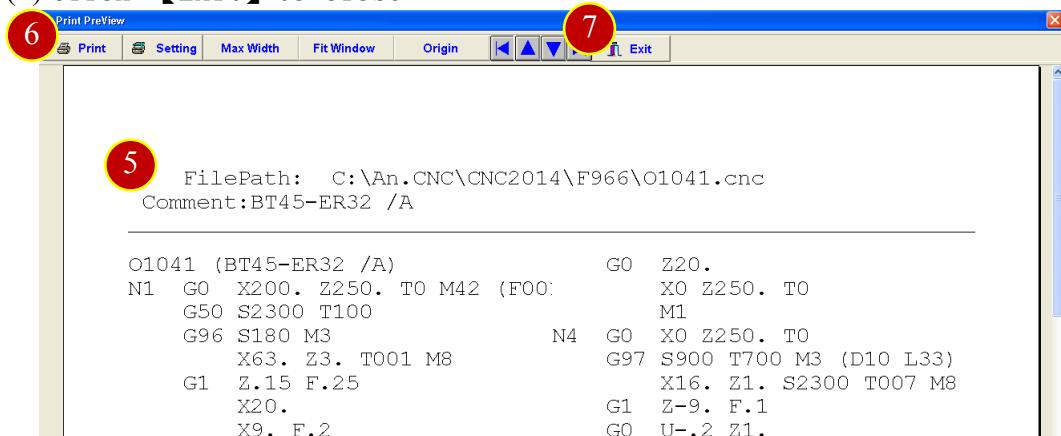
11.3.1 Double Columns for One Page Settings

When CNC program is too long, you can divide one single page into two columns to save the paper consumption

- (1) Click pull-down list, and click 【File】 > 【Print Set】
- (2) Select 【Double columns for one page】
- (3) Click 【OK】
- (4) Click 【Print】 , enter 【Print Preview Window】



- (5) In 【Print Preview Window】 , one page is divided into two columns to save the paper consumption
- (6) Click 【Print】
- (7) Click 【Exit】 to close



12. Remote Mode for Transmission

12.1 Remote Transmission

After activate NcEditor remote mode for transmission of PC, use command to control NcEditor transmission in machine

12.1.1 Activate remote transmission

e.g. Activate remote mode for transmission

(1) Click 【Transmission】 tab

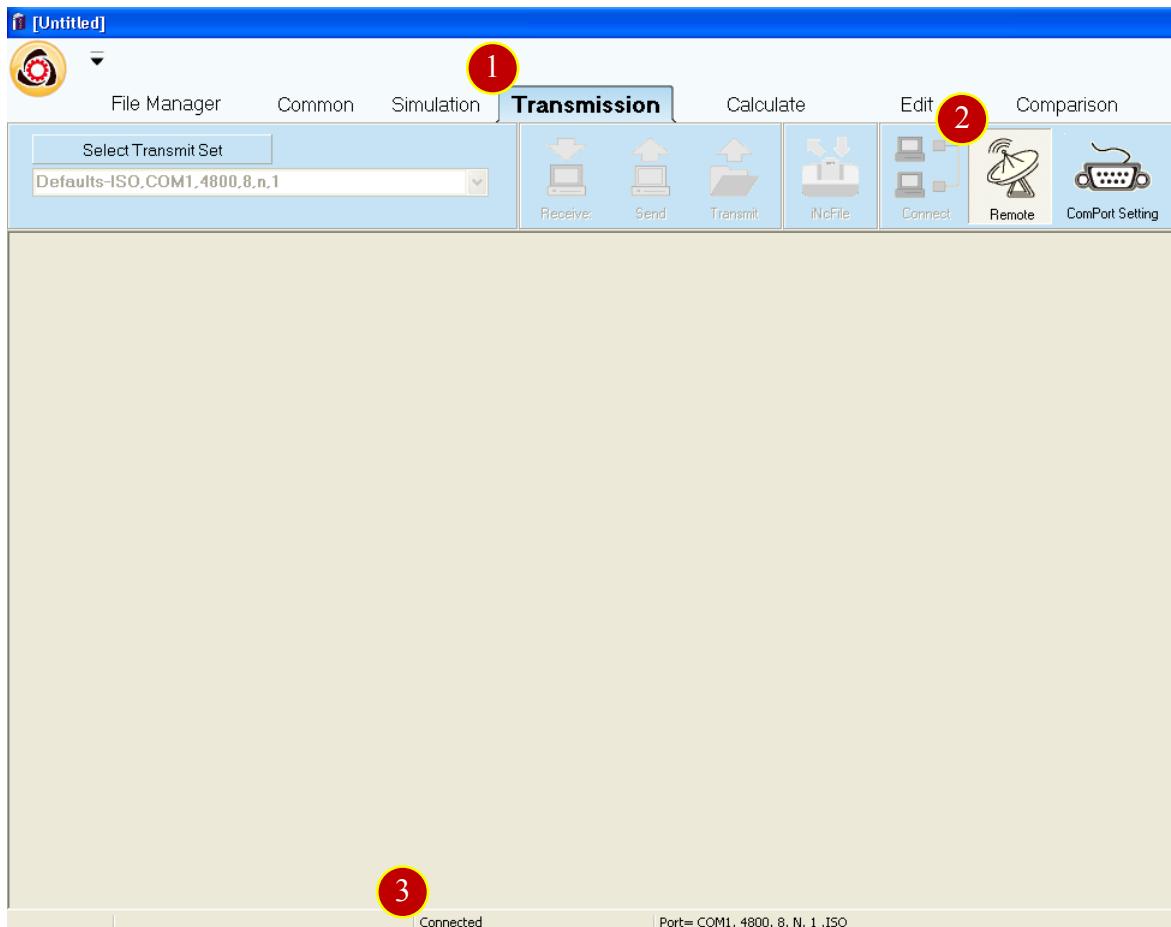
(2) Click 【Remote】 to connect CNC machine

※Please check following items when can't connect machine

(a) Use Remote Mode (Cancel normal mode)

(b) Check transmission setting, more information at CH10.5.2

(3) After complete connect, display 【Connected】





12.2 Remote Transmission Command List

| Remote Command | Explain |
|-----------------------------|---|
| O0001 | Specific program number O0001 for command |
| 1. Change folder | |
| F123 | Change to folder F123 |
| F123 F220 | Change to subsidiary folder F220 of folder F123 (same as F123\F220) |
| C1234 | Create factory classification folder F1234, C is the abbreviation of create |
| 2. Program transmission | |
| N1234 | Command to transmit program O1234 to CNC |
| N1234 N5678 | Command to transmit programs between O1234 and O5678 |
| (O1234) | Command to transmit program O1234 to CNC |
| (O1234)(O5678) | Command to transmit programs between O1234 and O5678 |
| 3.File content transmission | |
| / | Command to transmit program list(name, comment, size, and date) |
| // | Command to transmit folder list |
| /(ER32) | Command to transmit file content available of ER32 in the folder |
| S99 | Send all files available to CNC Use with /(XXX), S99 is the designation code |
| / N55 | Command to transmit program list(name, comment) |
| / N99 | Search the biggest number in the folders in command,N99 is the designation code |
| M900 | Not transmit message regardless of setting, absolutely no message |
| M901 | Command to transmit message regardless of setting, must transmit message |
| F123 | Select F123 as default folder number |
| M905 | |
| M906 | Cancel default folder number |
| 4.Rename and delete file | |
| D1234 | Delete program O1234 |
| R1234 R5678 | Rename program O1234 to O5678 |
| 5.Direct command | (Except F folders) |
| [ABC123] | Change to folder ABC123 |
| ((ABC123)) | Change to folder ABC123 |
| ((C./AN.CNC/ABC123)) | Change to folder C:\AN.CNC\ABC123 |
| ((AAA))((BBB)) | Change to folder AAA\BBB\ |
| (BT50ER16.CNC) | Command to transmit the file BT50ER16.CNC |

12.3 Remote Transmission Message Explain List

12.3.1 Normal Message Explain

| Message | Explain |
|--|--|
| O0002(MESSAGE) | O0002(MESSAGE) |
| (MESSAGE)(F020,CREATE=OK) | (MESSAGE)(Complete create folder F123) |
| (MESSAGE)(F123, DIR. CHANGE=OK) | (MESSAGE)(Complete change to folderF123) |
| (MESSAGE)(F123, DIR. TRANSMIT=OK) | (MESSAGE)(Complete transmit Files content of F123) |
| (MESSAGE)(O1234, PROG. TRANSMIT=OK) | (MESSAGE)(Complete transmit O1234) |
| (MESSAGE)(O5678, PROG. RECEIVE=OK) | (MESSAGE)(Complete receive O5678) |
| (MESSAGE)(O1234, DELETE=OK) | (MESSAGE)(Complete delete O1234) |
| (MESSAGE)(O1234. TO. O5678. RENAME=OK) | (MESSAGE)(Complete rename O1234 to O5678) |

12.3.2 Alarm Message Explain

| Message | Explain |
|--|---|
| (ALARM/001)(O1234)(F123, NO.EXIST =?) | (ALARM/001)(F123 folder not exist) |
| (ALARM/111)(O1234. NO. SPECIFIC. F***=?) | (ALARM/111)(No command specific folder Fxxx when save the file) |
| (ALARM/114)(DIR, MAKE, FAIL =?) | (ALARM/114)(Fail to create new folder) |
| (ALARM/071)(O1234, PROG. NO. FOUND =?) | (ALARM/071)(Not find O1234) |
| (ALARM/071)(O6011. O6015. PROG. NO. VALID =?) | (ALARM/071)(Not find file from O6011 to O6015) |
| (S99. NO. FILE. AVAILABLE =?) | (No available file to transmit in S99) |
| (ALARM/112)(O1234, DELETE. FAIL =?) | (ALARM/112)(Fail to delete O1234) |
| (ALARM/113)(O1234, RENAME. FAIL =?) | (ALARM/113)(Fail to rename O1234) |
| (ALARM/073)(O1234, ALREADY EXIST =?) | (ALARM/073)(File O1234 has already exit) |
| (PLEASE CHECK !) | (Please check again) |
| (ERROR. FILE. O1234, READ. ONLY =?) (PLEASE CHECK !) | (ERROR, O1234 read only, system not allowed to replace it)(Please check again)(Modify read only settings or comport settings) |

Description:

- (1) When displays " OK" on the right of the message , it means all command completed
- (2) When displays " ?" on the right of the message, it means command not completed
- (3) When displays ALARM on the left side, please check the error code according to alarm number

12.4 Programs from CNC to PC

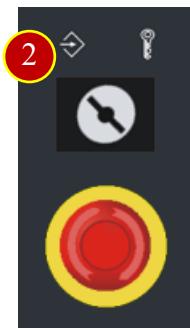
12.4.1 Programs from CNC to PC Settings Steps (FANUC 0i-MD, 0i-TD)

| Steps | Operation Buttons | Explain |
|-------|---|---|
| 1 | 【EDIT】 | Change 【Mode Switch】 to 【EDIT】 mode |
| 2 | 【OFF】 | Switch the Edit key to 【OFF】 |
| 3 | O1234 【↓】 | Key in O1234, press 【↓】 to open CNC code O1234. Then window displays the number O1234 and its content |
| 4 | Move the cursor Edit the program (F123) 【INSERT】 | O1234 (DEMO TEST) N1 G50 S2000 (F123) Key in folder number in the second line to command program to save at folder F123 G90 S200 M3 T0101 M8 |
| 5 | O1234 【PUNCH】 | Press the rightest operation button 【OPRT】 at the bottom of the screen Press operation button for details 【>】 until display 【READ】【PUNCH】 Key in O1234, and press transmission button 【PUNCH】 When display 【EXEC】【STOP】 , press 【EXEC】 to transmit CNC code from CNC to PC to save, or press 【STOP】 to stop transmitting |
| 6 | 【READ】 | Press 【READ】 , then display 【EXEC】【STOP】 Press 【EXEC】 to receive message from PC Press 【STOP】 to stop receiving CNC code and message |
| 7 | O0002 【DELETE】 | After check the message, key in O0002 and press 【DELETE】 to delete O0002 |

(1) Change 【Mode Switch】 to 【EDIT】 mode



(2) Switch the Edit key to 【OFF】



NcEditor MANUAL

Remote Mode for Transmission

(3) Key in 01234, press **【↓】** to open CNC code 01234

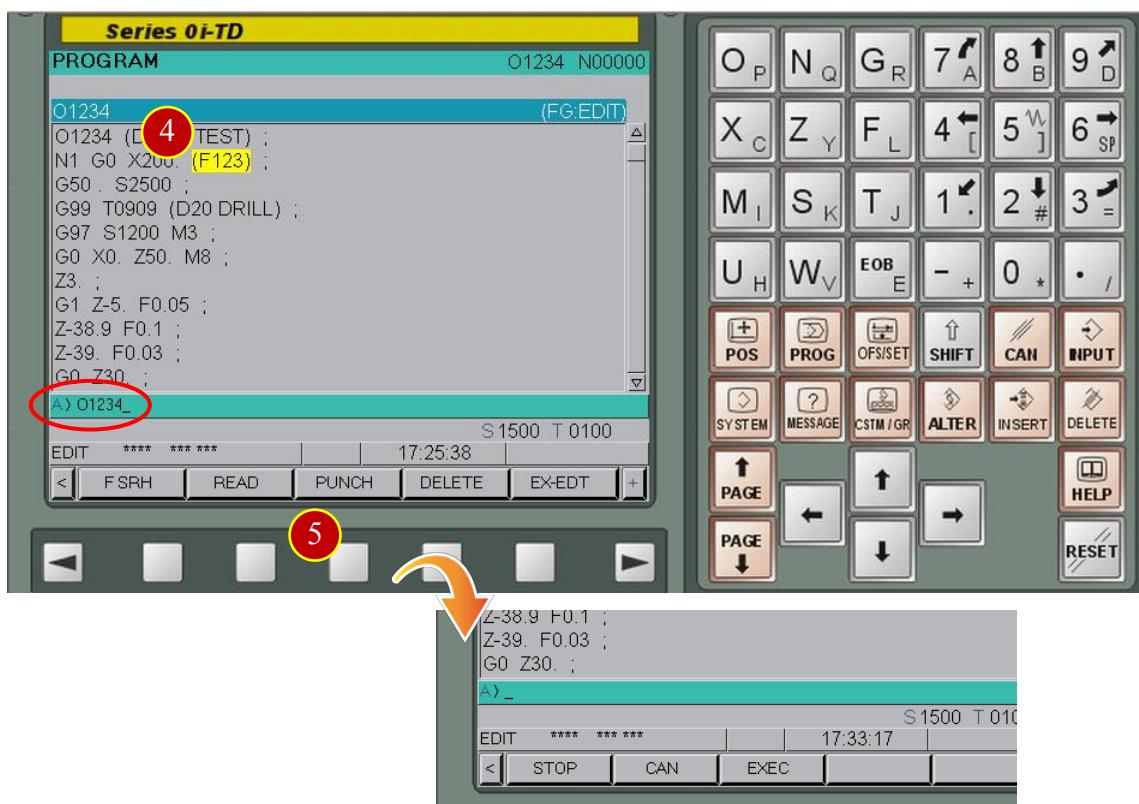


(4) To edit CNC code, key in folder number in the second line

e.g. CNC code 01234 to save at folder F123

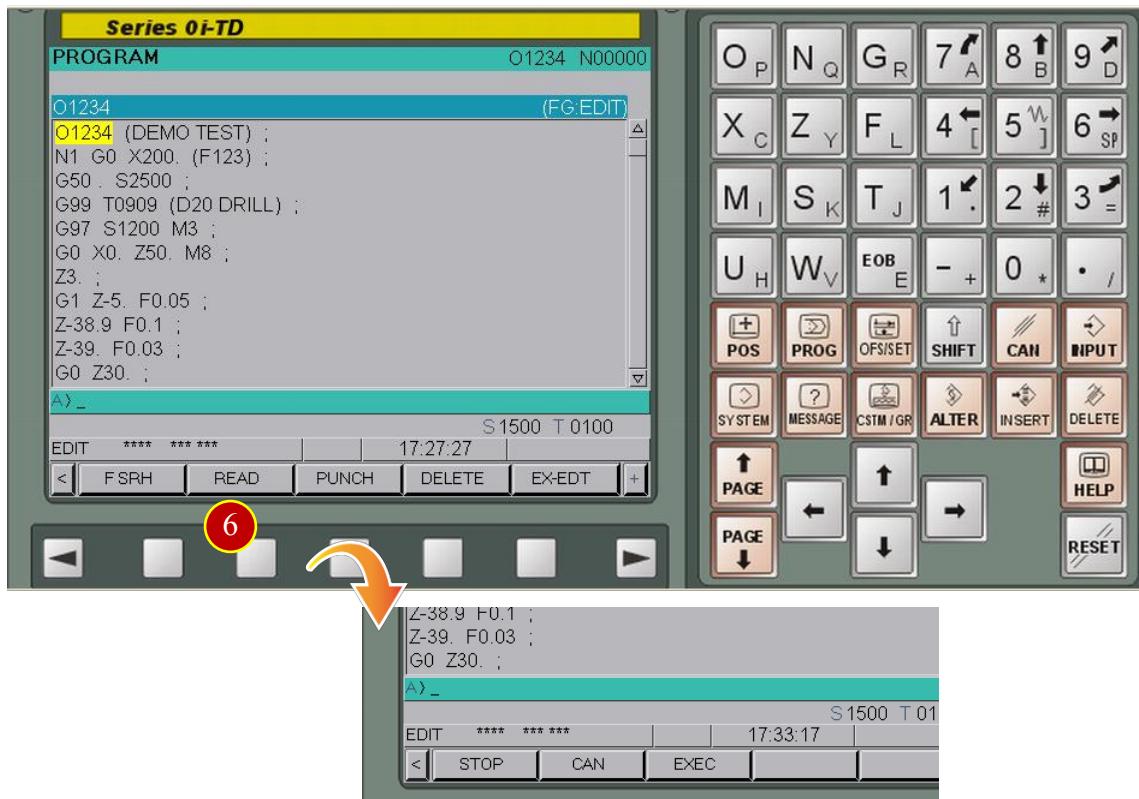
(5) Key in 01234, and press **【PUNCH】**

When display **【EXEC】** **【STOP】**, press **【EXEC】** to transmit CNC code from CNC to PC to save

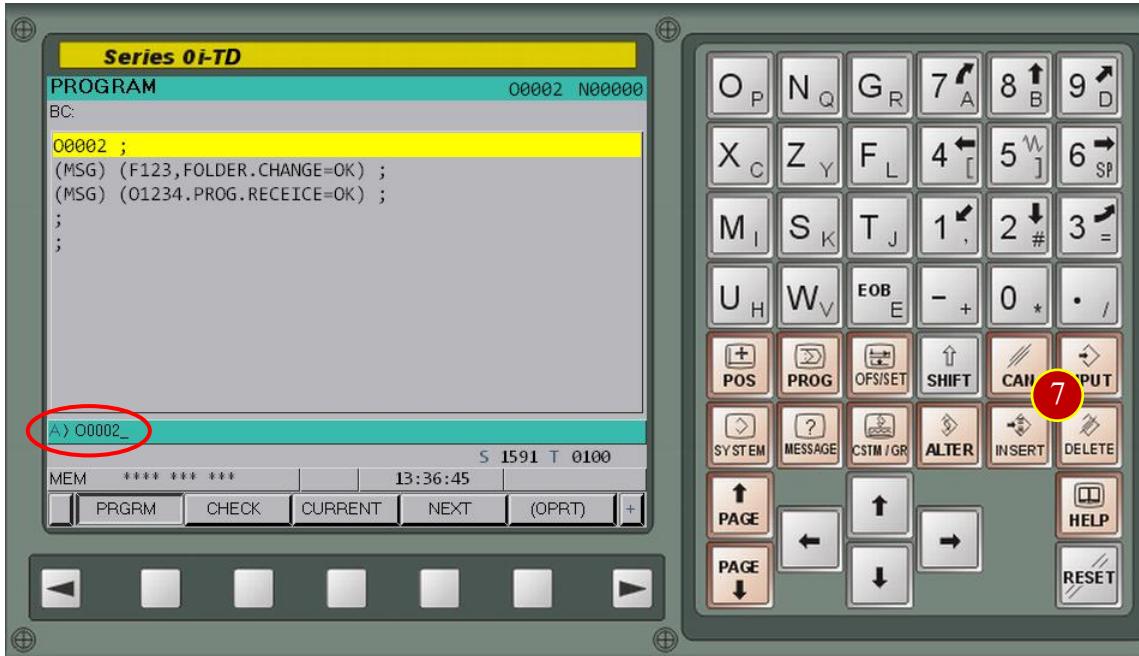


(6) Press 【READ】 , then display 【EXEC】【STOP】

Press 【EXEC】 to receive message from PC



(7) After check the message, key in 00002 and press 【DELETE】 to delete 00002



12.4.2 Message Explain(Folders)

[=OK] means execute programs accurately, [=?] means execute programs wrongly

(1) Must command program to classification folder in the second line by key in number begin with **[F]** e.g. F123

If no command classification folder, displays:

00002(MESSAGE)

(ALARM/111)(01234, NO. SPRCIFIC. F*** =?)(not yet command classification folder when save the program)

(2) Command program to classification folder not exist, displays:

00002(MESSAGE)

(ALARM/001)(01234)(F123. NO. EXIST=?)(The classification folder F123 not exist)

12.4.3 Message Explain(Program file)

(1) After PC receive program and save accurately, CNC machine displays message:
00002(MESSAGE)

(MESSAGE)(01234, PROG. RECEIVE=OK)(01234 Complete receive and save)

(2) If the program number has already in the PC, CNC machine displays:
00002(MSEEAGE)

(ALARM/073)(01234, PROG. ALREADY. EXIST=?)(the program number has already exist)

(PLEASE CHECK !)(Please check again)

(3) To replace the old one when program number has already exist in PC, key in sign **[/]** in the second line in the program
e.g. 01234

N1 G0 X120. Z120. T0(F123/)

G96 S180 M3

(4) To automatically add another new number when program number has already exist in PC, key in sign **[/]** in the second line in the program
e.g. 01234

N1 G0 X120. Z120. T0(F123//)

G96 S180 M3

(5) Read only settings not allow to modify when program number has already exist in PC, CNC machine displays:

00002(MESSAGE)

(ERROR, FILE. 01234. READ. ONLY=?)(Program Number has already exist , read only not allow to modify)

Check items:



Remote Mode for Transmission

- (a) Modify CNC comport settings – advance settings to allow remote cover and delete file (DELETE FILE)
- (b) Windows system, to set the file as **【READ ONLY】**

12.4.4 Programs from CNC (FANUC 0M 0T) to PC

| Steps | Operation Buttons | Explain |
|-------|---|---|
| 1 | 【EDIT】 | Change 【Mode Switch】 to 【EDIT】 mode |
| 2 | 【OFF】 | Switch the Edit key to 【OFF】 |
| 3 | O1234 【↓】 | Key in O1234, click【↓】to open program O1234. It displays the number O1234 and its content |
| 4 | Move the cursor Edit the program (F123) 【INSERT】 | O1234 (DEMO TEST) N1 G50 S2000 (F123) G90 S200 M3 T0101 M8 |
| | | Key in folder number in the second line to command program to save at folder F123 |
| 5 | O1234 【OUTPUT】 | Key in O1234, press 【OUTPUT】 to transmit program to PC to save |
| 6 | 【INPUT】 | Press 【INPUT】 to receive message from PC Specially notice not to key in program number |
| 7 | O0002 【DELETE】 | After check the program, delete O0002 |

12.4.5 Programs from CNC (FANUC 18M 21T machine) to PC

| Steps | Operation Buttons | Explain |
|-------|---|--|
| 1 | 01234 【↓】 | Key in O1234, press 【↓】 to open program O1234. Then window displays the number O1234 and its content |
| 2 | Move the cursor Edit the program (F123) 【INSERT】 | O1234 (DEMO TEST) N1 G50 S2000 (F123) G90 S200 M3 T0101 M8 |
| | | Key in folder number in the second line to command program to save at folder F123 |
| 3 | 01234 【PUNCH】 | Press the rightest operation button 【OPRT】 at the bottom of the screen Press operation button for details 【>】 until display 【READ】 【PUNCH】 Key in O1234, and press transmission button 【PUNCH】 or press 【THIS】 to transmit editing program Press execute button 【EXEC】 or stop button 【STOP】 |
| 4 | 【READ】 | Press 【READ】 , and it displays 【EXEC】【STOP】 Press 【EXEC】 to receive program and message from PC, or press 【STOP】 to stop receiving program and message |
| 5 | O0002 【DELETE】 | After check the program, delete O0002 (Note: set comport to the avoid this step) |

12.4.6 Programs from CNC (MITSUBISHI) to PC

| Steps | Operation Buttons | Explain |
|-------|---|---|
| 1 | 1234 【INPUT】 | Key in O1234, press 【INPUT】 to open program O1234. It displays the number O1234 and its content |
| 2 | Move the cursor Edit the program F123 【INPUT】 | O1234 (DEMO TEST) N1 G50 S2000 (F123) G90 S200 M3 T0101 M8 |
| 3 | 1234 【INPUT】 | Press main list【ALARM/DIAGN】on the right side of screen And press 【MENU】 at the bottom of the screen Until display 【INPUT】【OUTPUT】 Press #(1) DATA(1234), #(1) means to output CNC main program DATA(1234) stand for program number O1234 Press 【INPUT】 to execute it |
| 4 | 【INPUT】 | Press #(1) DATA (When key in program, DATA column can be ignored and not key in program number) Press 【INPUT】 to receive program and message from PC |
| 5 | 00002 【DEL】 | After check the program, delete O0002 |

12.4.7 Programs from CNC(OKUMA U100L) to PC

| Steps | Operation Buttons | Explain |
|-------|---|---|
| 1 | 01234.MIN 01234 (DEMO TEST) N1 G50 S2000 F123 G96 S200 M3 T0101 M8 | Select file O1234.MIN = file name Search and open program number O1234 It displays program number O1234 and its content For example: Edit program (must have program number) Stand for folder number F123 Key in folder number F123 in the second line To command O1234 to save in folder F123 |
| 2 | 【F3】【PIP】 | Press 【F3】【PIP】 at the bottom of the screen |
| 3 | 【F2】【PUNCH】 (MD1: INDEX) 【WRITE】 | Press 【F2】【PUNCH】 at the bottom of the screen Click (MD1: INDEX) Move the cursor on program number 01.MIN and press 【WRITE】【Confirm】 |
| 4 | 【WRITE】【Confirm】 | Press 【WRITE】【Confirm】 complete transmission |
| 5 | 【F1】【READ】 【WRITE】【Confirm】 | Press 【F1】【READ】 at the bottom of the screen Press 【WRITE】【Confirm】 at the right of the screen (Note: no need to key in any word here) |
| | | Complete read program as A.MIN, or O0002.MIN (Note: if it has already exist, it will display “ALREADY EXIST SAME PROGRAM NO. LOAD OK?”, please key in [Y]) |
| | | Select file A.MIN or O0002.MIN, check the program content |

12.4.8 Programs from CNC (Brother TC-S2A) to PC

| Steps | Operation Buttons | Explain |
|-------|---|--|
| 1 | 【PROGRAM EDIT】 F123 ; N5678 ; | Select function 【PROGRAM EDIT】 to edit program , number 0001 for example: F123 ; N5678 ; complete edit |
| 2 | 【EXTERNAL PROGRAM I/O】 | Select function 【EXTERNAL PROGRAM I/O】 (Send, input/output external program) |
| 3 | 【OUTPUT TO PTP】 【PROGRAM】 | Select function 【OUTPUT TO PTP】 Select function 【PROGRAM】 |
| 4 | 0001 【E.STA】 | At PROGRAM_ cursor flashing spot key in program number 0001 Press 【E. STA】 to execute |
| 5 | 【F0】 【F0】 | When complete transmitting files, double press 【F0】 (RETURN MENU) (Functions option , return twice) |
| 6 | 【INPUT FROM PTR】 | Select function 【INPUT FROM PTR】 |
| 7 | 【PROGRAM】 【E.STA】 | Select function【PROGRAM】 , PROGRAM_ flashing, No need to key in program number, directly press 【E. STA】 to execute |
| 8 | | If the program number has already exist, it displays “ALREADY EXIST SAME PROGRAM NO. LOAD OK?” Be sure to replace and press 【F0】 [YES] |
| 9 | | Select files and check the program content |

12.5 Transmit NC Code from PC to CNC

12.5.1 Transmit NC Code from PC to CNC (FANUC 0T, 0M)

| Steps | Operation Buttons | Explain |
|-------|--------------------------------------|---|
| 1 | 【EDIT】 | Change 【Mode Switch】 to 【EDIT】 mode |
| 2 | 【OFF】 | Change Edit key switch to 【ON】 position |
| 3 | O0001 【↓】 (Fixed Number) | Key in O0001, press 【↓】 to open CNC code O0001. It displays the number O0001 and its content (If not exist, please add O0001) |
| 4 | O0001 F123 N1234 or (O1234) | Edit O0001 CNC code as to fill up delivery order and dispatch list O0001 >> Fill up delivery order, designation number F123 >> Means classification folder number F123 N1234 >> Request to transmit program number O1234 Note: FANUC replace O1234 with N1234 e.g. (O1234) please key in N1234 |
| 5 | O0001 【OUTPUT】 | O0001 【OUTPUT】 transmit order to PC for delive goods |
| 6 | 【INPUT】 | Press 【INPUT】 to receive CNC code and message from PC, and it displays O0002(MESSAGE) (O1234, PROG. TRANFORE =OK) complete transmission |
| 7 | O0002 【DELETE】 | After check the CNC code, delete O0002 (Note: set parameter to the avoid this step) |
| 8 | O1234 【↓】 | Key in O1234 and press 【↓】 to open it |

Message Explain

[=OK] means execute CNC code accurately, [=?] means execute CNC code wrongly

(1) When CNC code transmit accurately, CNC controller displays:

00002(MESSAGE)
(MESSAGE)(01234, PROG. TRANFORE = OK)

(2) When the program number can't be found in PC ,CNC machine displays:

00002(MESSAGE)
(ALARM/071)(01234, PROG. NO. FOUND =?)

(3) When assign a classification folder that not exist in PC, CNC machine displays:

00002(MESSAGE)
(ALARM/001)(F123. NO. EXIST =?)

12.5.2 Transmit NC Code from PC to CNC (FANUC 18M 21T)

| Steps | Operation Buttons | Explain |
|-------|--------------------------------------|---|
| 1 | 00001 【↓】 | Key in O0001, click 【↓】 to open CNC code O0001. If not exist, please add O0001 It displays the number O0001 and its content |
| 2 | 00001 F123 N1234 or (01234) | Edit O0001 CNC code content as to fill up delivery order and dispatch list O0001 fill up delivery order, fixed number F123 classification folder number F123 N1234 request to transmit program number O1234 Note: FANUC replace O1234 with N1234 e.g. (01234) please key in N1234 |
| 3 | 00001 【PUNCH】 | O0001 【PUNCH】 transmit order to PC for delive goods Press the rightest operation button 【OPRT】 at the bottom of the screen Press operation button for details【>】until it displays【READ】 【PUNCH】 Key in O0001, and press transmission button 【PUNCH】 or click 【THIS】 to transmit editing program Press execute button 【EXEC】 or stop button 【STOP】 |
| 4 | 【READ】 | Press 【READ】 , and it displays 【EXEC】【STOP】 Press 【EXEC】 to receive CNC code and message from PC or press 【STOP】 to stop receiving CNC code and message |
| 5 | 00002 【DELETE】 | After check the CNC code, delete O0002 (Note: set parameter to the avoid this step) |
| 6 | 01234 【↓】 | Open and check CNC code O1234 |

12.5.3 Transmit NC Code from PC to CNC (MITSUBISHI)

| Steps | Operation Buttons | Explain |
|-------|--------------------------------|---|
| 1 | 0001 【INPUT】 (Fixed Number) | Key in 0001, click 【INPUT】 to open CNC code 0001 If not exist, please add O0001 It displays the number 0001 and its content |
| 2 | O0001 F123 N1234 | Edit program content as to fill up delivery order and dispatch list O0001 fill up delivery order, fixed number F123 classification folder number F123 N1234 request to transmit program number O1234 |
| 3 | 0001 【INPUT】 | Click main list【ALARM/DIAGN】on the right side of screen And press 【MENU】 at the bottom of the screen Until display 【INPUT】【OUTPUT】 Press #(1) DATA(1), #(1) means to output CNC code DATA(1) stand for program number O0001 Press 【INPUT】 to execute it |
| 4 | 【INPUT】 | Press #(1) DATA (When key in CNC code, DATA column can be ignored and not key in program number) Press 【INPUT】 to receive CNC code and message from PC |
| 5 | O0002 【DEL】 | After check the CNC code, delete O0002 |
| 6 | O1234 【INPUT】 | Key in 1234, and then press【INPUT】to open and check CNC code O1234 |

12.5.4 Transmit NC Code from PC to CNC (OKUMA U100L)

| Steps | Operation Buttons | Explain |
|-------|---|---|
| 1 | O0001 F123 N5678 | Select file O1.MIN For example: O0001 F123 N5678 |
| 2 | 【F3】【PIP】 | Press 【F3】【PIP】 at the bottom of the screen |
| 3 | 【F2】【PUNCH】 | Press 【F2】【PUNCH】 at the bottom of the screen |
| 4 | (MD1:INDEX) 【WRITE】 【Confirm to Select】 | Press (MD1: INDEX) Move the cursor on program number 01.MIN press 【WRITE】【Confirm to Select】 (Or directly key in : O1.MIN , press 【WRITE】 【Confirm】) |
| 5 | 【WRITE】【Confirm】 | Click 【WRITE】【Confirm】 to complete document transmission |
| 6 | 【F1】【READ】 | Press 【F1】【READ】 at the bottom of the screen |
| 7 | 【WRITE】【Confirm】 | Press 【WRITE】【Confirm】 at the right of the screen |
| 8 | | Complete read CNC code as O5678.MIN or A.MIN, please rename it later (Note: if it has already exist, it will display “ALREADY EXIST SAME PROGRAM NO. LOAD OK?”, please key in [Y]) |
| 9 | | Select file, and check the CNC code |

12.5.5 Transmit NC Code from PC to CNC (Brother TC-S2A)

| Steps | Operation Buttons | Explain |
|-------|------------------------------|---|
| 1 | F123 ; N5678 ; | Select function PROGRAM EDIT to edit program , number 0001 for example: F123 ; N5678 ; complete edit |
| 2 | 【EXTERNAL PROGRAM I/O】 | Select function 【EXTERNAL PROGRAM I/O】 (external program send, input/output) |
| 3 | 【OUTPUT TO PTP】 【PROGRAM】 | Select function 【OUTPUT TO PTP】 Select function 【PROGRAM】 |
| 4 | 0001 【E.STA】 | At PROGRAM_ cursor flashing spot key in program number 0001 Press 【E. STA】 to execute it |
| 5 | 【F0】 【F0】 | When complete transmit document, double press 【F0】 (RETURN MENU) (Function option , return twice) |
| 6 | 【INPUT FROM PTR】 | Select function 【INPUT FROM PTR】 |
| 7 | 【PROGRAM】 【E.STA】 | Select function 【PROGRAM】 , PROGRAM_ flashing, No need to key in program number, directly press 【E. STA】 to execute it |
| 8 | | If the program number has already exist, it displays “ALREADY EXIST SAME PROGRAM NO. LOAD OK?” Be sure to replace and click 【F0】 [YES] |
| 9 | | Select files and check CNC code |

12.6 Remote Advanced Functions

12.6.1 Transmit NC Code (continuous number) from PC to CNC

| Steps | Operation Buttons | Explain |
|-------|------------------------------|--|
| 1 | 【EDIT】 | Switch to 【EDIT】 mode |
| 2 | 【OFF】 | Open edit key by switch the key to 【ON】 |
| 3 | O0001 【↓】 (Fixed number) | Key in O0001, press【↓】to open CNC code O0001. It displays the number O0001 and its content |
| 4 | O0001 F006 N1011 N1015 | Edit CNC code as to fill up delivery order and dispatch list O0001 fill up delivery order, fixed number F006 classification folder number F006 N1011 N1015 request to transmit all exist CNC codes number between O1011 and O1015 |
| 5 | O0001 【OUTPUT】 | O0001 【OUTPUT】 transmit order to PC for deliver goods |
| 6 | 【INPUT】 | Press 【INPUT】 to receive CNC code and message from PC |
| 7 | O0002 【DELETE】 | After check the CNC code, delete O0002 (Note: set parameter to avoid this step) |
| 8 | O1011 【↓】 | Open and check O1011 |

Message Explain

[=OK] means execute CNC code accurately, [=?] means execute CNC code wrongly

(1) When CNC code transmit accurately, CNC controller displays:

00002(MESSAGE)
(MESSAGE)(01011, PROG. TRANSMIT =OK)
(MESSAGE)(01012, PROG. TRANSMIT =OK)
(MESSAGE)(01013, PROG. TRANSMIT =OK)
(MESSAGE)(01014, PROG. TRANSMIT =OK)
(MESSAGE)(01015, PROG. TRANSMIT =OK)

(2) When program number commanded isn't valid in PC, CNC machine displays:

00002(MESSAGE)
(ALARM/071)(01011. 01015. PROG. NO. VALID =?)

(3) Transmit all CNC codes in the folder to CNC

| | |
|----------|---|
| 00001 | |
| F006 | N1 N9999 |
| N1 N9999 | Request to transmit all existed programs number between 00001 and 09999 |

12.6.2 Transmit Folder list from PC to CNC

| Steps | Operation Buttons | Explain |
|-------|-----------------------------|---|
| 1 | 【EDIT】 | Change to 【EDIT】 mode |
| 2 | 【OFF】 | Open edit key by switch the key to 【ON】 |
| 3 | O0001 【↓】 (Fixed number) | Key in O0001, press【↓】to open CNC code O0001. It displays the number O0001 and its content |
| 4 | O0001 F006 / / | Edit CNC code as to fill up delivery order and dispatch list O0001 fill up delivery order, fixed number F006 classification folder number F006 / request to transmit file content in the folder this example is for file content in folder F006 |
| 5 | O0001 【OUTPUT】 | O0001 【OUTPUT】 transmit order to PC for deliver goods |
| 6 | 【INPUT】 | Press 【INPUT】 to receive CNC code and message from PC |
| 7 | O0002 【DELETE】 | After check the CNC code , delete O0002 (Note: set parameter to the avoid this step) |
| 8 | O0003 【↓】 | Displays CNC code O0003, O0003 is the CNC code specific to folder |
| 9 | O0003 【DELETE】 | After check the CNC code, delete O0003 |

12.6.3 Transmit folder list (fit to additional conditions) from PC to CNC

| Steps | Operation Buttons | Explain |
|-------|-----------------------------|---|
| 1 | 【EDIT】 | Switch to 【EDIT】 mode |
| 2 | 【OFF】 | Open edit key by switch the key to 【ON】 |
| 3 | O0001 【↓】 (Fixed number) | Key in O0001, press【↓】to open CNC code O0001. It displays the number O0001 and its content |
| 4 | O0001 F006 /(ER16) | Edit CNC code as to fill up delivery order and dispatch list O0001 fill up delivery order, fixed number F006 classification folder number F006 /(ER16) write conditions after sign “/”, this example is the files with ER16 in file name |
| 5 | O0001 【OUTPUT】 | O0001 【OUTPUT】 transmit order to PC for deliver goods |
| 6 | 【INPUT】 | Press 【INPUT】 to receive CNC code and message from PC |
| 7 | O0002 【DELETE】 | After check the CNC code, delete O0002 (Note: Set parameter to the avoid this step) |
| 8 | O0003 【↓】 | Displays CNC code O0003, O0003 is the CNC code specific to folder |
| 9 | O0003 【DELETE】 | After check the CNC code, delete O0003 |

12.6.4 Transmit folder list (fit to additional conditions) from PC to CNC

| Steps | Operation Buttons | Explain |
|-------|--------------------------------|--|
| 1 | 【EDIT】 | Change to 【EDIT】 mode |
| 2 | 【OFF】 | Open edit key by switch the key to 【ON】 |
| 3 | O0001 【↓】 (Fixed number) | Key in O0001, press【↓】to open CNC code O0001. It displays the number O0001 and its content |
| 4 | O0001 F006 /(MT4) S99 | Edit CNC code as to fill up delivery order and dispatch list O0001 fill up delivery order, fixed number F006 classification folder number F006 /(MT4) “/” stands for content, assume “MT4” as search condition S99 S99 is the fixed code |
| 5 | O0001 【OUTPUT】 | O0001 【OUTPUT】 transmit order to PC for delive goods |
| 6 | 【INPUT】 | Press 【INPUT】 to receive CNC code and message from PC |
| 7 | O0002 【DELETE】 | After check the CNC code, delete O0002 |

(S99. NO. FILE. AVAILABE= ?) (No available file to transmit)

12.6.5 Transmit NC Code (last number of search) from PC to CNC

| Steps | Operation Buttons | Explain |
|-------|-----------------------------|--|
| 1 | 【EDIT】 | Change to 【EDIT】 mode |
| 2 | 【OFF】 | Open edit key by switch the key to 【ON】 |
| 3 | O0001 【↓】 (Fixed number) | Key in O0001, press 【↓】 to open program O0001. It displays the number O0001 and its content |
| 4 | O0001 F006 /N99 | Edit CNC code as to fill up delivery order and dispatch list O0001 fill up delivery order, fixed number F006 classification folder number F006 /N99 add N99 after sign "/" request to transmit last program number in the folder |
| 5 | O0001 【OUTPUT】 | O0001 【OUTPUT】 transmit order to PC for delive goods |
| 6 | 【INPUT】 | Press 【INPUT】 to receive program and message from PC |
| 7 | O0002 【DELETE】 | After check the program, delete O0002 (Note: Set parameter to the avoid this step) |
| 8 | O0003 【↓】 | Displays CNC code O0003, O0003 is the CNC code specific to folder |
| 9 | O0003 【DELETE】 | After check the CNC code, delete O0003 |

Message Explain

When programs transmit accurately, CNC controller displays:

00003 (CONTENTS)

(01231.)

(01233.)

(01234.)

00002 (MESSAGE)

(LAST NUMBER = 01234)

12.6.6 Transmit folder list from PC to CNC

| Steps | Operation Buttons | Explain |
|-------|-----------------------------|---|
| 1 | 【EDIT】 | Change to 【EDIT】 mode |
| 2 | 【OFF】 | Open edit key by switch the key to 【ON】 |
| 3 | 00001 【↓】 (Fixed number) | Key in O0001, press【↓】to open CNC code O0001. It displays the number O0001 and its content |
| 4 | 00001 F006 // | Edit program content as to fill up delivery order and dispatch list O0001 fill up delivery order, fixed number F006 means classification folder number F006 // request to transmit subsidiary folder content in the folder |
| 5 | 00001 【OUTPUT】 | O0001 【OUTPUT】 transmit order to PC for delive goods |
| 6 | 【INPUT】 | Press 【INPUT】 to receive program and message from PC |
| 7 | 00002 【DELETE】 | After check the CNC code, delete O0002 (Note: Set parameter to the avoid this step) |
| 8 | 00003 【↓】 | Displays CNC code O0003, O0003 is the CNC code specific to file content |
| 9 | 00003 【DELETE】 | After check the CNC code, delete O0003 |

12.7 Remote Deletion and Assigned File Name Functions

12.7.1 【DELETE】 Program File Number in PC

| Steps | Operation Buttons | Explain |
|-------|-----------------------------|--|
| 1 | O0001 【↓】 (Fixed number) | Key in O0001, press【↓】to open CNC code O0001. It displays the number O0001 and its content |
| 2 | O0001 F005 D1234 | Edit CNC code as to fill up delivery order and dispatch list O0001 fill up delivery order, fixed number F005 classification folder number F005 D1234 delete CNC code O1234, D is the abbreviation of 【DELETE】 |
| 3 | O0001 【OUTPUT】 | O0001 【OUTPUT】 transmit order to PC for delive goods |
| 4 | 【INPUT】 | Press 【INPUT】 to receive CNC code and message from PC, and it displays (MESSAGE)(O1234, DELETE =OK) complete transmission |

12.7.2 【Rename】 program number in PC

| Steps | Operation Buttons | Explain |
|-------|------------------------------|---|
| 1 | O0001 【↓】 (Fixed number) | Key in O0001, press 【↓】 to open CNC code O0001. It displays the number O0001 and its content |
| 2 | O0001 F005 R1234 R5678 | Edit CNC code as to fill up delivery order and dispatch list O0001 fill up delivery order, fixed number F005 classification folder number F005 R1234 original program number O1234 R5678 rename to program number O5678 |
| 3 | O0001 【OUTPUT】 | O0001 【OUTPUT】 transmit order to PC for delive goods |
| 4 | 【INPUT】 | Press 【INPUT】 to receive CNC code and message from PC, and it displays (MESSAGE)(O1234. TO. O5678. RENAME =OK) complete transmission |

12.7.3 NC Code from CNC to PC (Save as another file name)

| Steps | Operation Buttons | Explain |
|-------|---|--|
| 1 | 【EDIT】 | Change to 【EDIT】 mode |
| 2 | 【OFF】 | Open edit key by switch the key to 【ON】 |
| 3 | 01234 【↓】 | Key in O1234, press【↓】to open CNC code O1234. It displays the number O1234 and its content |
| 4 | Move the cursor Edit the program ([BT50ET16]) | O1234 (DEMO TEST) N1 G0 X120. Z120. T0 (F006) G96 S180 M3 ([BT50ET16]) Key in complete file name in the third line in the CNC code Save O1234 in folder F006 with file name BT50ET16 |
| 5 | 01234 【OUTPUT】 | O1234 【OUTPUT】 to transmit CNC code to PC to save |
| 6 | 【INPUT】 | Press 【INPUT】 to receive CNC code and message from PC |
| 7 | 00002 【DELETE】 | After check the CNC code , delete O0002 |

Message Explain

- (1) When PC receive the CNC code accurately, CNC machine displays:
00002 (MESSAGE)
(MESSAGE) (BT50ER16, PROG, RECEIVE =OK)(Complete receive and save)
- (2) When no assign classification folder, CNC machine displays:
00002 (MESSAGE)
(ALARM/111) (BT50ER16, NUMBER SPECIFIC. F*** =?)(not yet assign classification folder when save the program)
- (3) When assign classification content not exist in PC, CNC machine displays:
00002(MESSAGE)
(ALARM/001) (BT50ER16) (ABC123, NUMBER EXIST =?)
- (4) If the program number has already exist in PC, CNC machine displays:
00002(MSEEAGE)
(ALARM/001)(BT50ER16, PROG. ALREADY. EXIST=?) (the program number has already exist)
(PLEASE CHECK)
- (5) To replace the old one when program number has already exist in PC, key in sign “/” in the second line in the CNC code
e.g. 01234 (DEMO TEST)
N1 G0 X120. Z120. T0 F006 /

12.7.4 Transmit NC Code from PC to CNC

12.7.5 ([Assign complete path]) → (Assign complete file name) **last method**

| Steps | Operation Buttons | Explain |
|-------|---|---|
| 1 | 【EDIT】 | Change to 【EDIT】 mode |
| 2 | 【OFF】 | Open edit key by switch the key to 【ON】 |
| 3 | O0001 【↓】 (Fixed number) | Key in O0001, press 【↓】 to open CNC code O0001. It displays the number O0001 and its content |
| 4 | Edit the program O0001 ([ABC123]) (BT50ER16) | Edit CNC code as to fill up delivery order and dispatch list O0001>>Fill up delivery order, fixed number ([ABC123])>>Switch to folder number ABC123 (BT50ER16)>>Request to transmit program BT50ER16 |
| 5 | O0001 【OUTPUT】 | O0001 【OUTPUT】 transmit order to PC for delive goods |
| 6 | 【INPUT】 | Press 【INPUT】 to receive CNC code and message from PC |
| 7 | O0002 【DELETE】 | After check the CNC code , delete O0002 |
| 8 | O1234 【↓】 | Open and check O1234 |

Message Explain

(1) When transmit CNC code accurately, CNC machine displays:

00002(MESSAGE)

(MESSAGE) (BT50ER16, PROG. TRANSMIT =OK)

(2) When the program number can't be found in PC, CNC machine displays:

00002(MESSAGE)

(ALARM/071)(BT50ER16, PROG. NUMBER FOUND =?)

(3) Parentheses stand for file name, for example (BT50ER16)

Parentheses plus brackets stand for folder, for example ([ABC123])

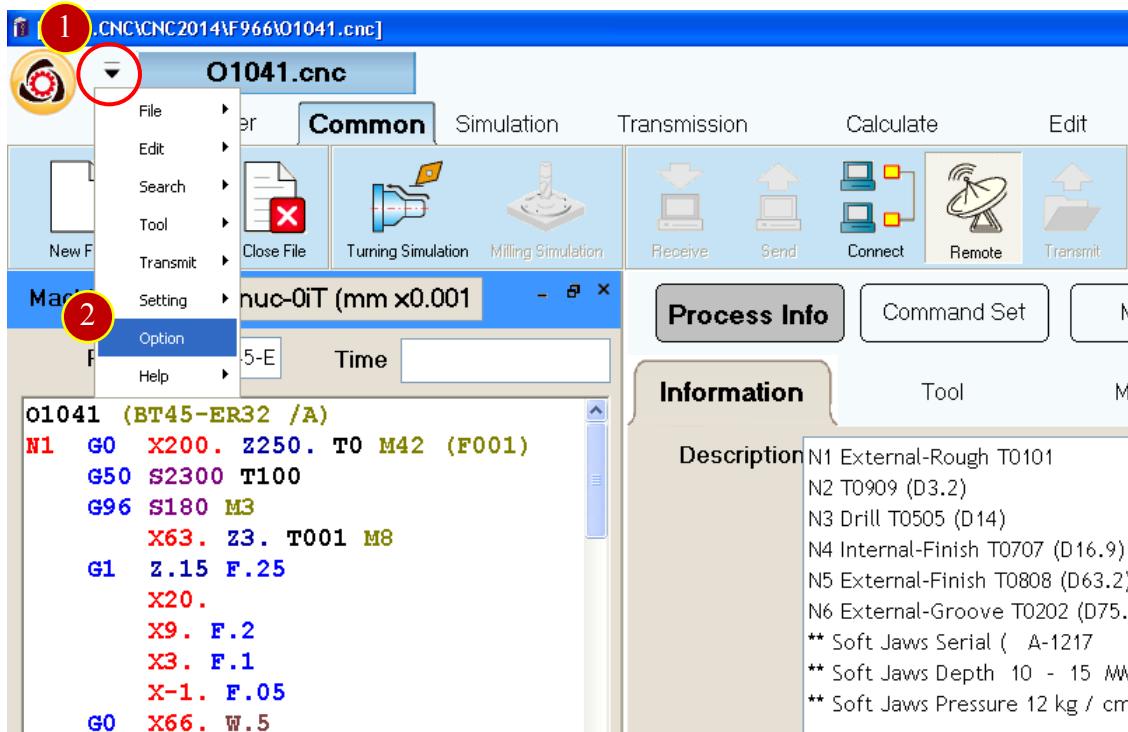
13. Software System Settings

13.1 Debug Mode Settings

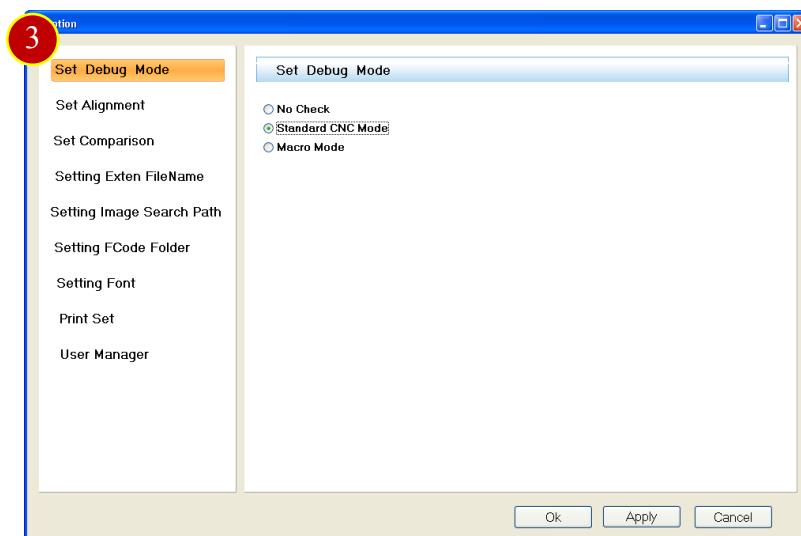
The mode used to debug CNC code

13.1.1 Open Setting Options

- (1) Click pull-down main list
- (2) Click 【Option】

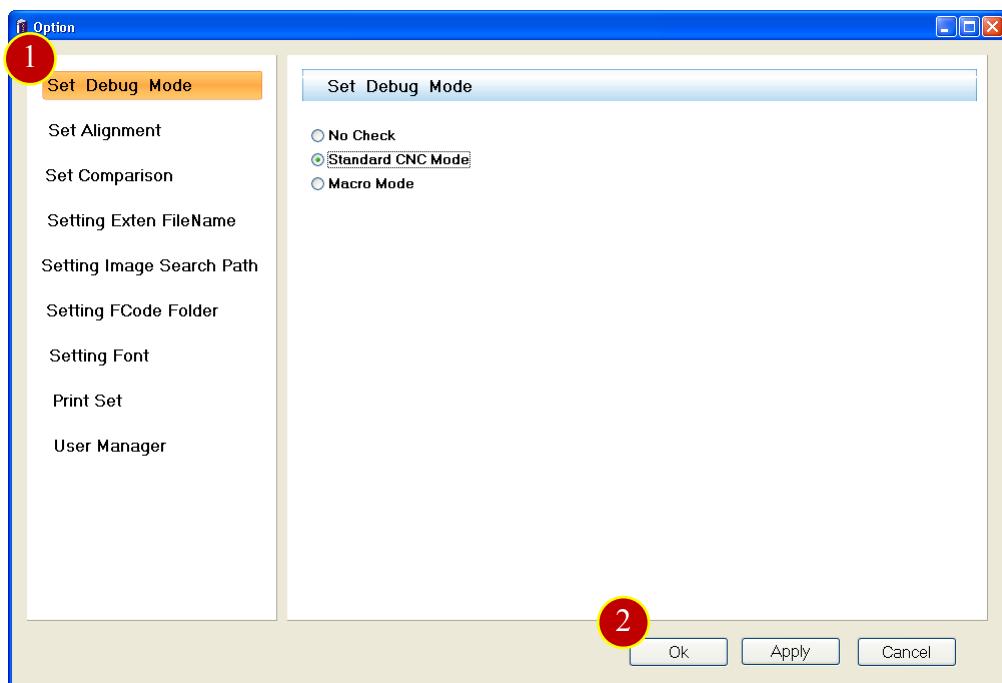


- (3) Click 【Set Debug Mode】



13.1.2 Set Debug Mode Steps

- (1) Check options in 【Set Debug Mode】
 - (a) No Check: Stop system checking
 - (b) Standard CNC Mode: Let system check wrong CNC codes for you
 - (c) Macro Mode: Let system check wrong MACRO variables and CNC codes for you
 - (d) Permanent settings: Be sure and check permanent settings to modify the settings of machine; do not check and keep original settings
- (2) Complete setting and click 【Apply】 > 【OK】 to action

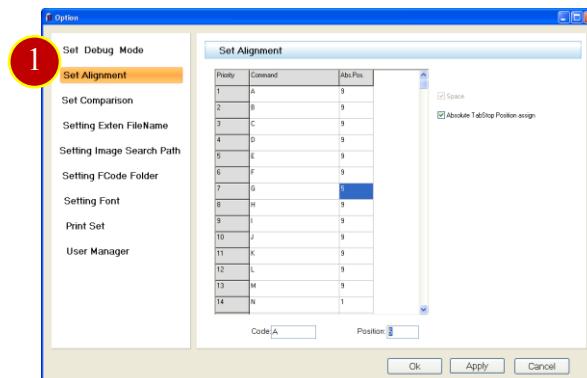


13.2 Alignment Settings

Set ways of alignment of each character in CNC code

13.2.1 Open Settings Options

(1) Click 【Set Alignment】



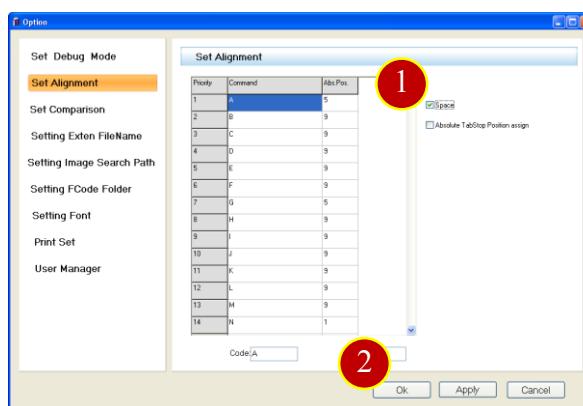
13.2.2 Space

There will be a blank between codes when use space to align

Settings Steps:

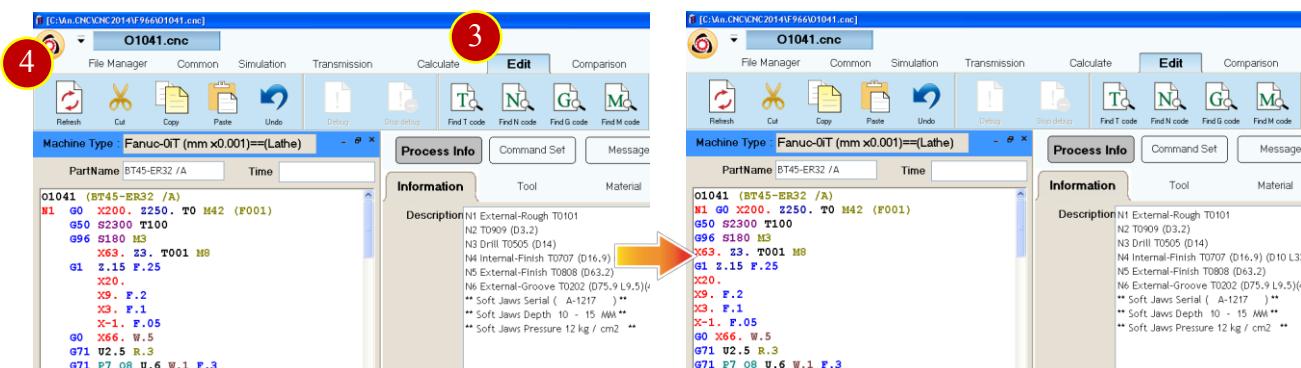
(1) Check 【Space】

(2) Click 【OK】 to save settings



(3) Click 【Edit】 to switch functions

(4) Click 【Refresh】 to reallocate code position



13.2.3 Absolute Tabstop Position Assign

System reallocate programs according to the position of each code when use Absolute Tabstop Position Assign to align

(1) Check 【Absolute Tabstop Position Assign】

(2) Set each code

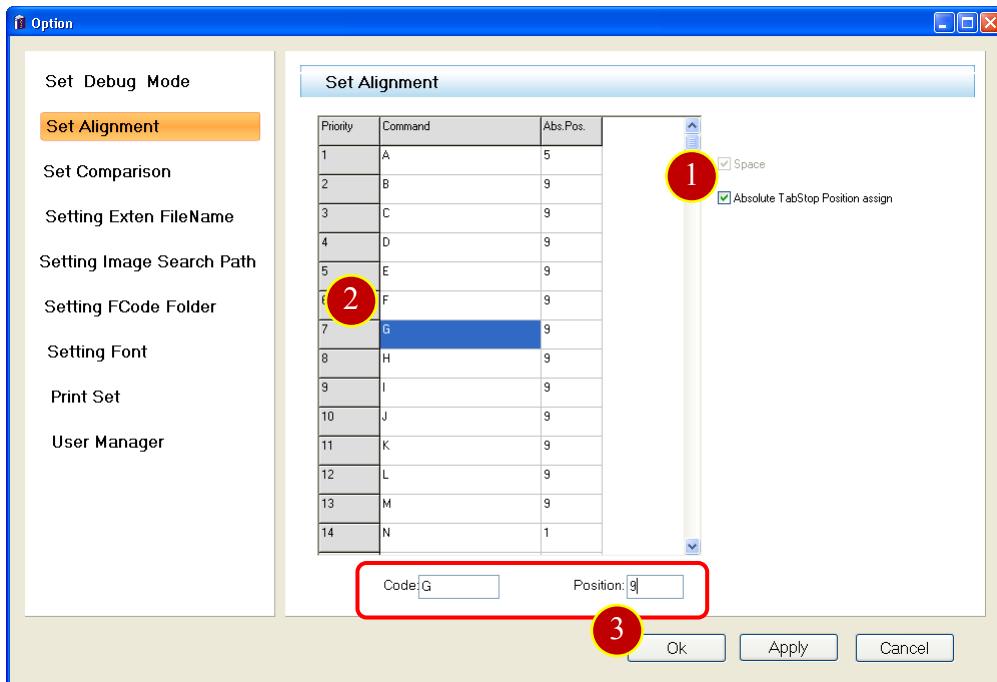
e.g. set code G with 9 blanks

(a) Select code G

(b) Code and position column will display the value that currently set

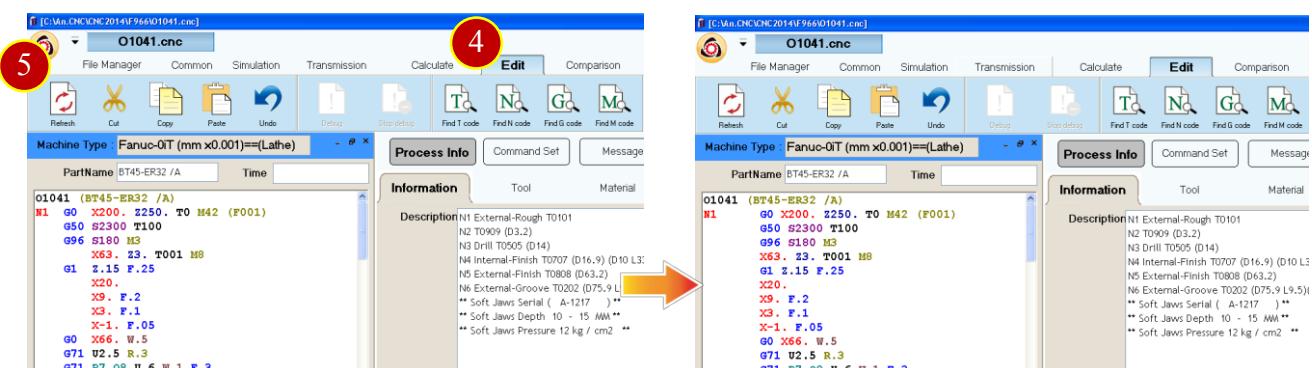
(c) Key in 9 in position column

(3) After complete settings, click 【OK】 to save settings



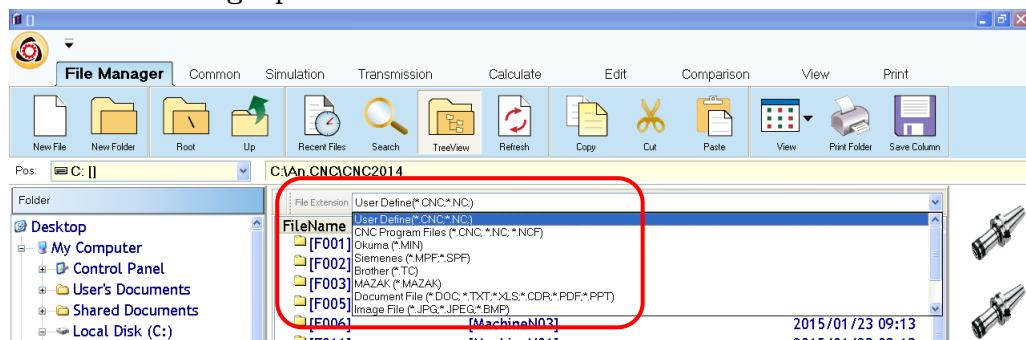
(4) Click 【Edit】 to switch functions

(5) Click 【Refresh】 to reallocate code position



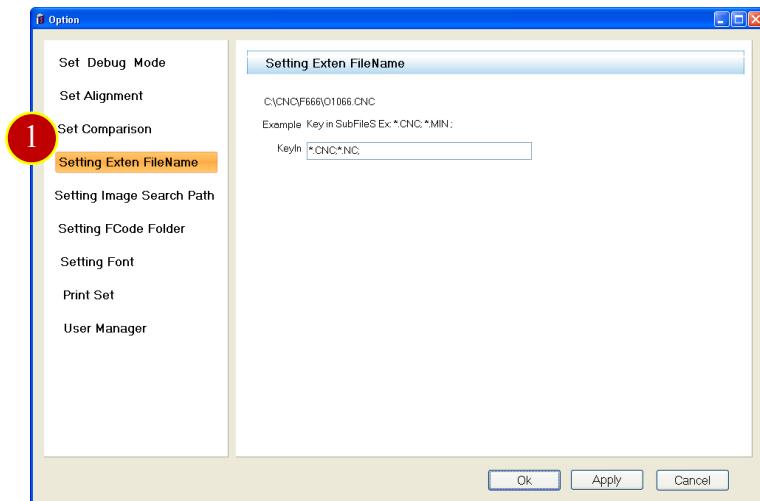
13.3 Extend Files Settings

User self-setting option of file name extension



13.3.1 Open Settings Options

(1) Click 【Setting Exten File Name】

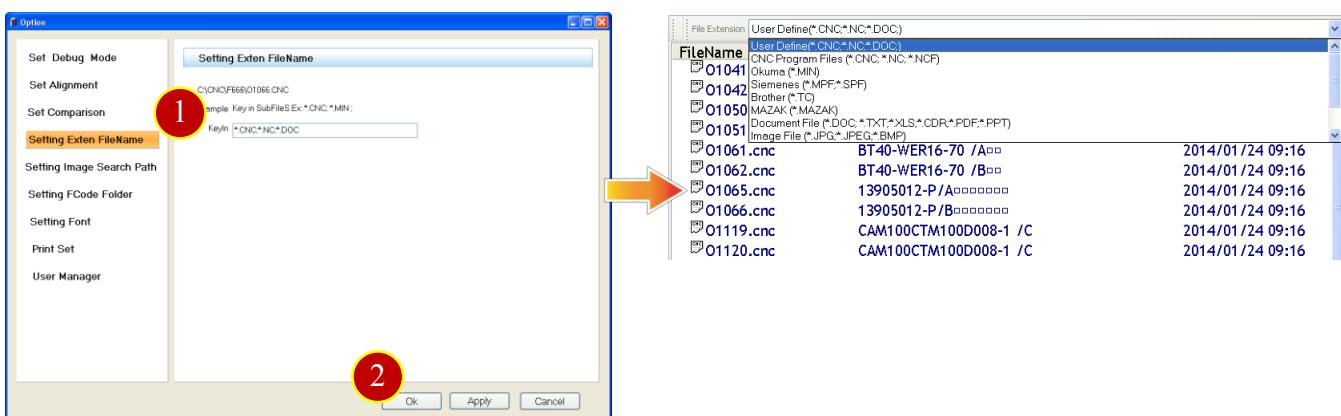


13.3.2 File Name Extension Settings Steps

(1) Key in File Name Extension to display at NcEditor Files Manager

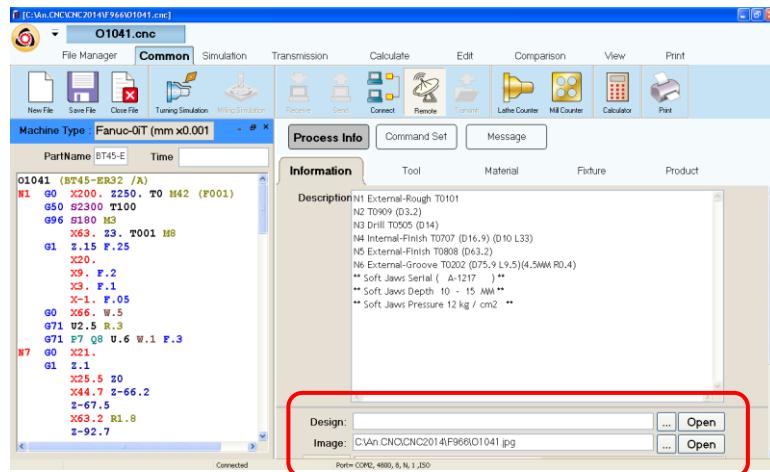
e.g. *.CNC; *.NC; *.DOC and use semicolon (;) to separate each

(2) Click 【OK】



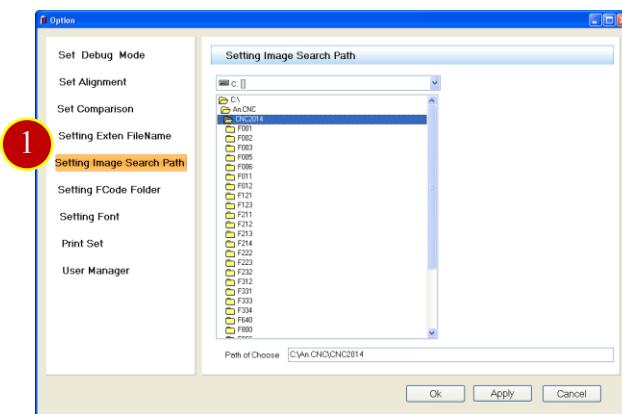
13.4 Image Search Path Settings

Set default image path when click [...] in the Information of Process Info



13.4.1 Open Settings Options

(1) Click [Setting Image Search Path]



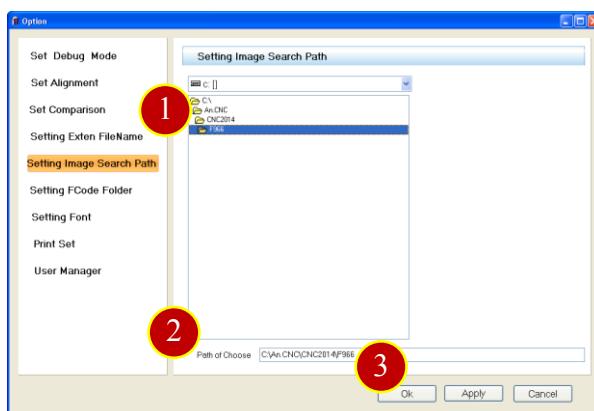
13.4.2 Setting Image Search Path Steps

(1) Click the path

(e.g. C:\An.CNC\CNC2014\F966)

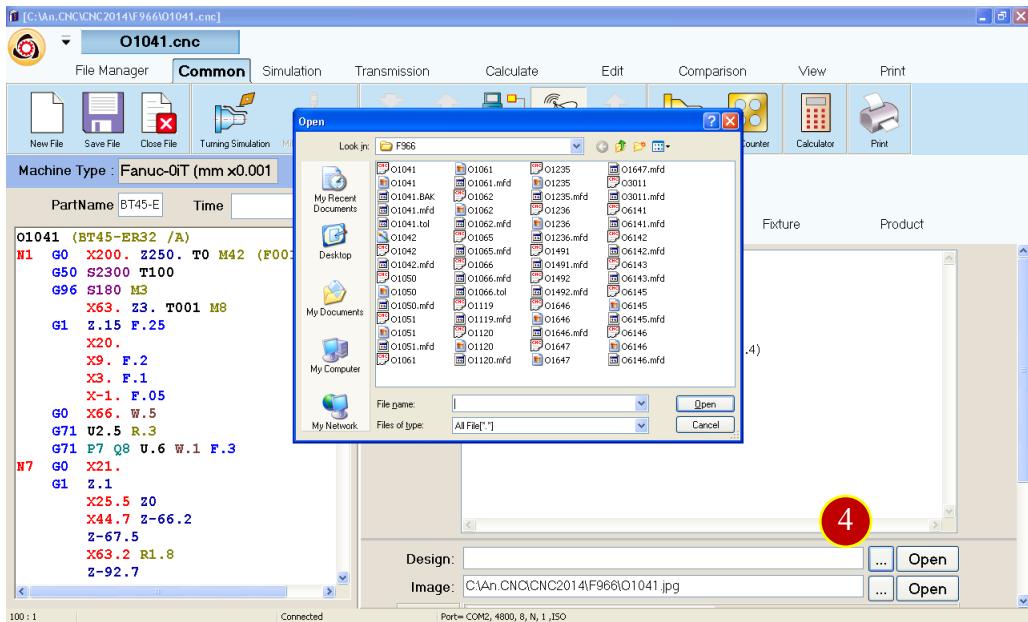
(2) After click, it will display at path of choose column

(3) Click [OK]



(4) Click [...] button right of design, and window display the path

C:\AN.CNC\CNC2014\F966

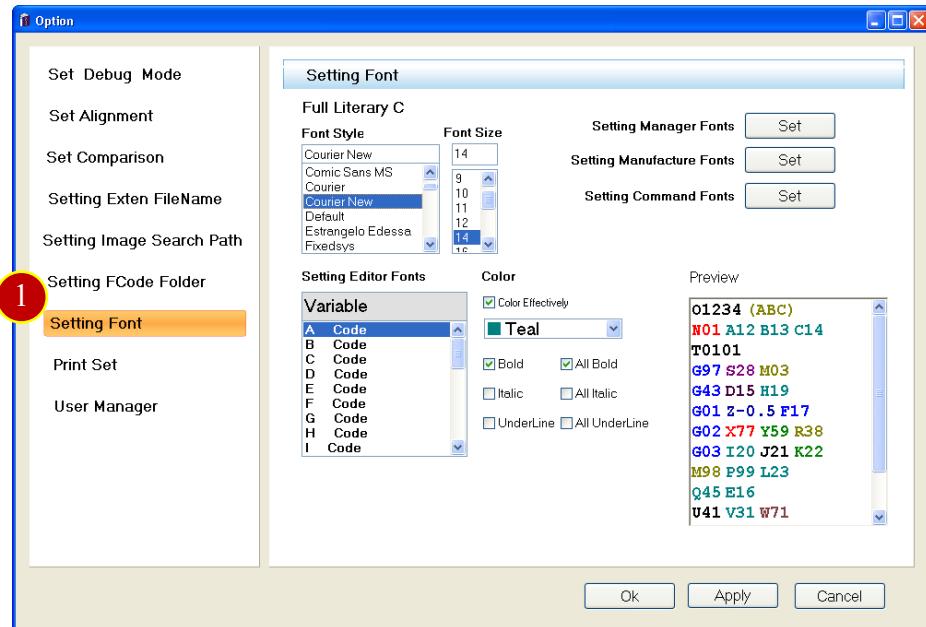


13.5 Setting Font

Set font of CNC code, editor, file manager, process description and command

13.5.1 Open Settings Options

(1) (1) Click 【Setting Font】

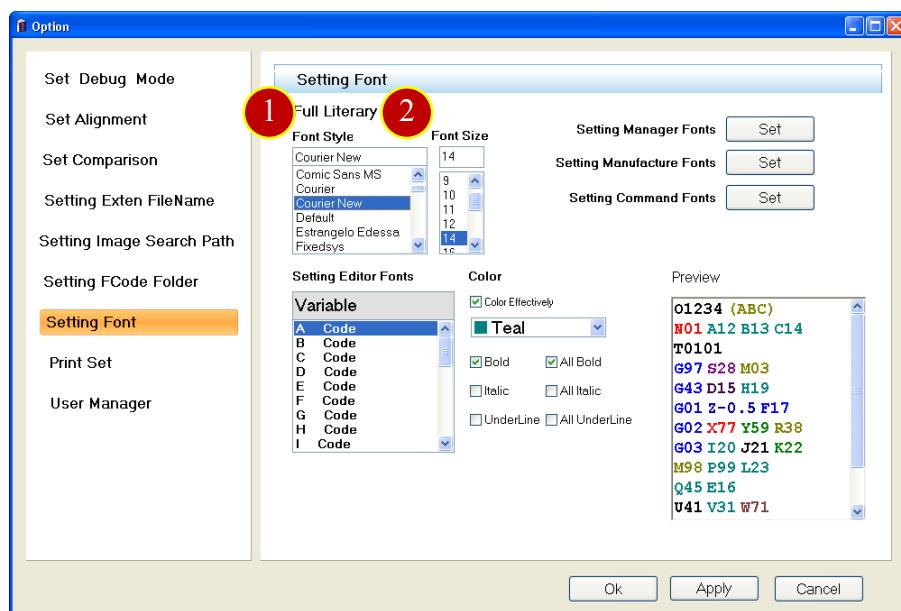


13.5.2 CNC code

Set font, size, color, style and underline in CNC code

(1) Font Style: select style at column

(2) Font Size: select size at column



(3) Color of code

Setting Steps for change color of each code:

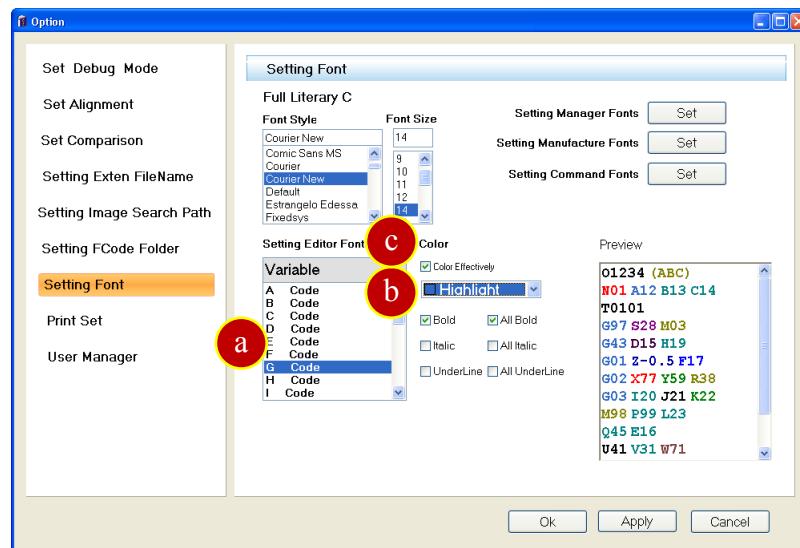
(a) Click a code

e.g. G code

(b) Click pull-down list to select color

e.g. Highlight

(c) Check 【Color Effectively】(or color won't modify after setting)

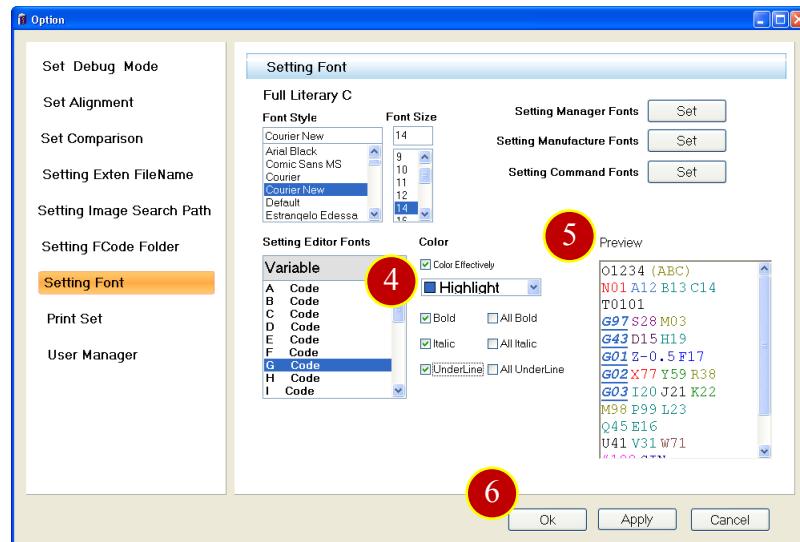


(4) Style and Underline of code

Separately set each code as bold, italic, and underline; set all bold, all italic, and all underline

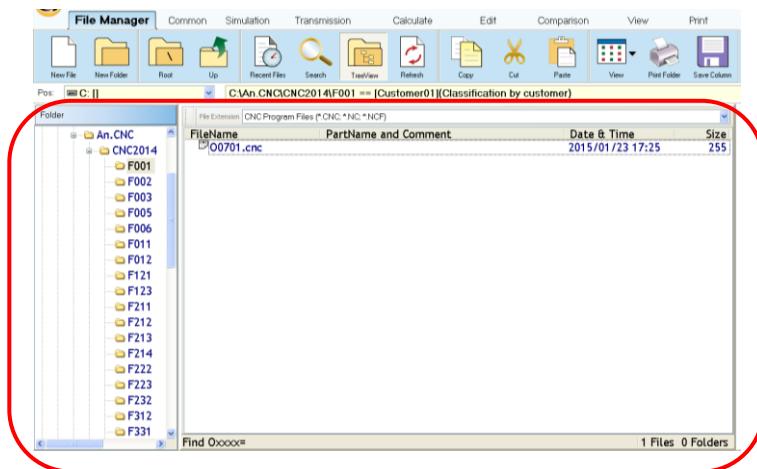
(5) After setting, check it at preview area

(6) Click 【OK】 to apply settings



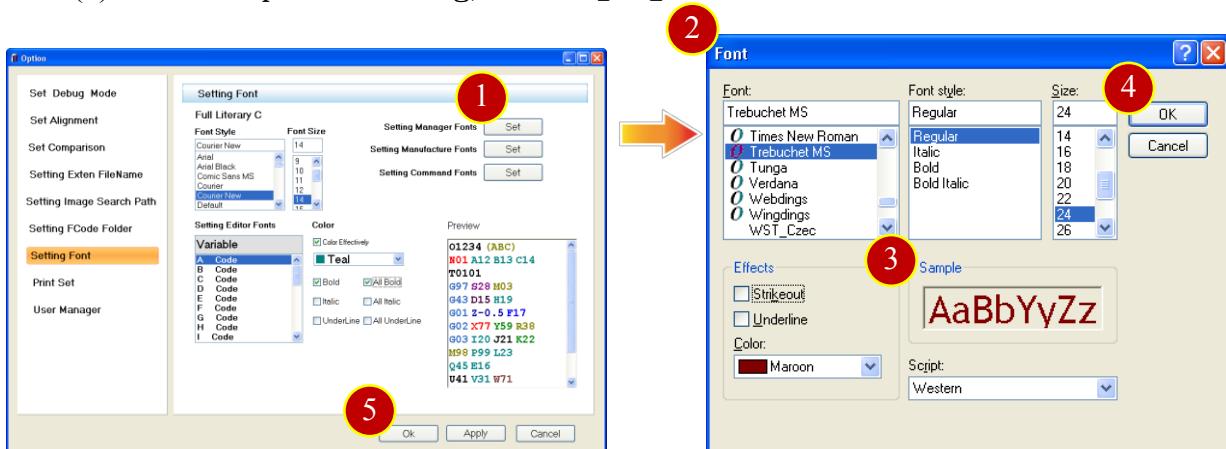
13.5.3 Font of File Manager

Set font, size, and color of folders and file list in file manager

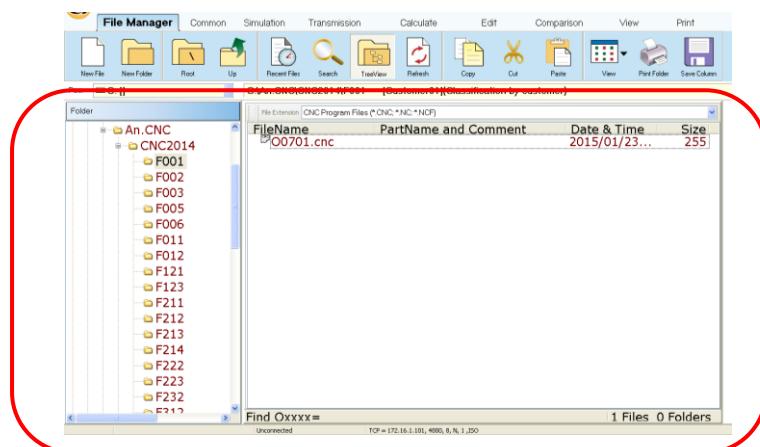


Setting Steps:

- (1) Click **【Set】** button right of Setting Manager Fonts
- (2) Set font after pop out font setting window
- (3) Below the example will preview the effect of settings
- (4) After complete setting, click **【OK】**

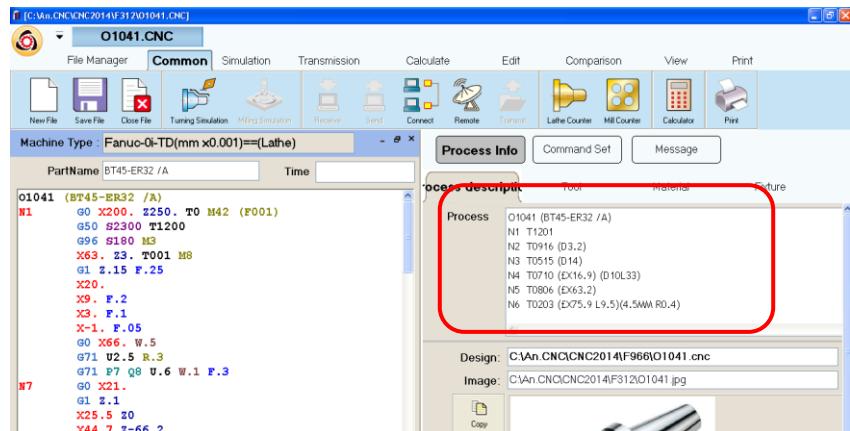


- (5) Click **【OK】** and close setting window to see actual effect of settings



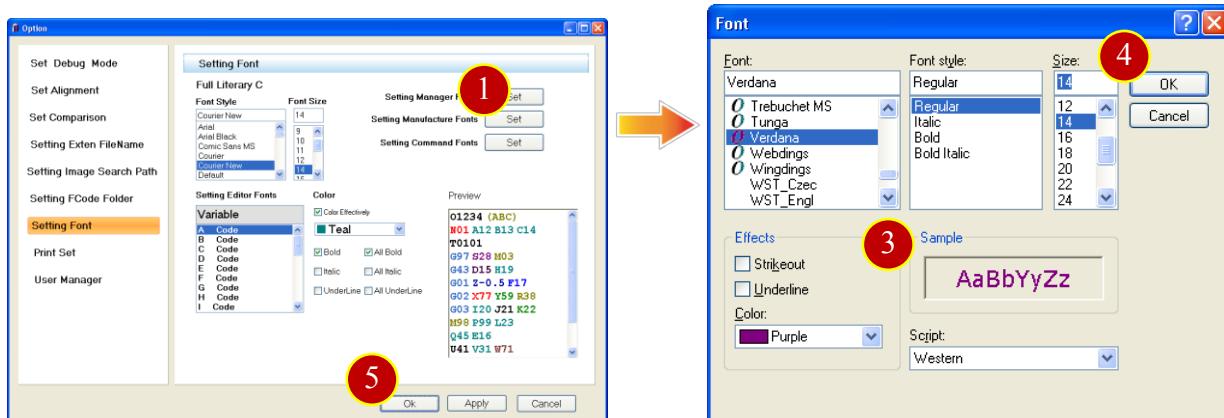
13.5.4 Font of Process Description

Set font, size, style, and color of process, tool, material, fixture, and product

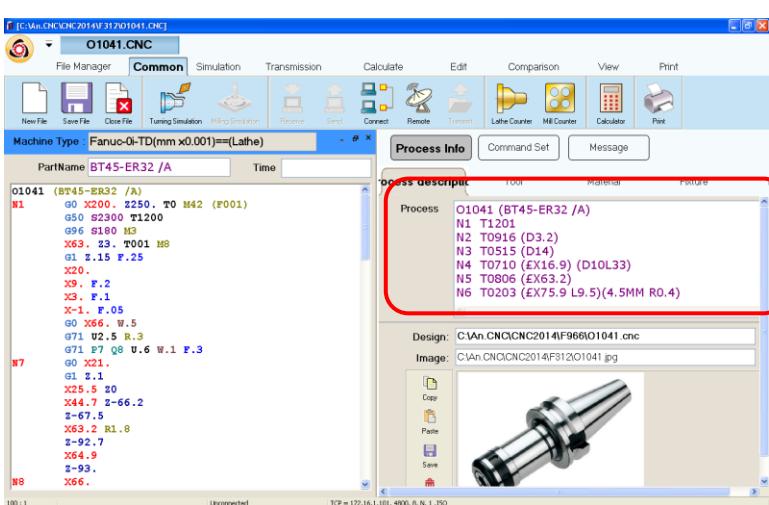


Setting Steps:

- (1) Click 【Set】 right of Setting Manufacture Fonts
- (2) Set font after pop out font setting window
- (3) Below the example will preview the effect of settings
- (4) After complete setting, click 【OK】

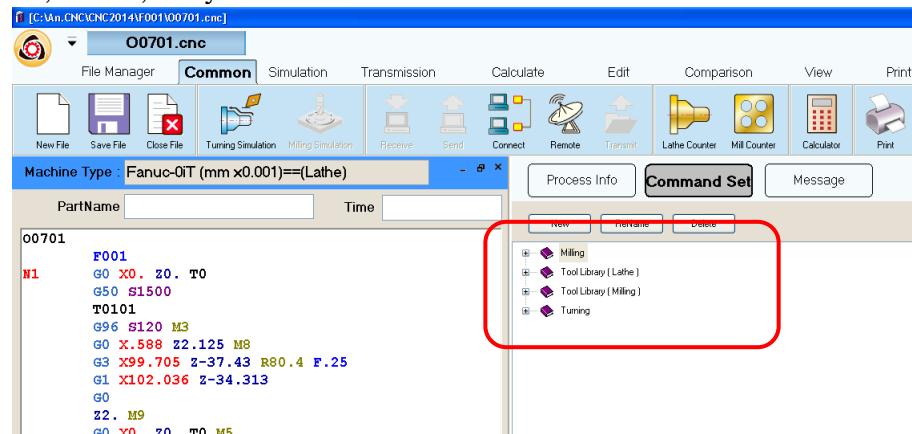


- (5) Click 【OK】 and close setting window to see actual effect of settings



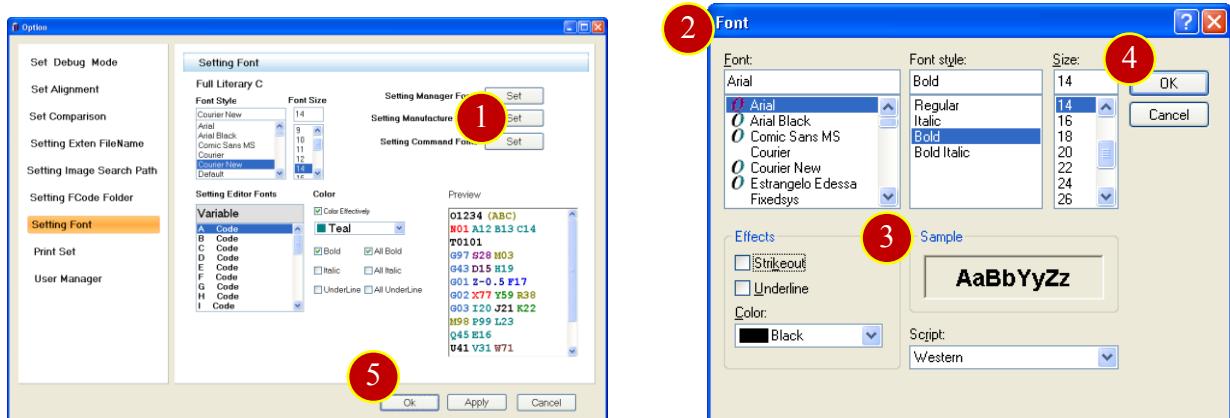
13.5.5 Font of Command

Set font, size, style and color of tree view in command

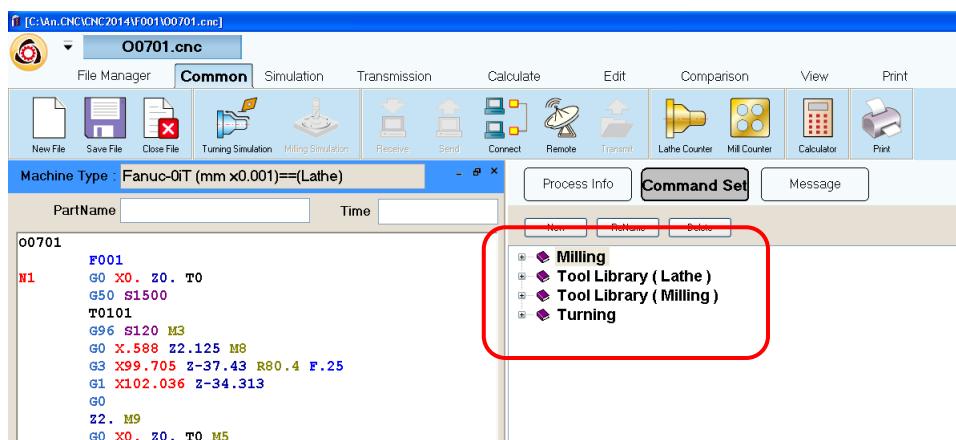


Setting Steps:

- (1) Click **【Set】** right of Setting Command Fonts
- (2) Set font after pop out font setting window
- (3) Below the example will preview the effect of settings
- (4) After complete setting, click **【OK】**



- (5) Click **【OK】** and close setting window to see actual effect of settings

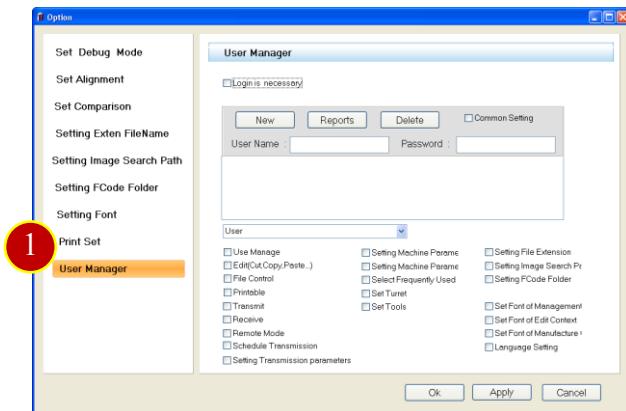


13.6 User Manage

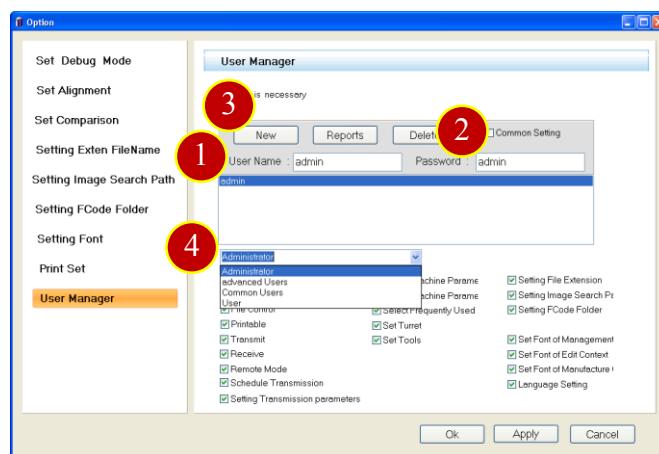
Authorized the user permissions in NcEditor

13.6.1 Open Setting Interface

(1) Click 【User Manager】



13.6.2 Add new users



(1) Key in user name (e.g. Admin)

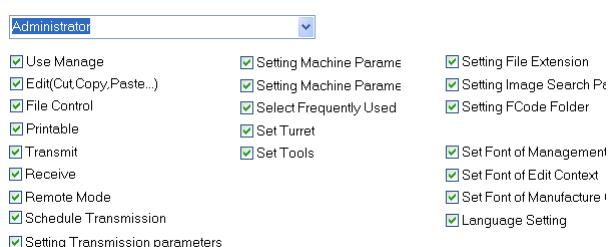
(2) Key in the login password (e.g. admin)

(3) Click 【New】

(4) Set the user permissions by clicking the drop-down list

Items in the drop-down lists are divided into four levels

(a) Administrator: Own the limits with all access



(b) Advanced Users: All access except manager

advanced Users

| | | |
|---|--|---|
| <input type="checkbox"/> Use Manage | <input checked="" type="checkbox"/> Setting Machine Param | <input checked="" type="checkbox"/> Setting File Extension |
| <input checked="" type="checkbox"/> Edit(Cut,Copy,Paste...) | <input checked="" type="checkbox"/> Setting Machine Param | <input checked="" type="checkbox"/> Setting Image Search Pt |
| <input checked="" type="checkbox"/> File Control | <input checked="" type="checkbox"/> Select Frequently Used | <input checked="" type="checkbox"/> Setting FCode Folder |
| <input checked="" type="checkbox"/> Printable | <input checked="" type="checkbox"/> Set Turret | <input checked="" type="checkbox"/> Set Font of Management |
| <input checked="" type="checkbox"/> Transmit | <input checked="" type="checkbox"/> Set Tools | <input checked="" type="checkbox"/> Set Font of Edit Context |
| <input checked="" type="checkbox"/> Receive | | <input checked="" type="checkbox"/> Set Font of Manufacture ! |
| <input checked="" type="checkbox"/> Remote Mode | | <input checked="" type="checkbox"/> Language Setting |
| <input checked="" type="checkbox"/> Schedule Transmission | | |
| <input checked="" type="checkbox"/> Setting Transmission parameters | | |

(c) Users: All access except transmission and manage

Common Users

| | | |
|---|--|---|
| <input type="checkbox"/> Use Manage | <input checked="" type="checkbox"/> Setting Machine Param | <input checked="" type="checkbox"/> Setting File Extension |
| <input checked="" type="checkbox"/> Edit(Cut,Copy,Paste...) | <input checked="" type="checkbox"/> Setting Machine Param | <input checked="" type="checkbox"/> Setting Image Search Pt |
| <input checked="" type="checkbox"/> File Control | <input checked="" type="checkbox"/> Select Frequently Used | <input checked="" type="checkbox"/> Setting FCode Folder |
| <input checked="" type="checkbox"/> Printable | <input checked="" type="checkbox"/> Set Turret | <input checked="" type="checkbox"/> Set Font of Management |
| <input type="checkbox"/> Transmit | <input checked="" type="checkbox"/> Set Tools | <input checked="" type="checkbox"/> Set Font of Edit Context |
| <input type="checkbox"/> Receive | | <input checked="" type="checkbox"/> Set Font of Manufacture ! |
| <input type="checkbox"/> Remote Mode | | <input checked="" type="checkbox"/> Language Setting |
| <input type="checkbox"/> Schedule Transmission | | |
| <input type="checkbox"/> Setting Transmission parameters | | |

(d) Custom Users: Customize the access settings by the administrator

User

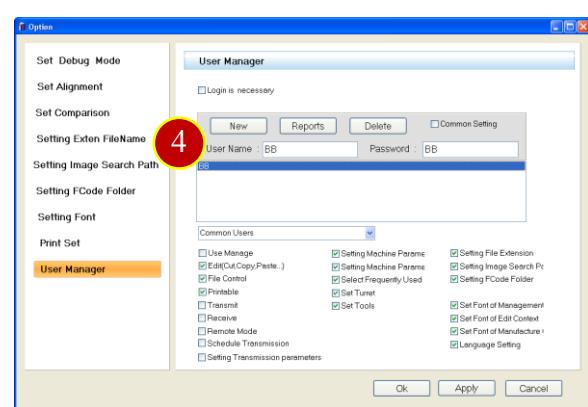
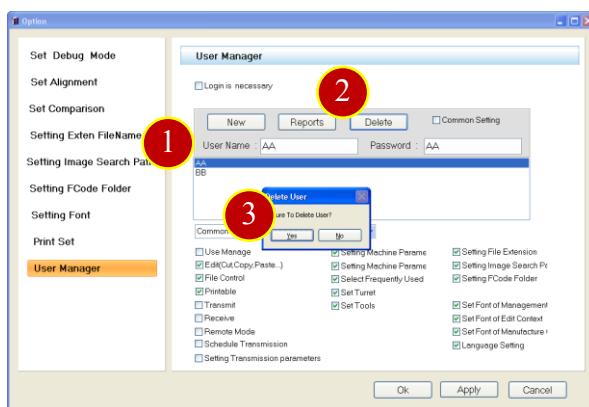
| | | |
|--|---|--|
| <input type="checkbox"/> Use Manage | <input type="checkbox"/> Setting Machine Param | <input type="checkbox"/> Setting File Extension |
| <input type="checkbox"/> Edit(Cut,Copy,Paste...) | <input type="checkbox"/> Setting Machine Param | <input type="checkbox"/> Setting Image Search Pt |
| <input type="checkbox"/> File Control | <input type="checkbox"/> Select Frequently Used | <input type="checkbox"/> Setting FCode Folder |
| <input type="checkbox"/> Printable | <input type="checkbox"/> Set Turret | <input type="checkbox"/> Set Font of Management |
| <input type="checkbox"/> Transmit | <input type="checkbox"/> Set Tools | <input type="checkbox"/> Set Font of Edit Context |
| <input type="checkbox"/> Receive | | <input type="checkbox"/> Set Font of Manufacture ! |
| <input type="checkbox"/> Remote Mode | | <input type="checkbox"/> Language Setting |
| <input type="checkbox"/> Schedule Transmission | | |
| <input type="checkbox"/> Setting Transmission parameters | | |

(Each limits of four levels can be decided by the administrator)

(5) Select the administrator and finish the setting

13.6.3 Delete Users

- (1) Click the user you want to delete
- (2) Click 【Delete】
- (3) Click 【Yes】
- (4) Finish



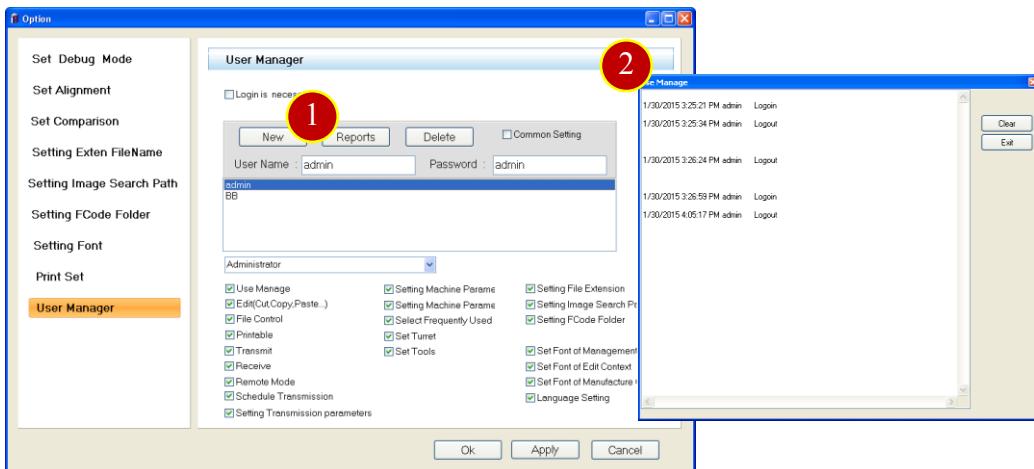
13.6.4 Users Report

Inspect login, logout time records of all users

Steps:

(1) Click 【Report】

(2) You can check the records by user manager interface



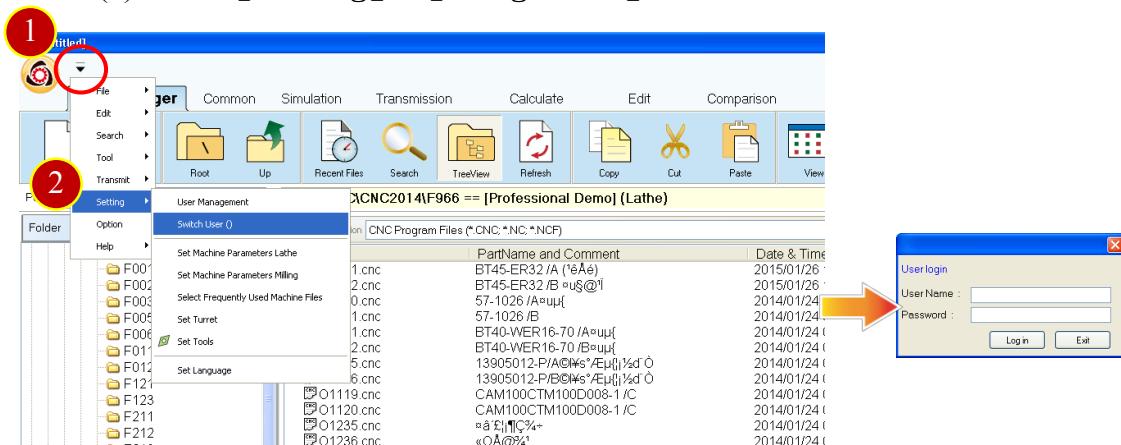
13.6.5 Change User

Change user to another

Steps:

(1) Click Main Menu

(2) Click 【Setting】 > 【Change User】

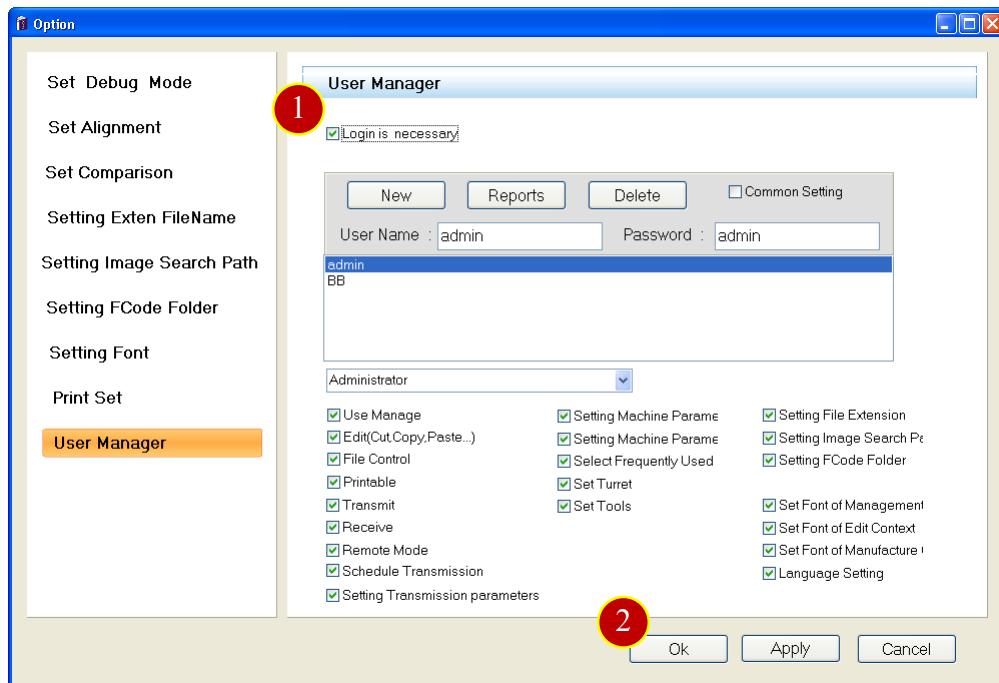


13.6.6 Login Setting

Users have to log in when open the software

Steps:

- (1) Check 【Login is necessary】
- (2) Click 【OK】



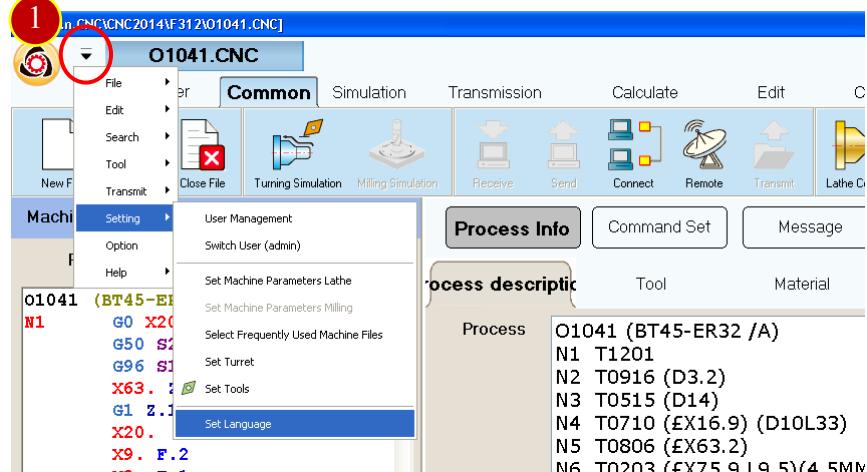
- (3) Users have to log in when they open the software next time



13.7 Language Setting

13.7.1 Steps

(1) Click 【Setting】 > 【Setting Language】 in the main menu



(2) Select the language

(3) Click 【OK】

(4) Restart the software



CNC Simulator
CAD/CAM System
Training Equipment



RenAn Information Technology Co., Ltd.

5F-3, No.1 Keji Road, Dali Dist., Taichung City 41264, Taiwan

+886(4)24918777
service@renan.com.tw
www.renan.com.tw



RenAn Website