

# CNC Simulator

Milling 3Axis

Fanuc-0i-MF

## Operation Manual



**AI | EQ**  
Smart Learning & Safety Technology



**RenAn**



# RenAn Information Technology Co., Ltd.

## Product Guarantee

<b>Product Name:</b>		
<b>Customer Name:</b>		<b>Purchase Date:</b>

### 1.1 Guarantee items description:

1.1.1 From this Product guarantee's purchase date, the customer can possess the following free services:

- (1) Software product: As from purchase date, the customer can change version within 90 days.
- (2) Hardware product: As from purchase date, free maintenance within a year.

1.1.2 During the free maintenance period, the company will ask for cost of the service fee when:

- (1) The company is unable to show product guarantee.
- (2) The content in the product guarantee doesn't match with the product, or the content has been modified, or cannot be identified.
- (3) After purchasing, the hardware has been damaged or has broken down during transportation.
- (4) Damage due to natural damages, or accidents, such as using inappropriate voltage.

During this free maintenance period, the replaced parts or machines possess all brand new functions. However, the original parts and equipment remains the possession of RenAn or the reseller.

During the free maintenance period, please contact our resellers for machine repair. When transporting the machines, please use the original RenAn package; otherwise, RenAn won't be responsible for the damage caused during transportation. Any questions regarding the transportation fee, please contact RenAn or our resellers.

Beyond the free maintenance period, the repair fee will differ according to different conditions.

### 1.1.3 Limited flaw guarantee:

- (1) RenAn assures all functions mentioned in the instruction manual within 90 days.
- (2) Within one year after delivery under normal use, RenAn will be responsible for any flaw.

### 1.1.4 Customer compensation:

During abnormal transportation, the product may only be returned or refunded with the product receipt.

RenAn won't be responsible for any lost files, stolen parts, and damages caused on purpose.

RenAn won't be responsible for any other flaws, such as the parts included with this product.

**This contract is under the Law of Taiwan, ROC**



# **RenAn Information Technology Co., Ltd.**

## **Product Guarantee**

The following are the terms and conditions under which RenAn will provide the software program described below.

Your acceptance of these terms and conditions by your indication of such acceptance in following the terms and conditions will indicate that you are legally accepting these terms and conditions.

### **1.2 [You, as the Customer, agree as follows:]**

#### **1.2.1 Definitions**

- (1) "Designated Computer" is the one computer upon which Customer runs the Licensed Software.
- (2) "License Software" is the software supplied by RenAn to this Agreement in executable form only.

#### **1.2.2 License**

- (1) You may not distribute copies of the licensed Software or its accompanying written materials to others.
- (2) You may not solely copy disks, unless under agreement. No other copies shall be made, sold, rent, loaned, published or exchanged unless authorized by RenAn.
- (3) You may not modify disk's codes, reverse-engineer the software, or decompile codes.
- (4) RenAn retains all rights, title and interest in the Licensed Software, and in all patent, copyright, trade secret and trademark and other intellectual property and proprietary rights.

#### **1.2.3 General**

- (1) The License Agreement is a legal agreement between you and RenAn Information Technology Co., Ltd. for the software Product(s) you have purchased from RenAn.
- (2) The license has been granted the authority to sub-license the Software to you the Customer.
- (3) This software product is protected by copyright laws and international copyright treaties, as well as other intellectual property laws and treaties. The software product is licensed, not sold. The software Product consists of product documentation, a server application, and support files individually identified as "component" and collectively referred to herein as "software".

This license constitutes the complete agreement between you and RenAn, and supersedes any prior discussions or agreements related to the software. This license may be modified only by the written agreement of both parties.

I Agree : \_\_\_\_\_



# RenAn Information Technology Co., Ltd.

## Product Guarantee

### Proprietary Notice

This document contains propriety information of RenAn Information Technology Co., Ltd. and is to be used only pursuant to and in conjunction with the license granted to you with respect to the accompanying RenAn licensed software.

**RenAn Information Technology Co., Ltd.  
Trademark Proclamation.**



**RenAn Information Technology Co., Ltd.**

All Rights Reserved. This manual content may not be copied, photocopied or published.

### COPYRIGHT.

All title and intellectual property rights in and to the software product, the accompanying printed materials, and any copies of the software product are owned by RenAn. All title and intellectual property rights in and to the content which may be accessed through use of the Software Product is the property of the respective content owner and may be protected by applicable copyright or other intellectual property laws and treaties. If you do not have a valid license for this Software Product, you have no rights under the foregoing section.

RenAn is a registered trademark.

All other trademark belongs to their respective owners.



## Contents

1. Basic Introduction.....	1-1
1.1 Manual Symbol Explanation.....	1-1
1.2 Start CNC Milling Simulator.....	1-2
1.3 Introduction.....	1-3
1.4 View and Application Toolbar.....	1-4
1.5 View Toolbar.....	1-6
2. Tool and Material.....	2-1
2.1 Material Setting.....	2-1
2.2 Magazine & Tool Installation.....	2-2
2.3 Tool Manager.....	2-9
3. Machine Operation Panel.....	3-1
3.1 Description of Main Knobs .....	3-1
3.2 Operation Mode Selection Knob .....	3-3
3.3 HANDLE Operation (Handwheel Feed).....	3-4
3.4 Common Buttons Description .....	3-6
4. Controller Panel Operation .....	4-1
4.1 Controller Panel Introduction.....	4-1
4.2 Program Edit.....	4-3
5. Transmission Parameter Setting and Operation .....	5-1
5.1 RS232 COM Setting.....	5-1
5.2 Transmission Parameter Setting.....	5-3
5.3 Receive Program from the Controller .....	5-7
5.4 Send Program from the Controller.....	5-9
5.5 CNC File Import/Export Management.....	5-11
6. Basic Machine Operation .....	6-1
6.1 ZRN Operation .....	6-1
6.2 Manual Tool Change.....	6-3
6.3 Edge Finder Operating Standard Procedures .....	6-4
6.4 Z Axial Gauge Operating Standard Procedures .....	6-5
6.5 Work Coordinate Setting .....	6-6
6.6 Tool Offset (Tool Gauge) .....	6-11
6.7 Auto Run.....	6-15
6.8 Machine Alarm .....	6-18

# TM Fanuc Oi-MF Manual



RenAn Information Technology Co., Ltd.  
[www.RenAn.com.tw](http://www.RenAn.com.tw)

---

7. Appendix 1, System Parameters.....	7-1
7.1 System Parameters.....	7-1
7.2 I/O input and output device parameters .....	7-1
7.3 About programming parameters.....	7-3
7.4 Background editing function setting .....	7-4



## 1. Basic Introduction

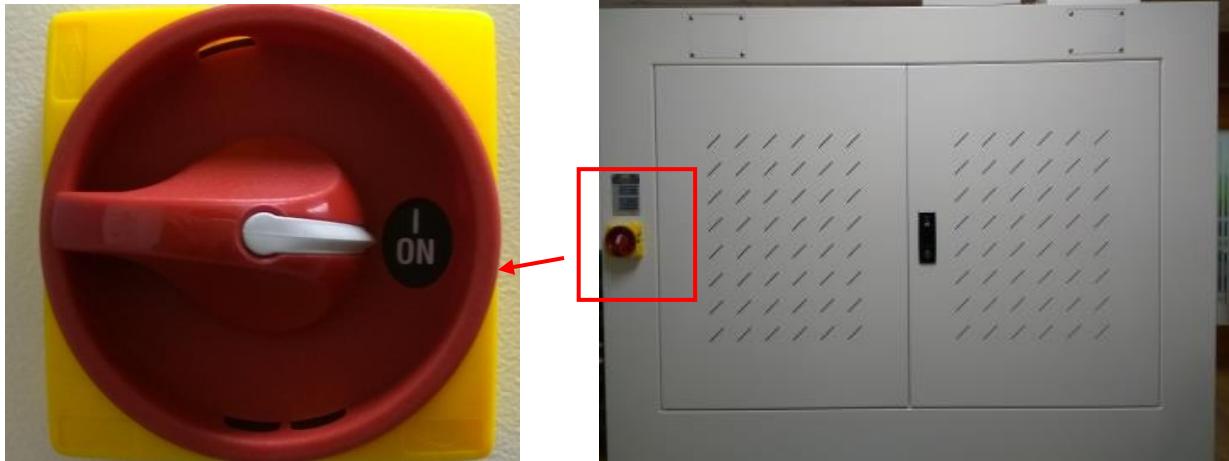
### 1.1 Manual Symbol Explanation

- |                        |       |   |
|------------------------|-------|---|
| 1.1.1 Button:          | [ ]   | Buttons related to the software interface.                          |
| 1.1.2 Pull-down Menu:  | < >   | Pull-down menu.   |
| 1.1.3 Key and Button:  | [( )] | Buttons and knobs related to<br>Machine Operation Panel, handwheel. |
| 1.1.4 Button:          | [ ]   | Buttons related to Controller.                                      |
| 1.1.5 Field and Edit:  | _____ | Edit field input.   |
| 1.1.6 Term:            | ' '   |   |
| 1.1.7 Steps:           | ①     |   |
| 1.1.8 Controller Edit: | " "   | Controller Edit input.  |

## 1.2 Start CNC Milling Simulator

### 1.2.1 How to Activate the System

- 1 (1) Power On the Simulator Main Power by rotating the switch to the [( ON )] position



- (2) Press the [( POWER ON )] on the Operation Panel of the Controller Box to activate CNC Simulator





## 1.3 Introduction

The workspace of the training system includes:

Machine Simulation Area, Controller and Toolbar, Operation Panel



### 1.3.1 Workspace Description

Workspace Components	Description
<b>Machine Simulation</b>	Display the entire machine and perform cutting Workpiece in realistic 3D space.
<b>Controller Function Panel</b>	Includes Controller Function Panel and Touch Panel Toolbar.
<b>Machine Operation Panel</b>	Control panel for machine operation.

## 1.4 View and Application Toolbar

#### 1.4.1 View and Function Description ឧអិលាល នឹង· កំពុងបាយ ស៊ីវិទ្យា

- (1) 3D View Display: Display the XYZ Axis angle of the current 3D view screen. ការរំលែកអង្គភាព 3D  
Touch and drag to rotate the view angle. នូវសម្រាប់រំលែក XYZ និងអង្គភាព 3D ដែលបានចូល និងរំលែក

(2) View Button: Commonly used basic view angle switching function. ផ្លូវអង្គភាព : ជំនួយដែលបានចូល និងរំលែក

(3) Application Toolbar: Sliding touch interface. Slide to switch between several types  
of function buttons. តើម្រួល ជំនួយដែលបានចូល : ចូលការងារដោយស្វែងរកដោយខ្សោយ និងផ្តល់ការងារ។  
 : Switch to previous function category ផ្តល់ការងារមុនពេលដែលបានចូល  
 : Switch to next function category ផ្តល់ការងារខ្លះពេលដែលបានចូល.

ବୋଲିଯାଙ୍କ ଫଳମୁଖରୁ ୩ ମୀଟ୍.

## 3D View Display



မြတ်စွာသော်လည်းကောင်းမှုများ

## Application Toolbar



**View Button**

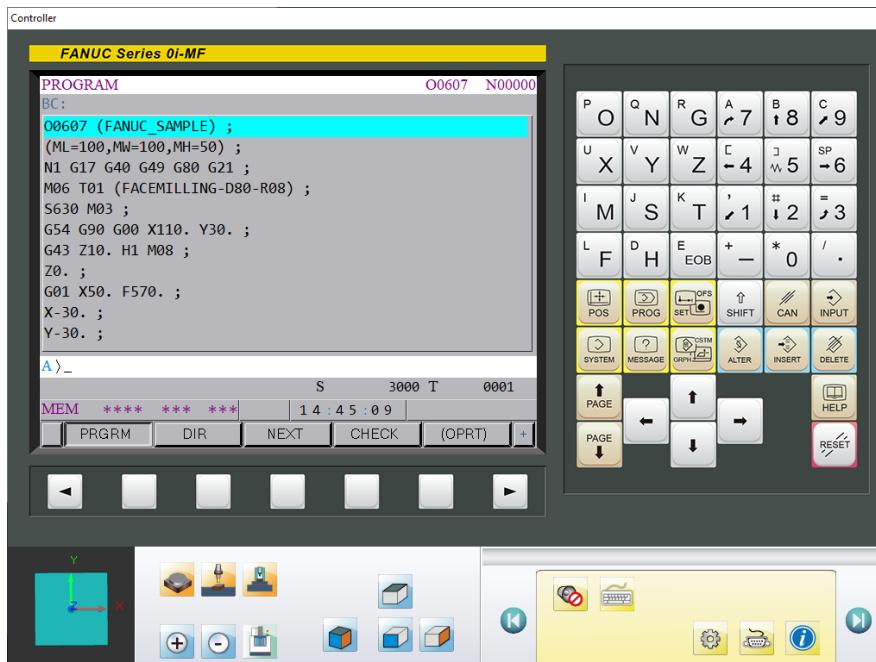
120.

1.4.2 Description of Application Function Button *កំណត់បន្ថែមសម្រាប់បង្កើតរូប*

អ្នកសង្គមនៃបង្កើតរូប	ការពិពណ៌នា
	ឯកសារ ចំណាំ ការផ្តល់ឈើ និងការកែតម្រូវ View Shift, Tool Offset Functions
	ឯកសារ ឯកសារកែតម្រូវ , ការដំឡើ និងការឲ្យចូល CNC , ក្រុងការកែតម្រូវ Material, Tool Magazine, CNC File Import/ Export, Tool Manager
	ឯកសារ / ឯកសារការកែតម្រូវ តាមការ រាយការ រាយការ ឯកសារ ឯកសារកែតម្រូវ ឯកសារ ឯកសារកែតម្រូវ Import/Save operation configuration, resume to default, Save product and import material
	ការកែតម្រូវ ឬការកែតម្រូវ / ការកែតម្រូវ Software/ Hardware Settings
	ការគ្រប់ការងារ ការកែតម្រូវ Simulation Speed Adjustment

## 1.5 View Toolbar

Commonly used basic view angle switching function.



### 1.5.1 Buttons Description

Button	Name	Description
	Machine view	ແກ້ໄຂພິບທັງໝົດ. <i>ແກ້ໄຂພິບທັງໝົດ</i> . Adjust to the view of the entire machine
	Clip view	ແກ້ໄຂພິບຂອງພົມ <i>ແກ້ໄຂພິບຂອງພົມ</i> Adjust to the view of Material and Tool
	Material view	ແກ້ໄຂພິບພະຍານ ຂະຫຼາກສົມ <i>ແກ້ໄຂພິບພະຍານ ຂະຫຼາກສົມ</i> Adjust to the view of Material
	Show/Hide Machine Case	ແກ້ໄຂ/ຍິ່ງເກີດ/ຍິ່ງເປັດ <i>ແກ້ໄຂ/ຍິ່ງເກີດ/ຍິ່ງເປັດ</i> Show/Hide the machine case
	Zoom Out	ພູມຕາກຳ ທຸລະນາ <i>ພູມຕາກຳ ທຸລະນາ</i> Zoom out on the view
	Zoom In	ພູມຕາກຳ ປັດຕິພຸດ <i>ພູມຕາກຳ ປັດຕິພຸດ</i> Zoom in on the view

# TM Fanuc 0i-MF Manual



[www.RenAn.com.tw](http://www.RenAn.com.tw)

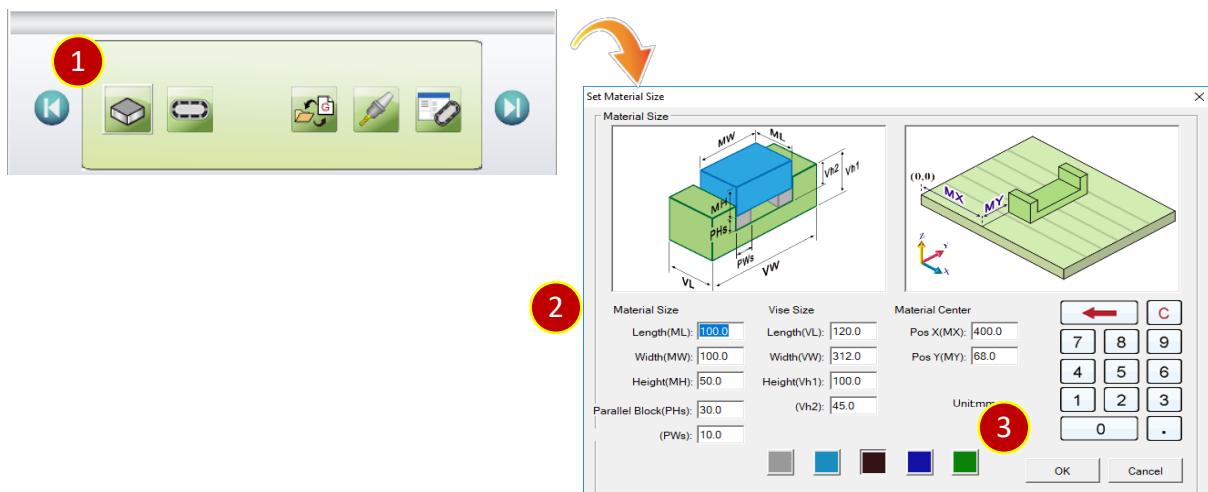
Button ໜົມ	Name ດີໂຫ	Description ຕິດຕັ້ງ
	Top View ຂູ່ຂະໜາດ ດົກນໍານານ	Switch to top view ສ່ວນຢັ້ງທີ່ມີຂູ່ຂະໜາດ ດົກນໍານານ
	Front View ຂູ່ຂະໜາດ ດົກນໍານານ	Switch to front view (XZ plane) ສ່ວນຢັ້ງທີ່ມີຂູ່ຂະໜາດ ດົກນໍານານ (ຄົງລົງ XZ)
	Side View ຂູ່ຂະໜາດ ດົກນໍານານ	Switch to side view (XY plane) ສ່ວນຢັ້ງທີ່ມີຂູ່ຂະໜາດ ດົກນໍານານ (ຮະແບບ XY)
	ISO View ຂູ່ຂະໜາດ ISO	Switch to ISO view (XYZ) ສ່ວນຢັ້ງທີ່ມີຂູ່ຂະໜາດ ISO (XYZ)
	UP ແກ້ໄຂ	Each click moves the current object up by 10%. ກາງກຳລິດໄປເກົ່າ. ດົກ ດົກກຳລິດໄປເກົ່າ 10%.
	DOWN ແກ້ໄຂ	Each click moves the current object down by 10%. ກາງກຳລິດໄປຕ້ອງ. ດົກ ດົກກຳລິດໄປຕ້ອງ 10%.
	LEFT ຫຼື້ນ	Each click moves the current object to the left by 10%. ກາງກຳລິດໄປຫຼື້ນ. ດົກ ດົກກຳລິດໄປຫຼື້ນ 10%.
	RIGHT ຫຼື້ນ	Each click moves the current object to the right by 10%.
	Load/Unload ໃຫ້ ຂຳ ລຶກ ຕາຫຸ່ນ Spindle Tool ໄຫວ້າ ນຳ ນຳ	Install or remove Spindle Tool ມີການ ລຶກ ຕາຫຸ່ນ ຕາຫຸ່ນ ນຳ
	Show Zero ໂຈຣ ຊົກ	Show program Zero Position
	Z Axial Gauge ເກົ່າຕົວແນວ ແນວ Z	Install or change position of Tool Gauge for Tool Offset
	Measure ປົມ	Activate to measure the size of the Workpiece (Only when the Spindle stops)

## 2. Tool and Material

### 2.1 Material Setting

#### 2.1.1 Procedure

- (1) In Application Toolbar, press [ Material Setting ] and proceed with setting material dimensions.
- (2) Enter the window of [ Material Setting ], input the diameter and length of the material.  
E.g.: Length (L)=100, Width(W)=100, Height(H)=50
- (3) After finishing setting, press [ OK ] to exit.



#### 2.1.2 Material Settings Buttons Description

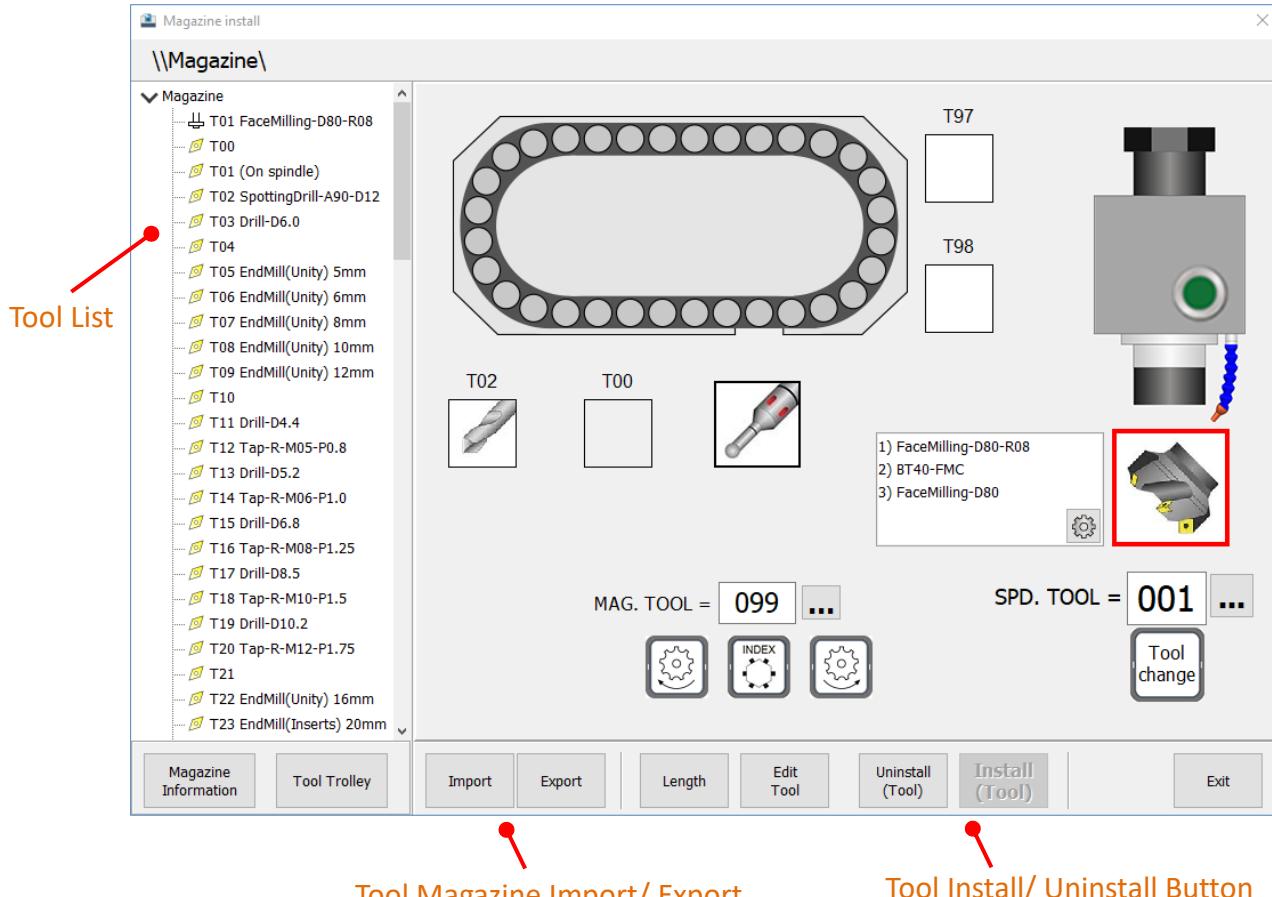
Button	Description
	Numeric keys from 0~9 and decimal point
	Backspace
	Clear
	Confirm the material settings
	Exit the Material Setting Window



## 2.2 Magazine & Tool Installation

Install Tool to the Tool Magazine on the machine before simulation.

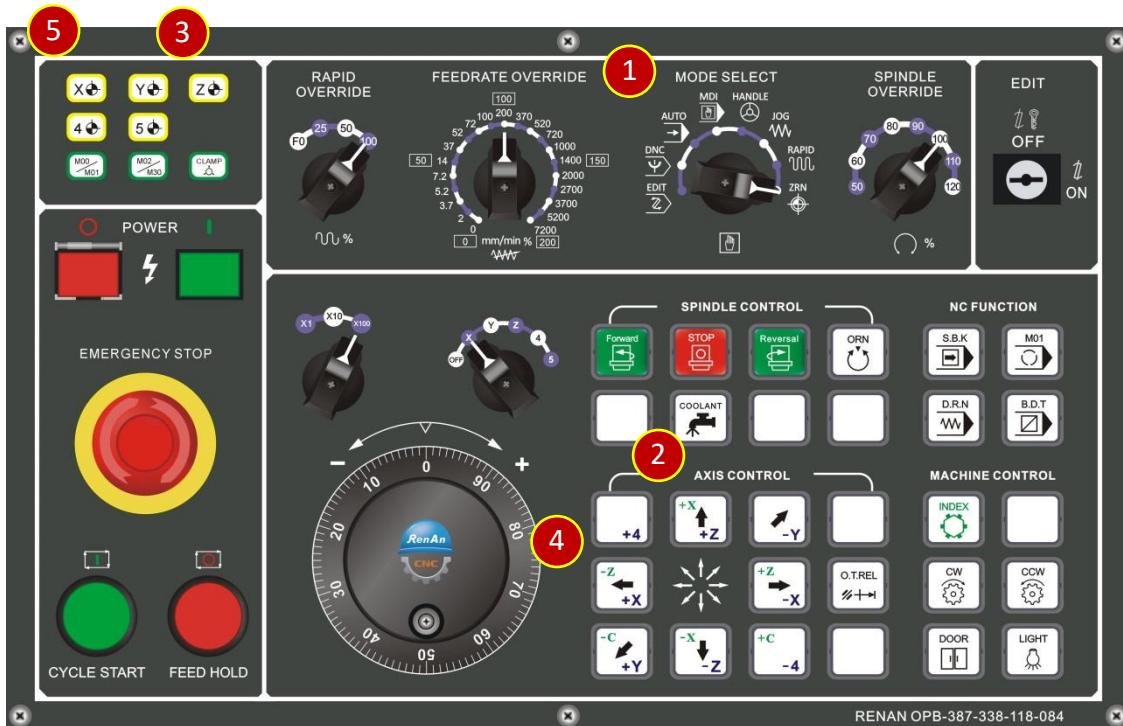
### 2.2.1 Tool Magazine Install Function Description



## 2.2.2 Change Spindle Tool

E.g.: Change Spindle Tool to No.35

- (1) Switch to [( ZRN )] Mode on the Machine Operation Panel.
- (2) Press [( +Z )], return Z Axis to Reference Point.
- (3) When the Z Axis ZRN is complete, will light up.
- (4) Press [( +X )] and [( +Y )], to return to X and Y Reference Point.
- (5) When the X and Y Axis ZRN is complete, and will light up.



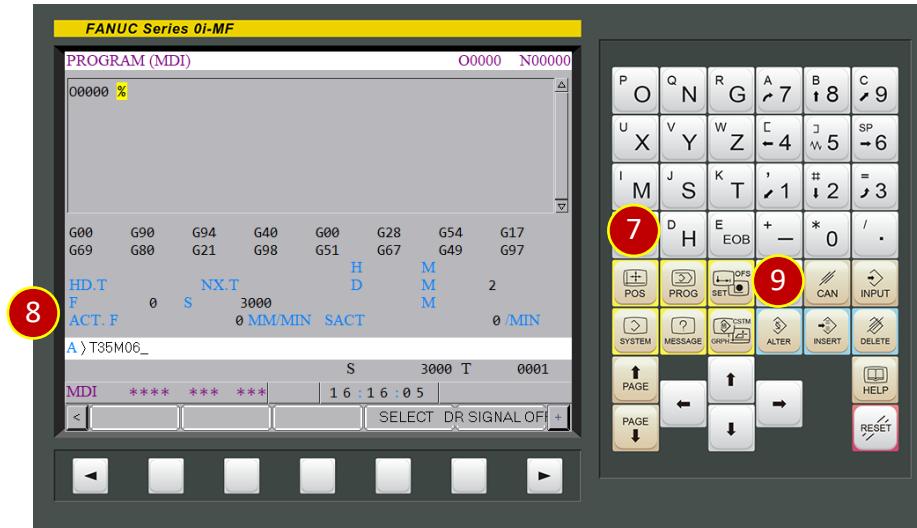
# TM Fanuc Oi-MF Manual



- (6) Switch to [(MDI)] Mode on the Machine Operation Panel.
- (7) Press [PROG] on the Controller Function Panel.
- (8) Key in “;T35 M06;” on the Controller Function Panel.
- (9) Press [INSERT].
- (10) Press [(CYCLE START)] on the Machine Operation Panel to change Tool.



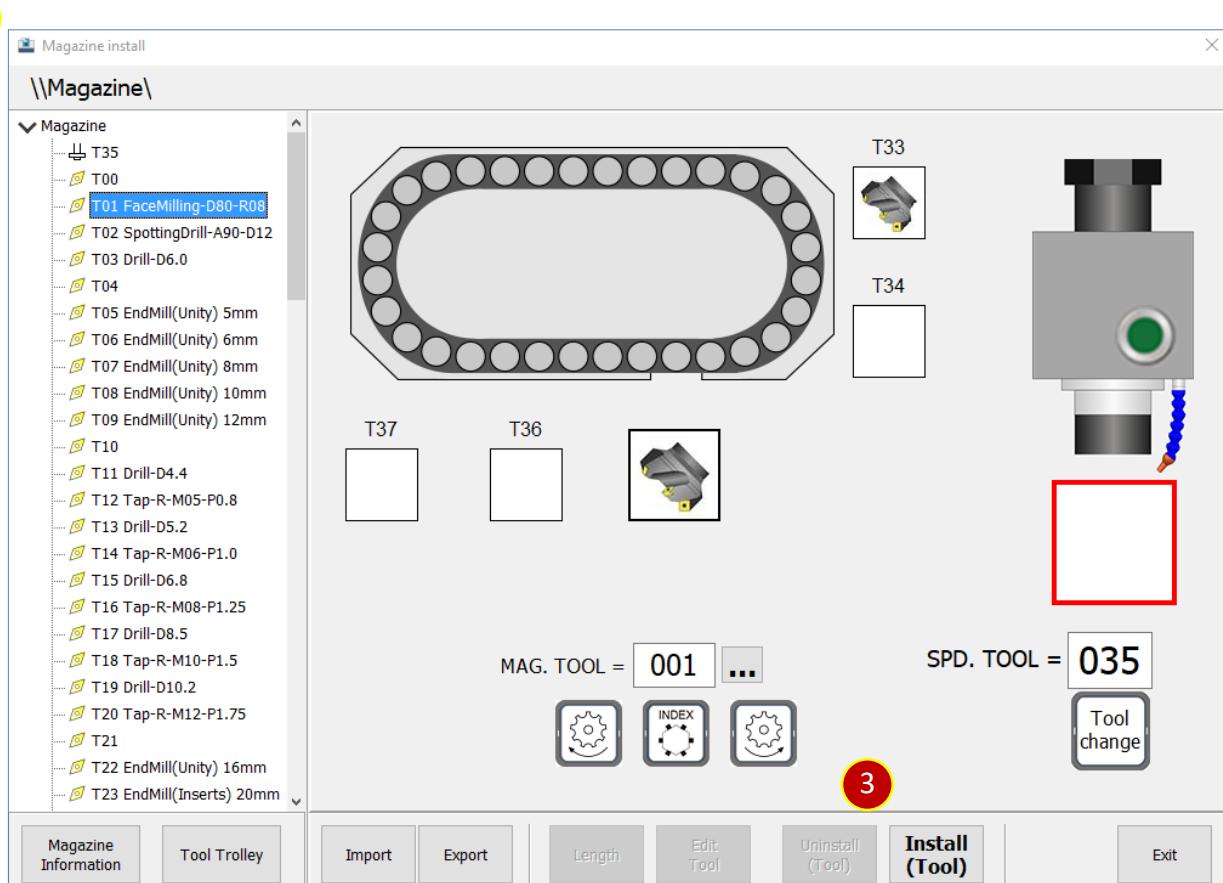
Controller



## 2.2.3 Install Tool

e.g., Install EndMill-D10 on Tool No.35

- (1) Press [ Magazine Tool Setting ] to set up Tool Magazine.
- (2) Enter Magazine Installation Window.
- (3) Press [ Install ] to open Tool Installation Window.



# TM Fanuc Oi-MF Manual

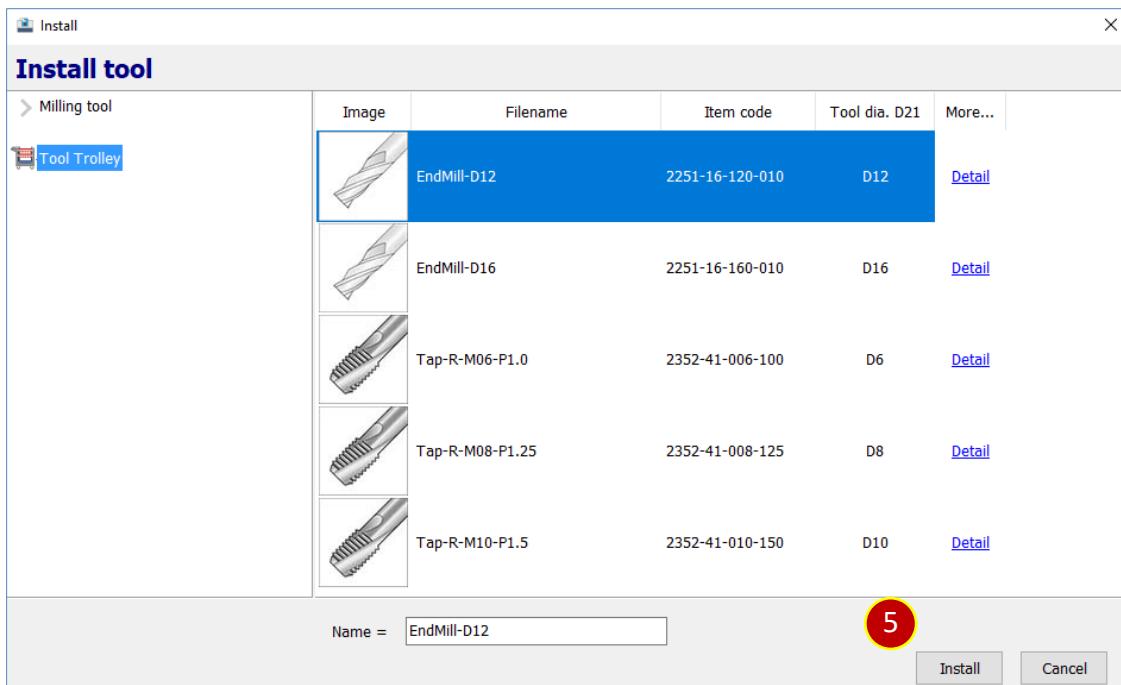


RenAn Information Technology Co., Ltd.  
www.RenAn.com.tw

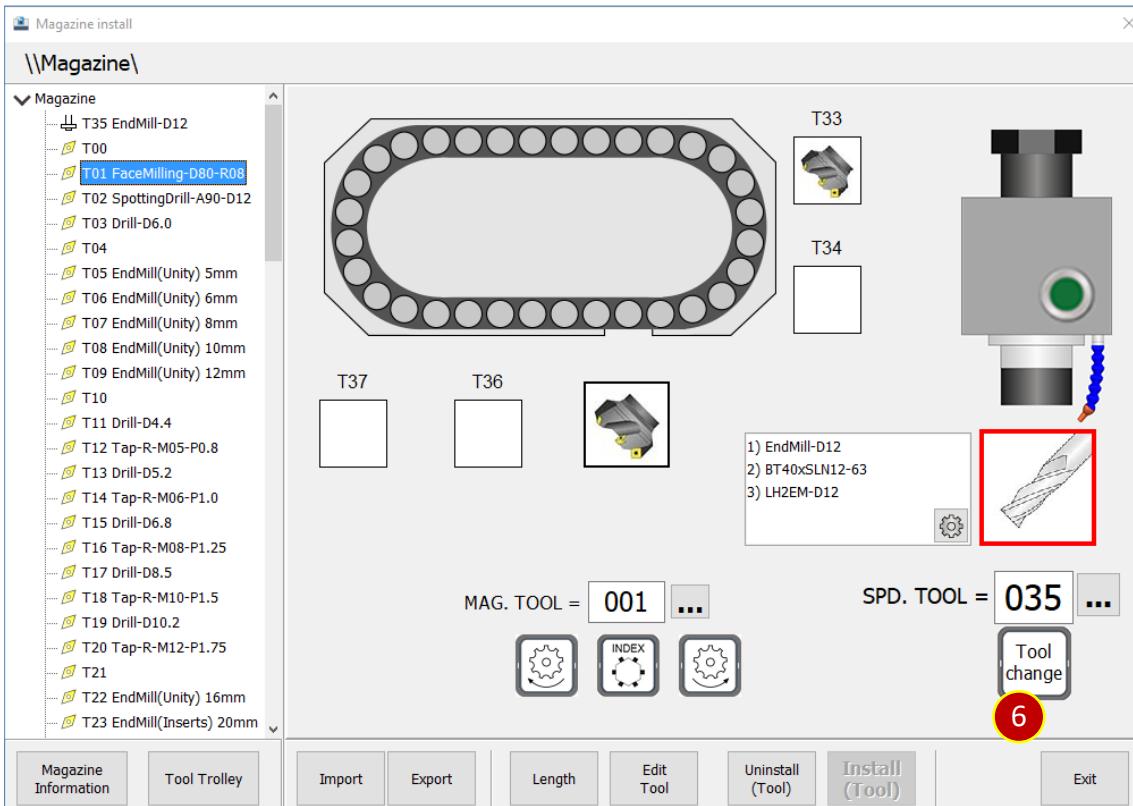
(4) Choose Tool in the EndMill Tool data.

e.g., [ EndMill-D12 ]

(5) Press [ Install ] to retrieve Tool to Tool Magazine.

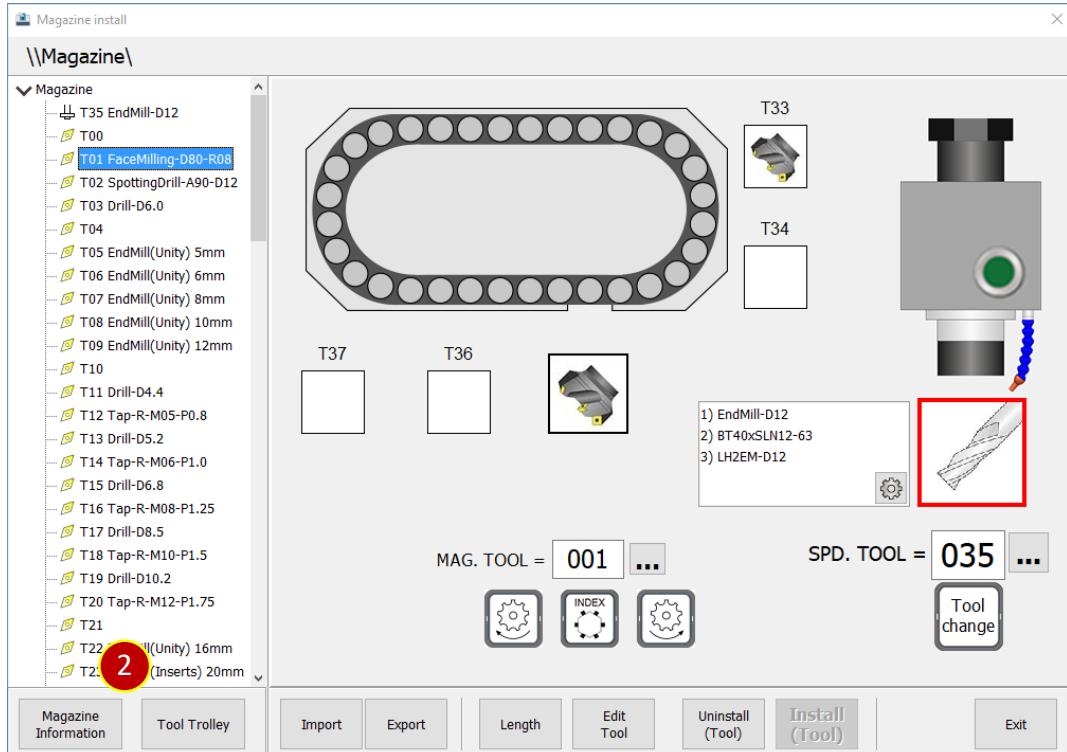


(6) Press [ Exit ] to finish Tool Magazine installation.

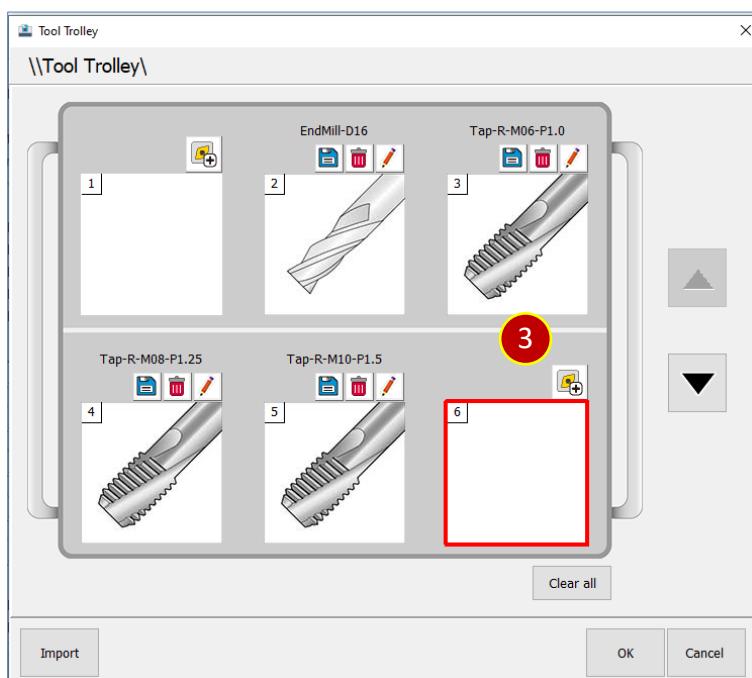


## 2.2.4 Tool Trolley Adjustment

- (1) In Application Toolbar press [ Magazine Tool Setting ].
- (2) Press [ Tool Trolley ] to open Tool Trolley Adjustment Window.



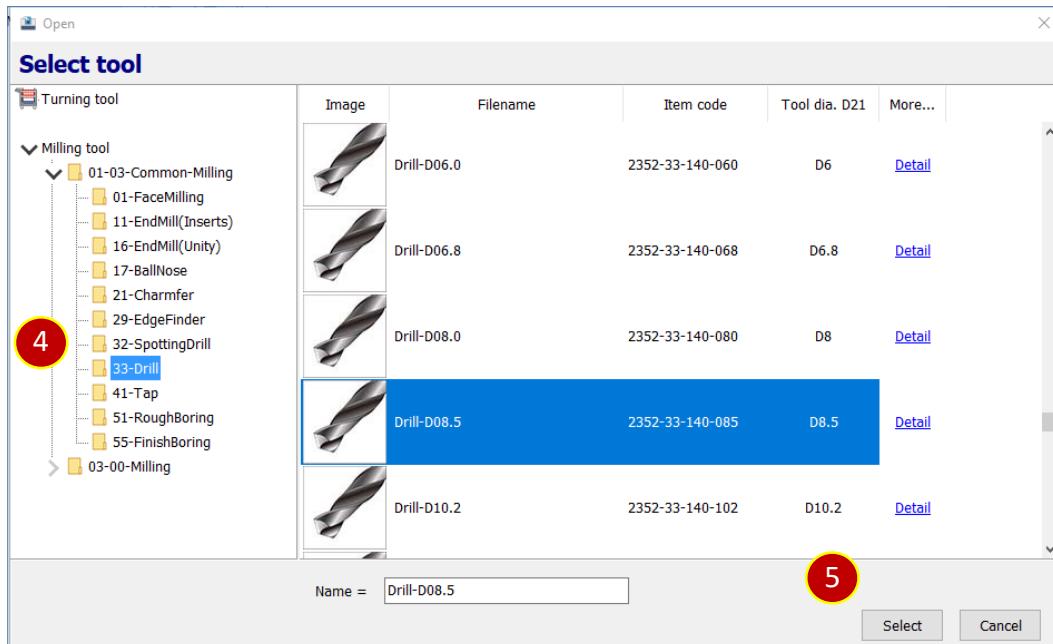
- (3) Press [ ] to add Tool in Select Tool Window.



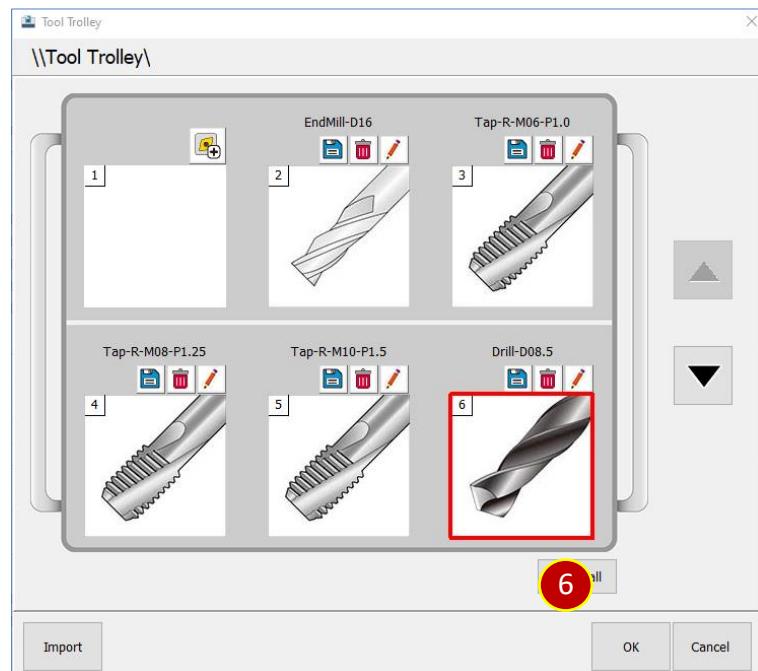
(4) Select Tool.

e.g., [ 01-03-Common Milling ] > [ 33-Drill ] > [ Drill-D08.5 ]

(5) Press [ Select ] add Tool to Tool Trolley.



(6) Press [ OK ] to return to Tool Trolley Window.



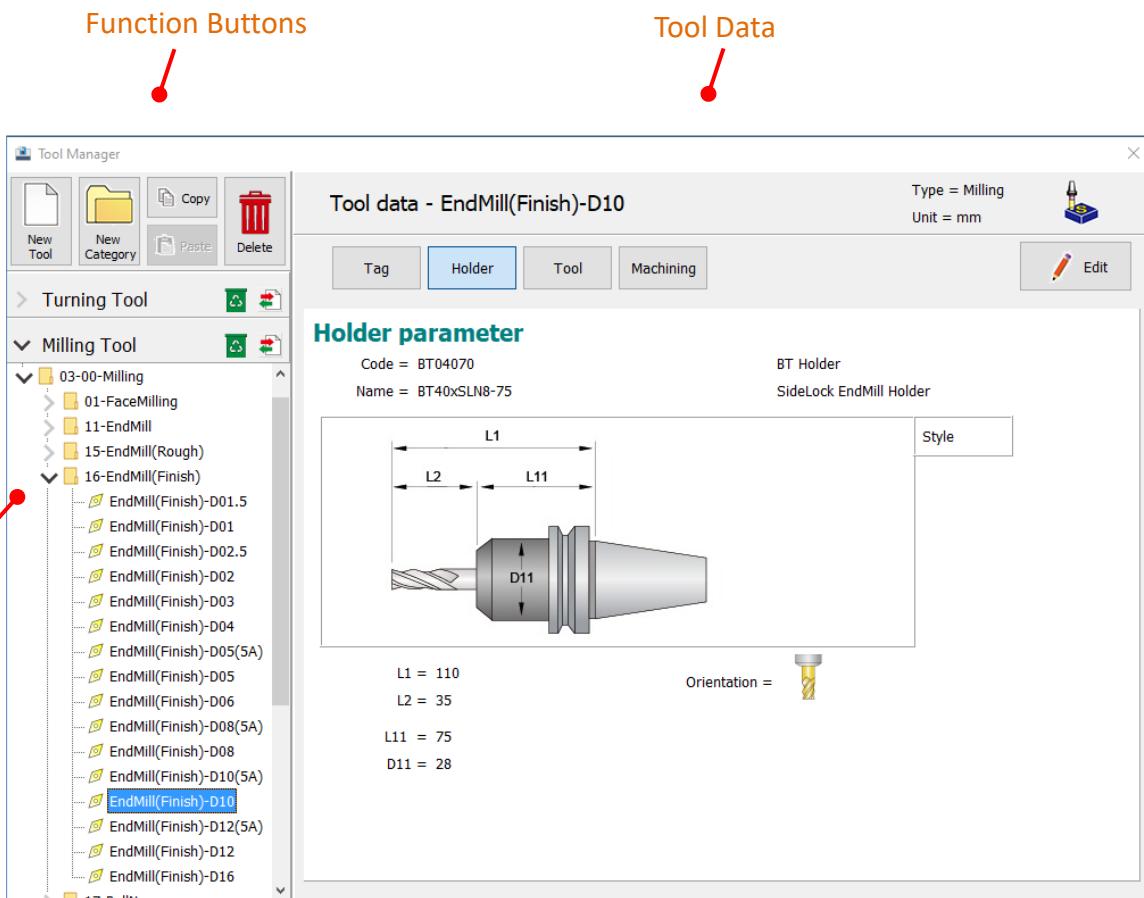
## 2.3 Tool Manager

Set up detailed Tool data, including Tool Tag, Holder Parameter, Tool Parameter, Machining Data, and Machining Condition as the basic data setting before simulation.

### 2.3.1 Enter Tool Setting Function Description:

- (1) In Application Toolbar press [ Tool Data ] .
- (2) After entering Tool Manager Window, the button description is down below:

### 2.3.2 [ Tool Manager ] Button Description:



## 2.3.3 New Tool

e.g., New Tool EndMill-D16

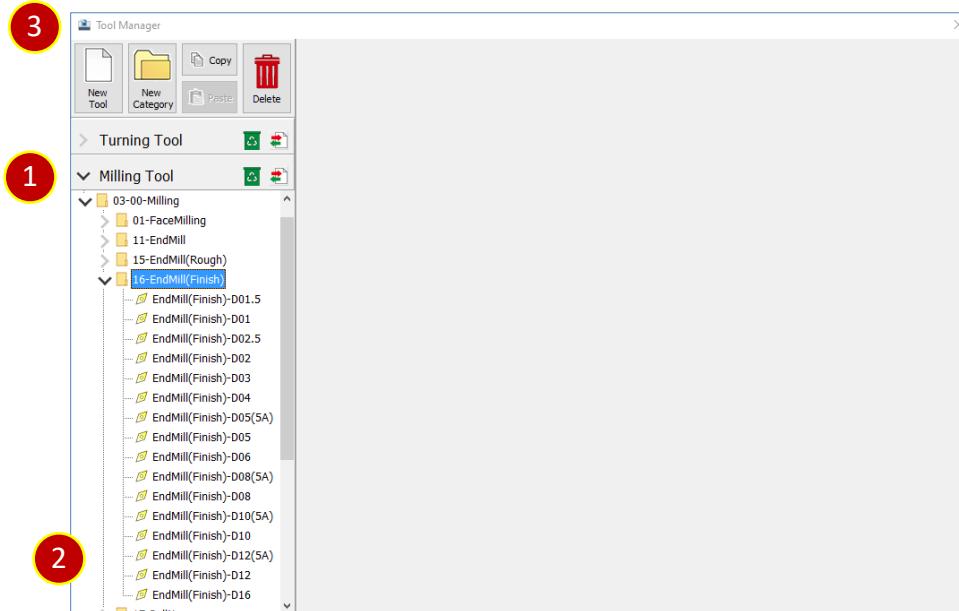
- (1) Select Tool category in Tool List.

e.g., [ 03-00-Milling ] > [ 16-EndMill(Finish) ]

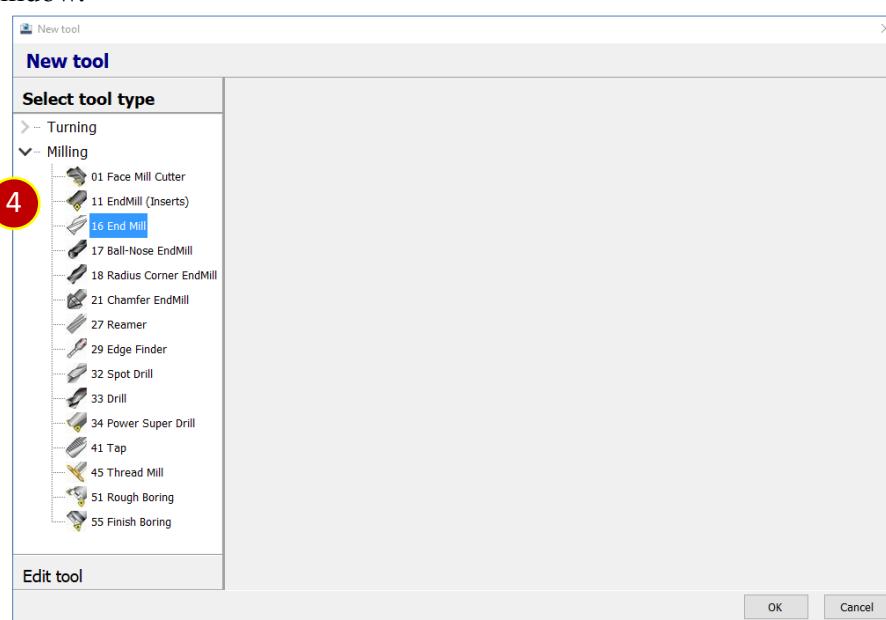
- (2) Press Tool number of a similar Tool.

e.g., EndMill-D16

- (3) Press [ New Tool ] to enter New Tool Window.

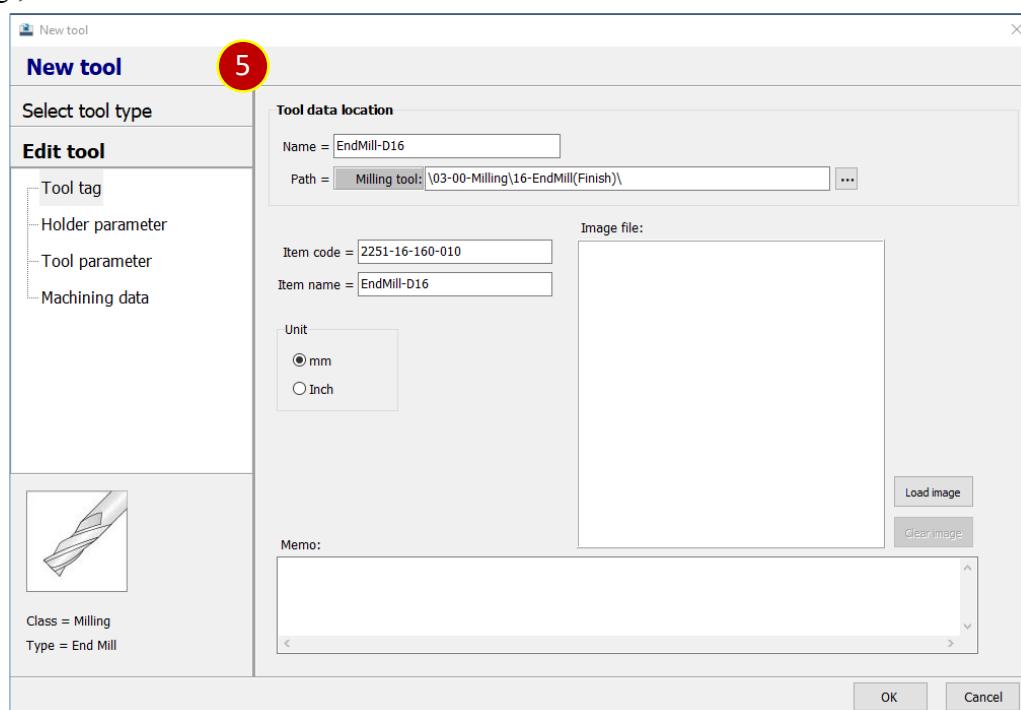


- (4) Press [ Select Tool Type ] > [ Milling ] > [ 16 End Mill ] to enter Edit Tool Window.



(5) Key in Name and Item name in Tool tag page.

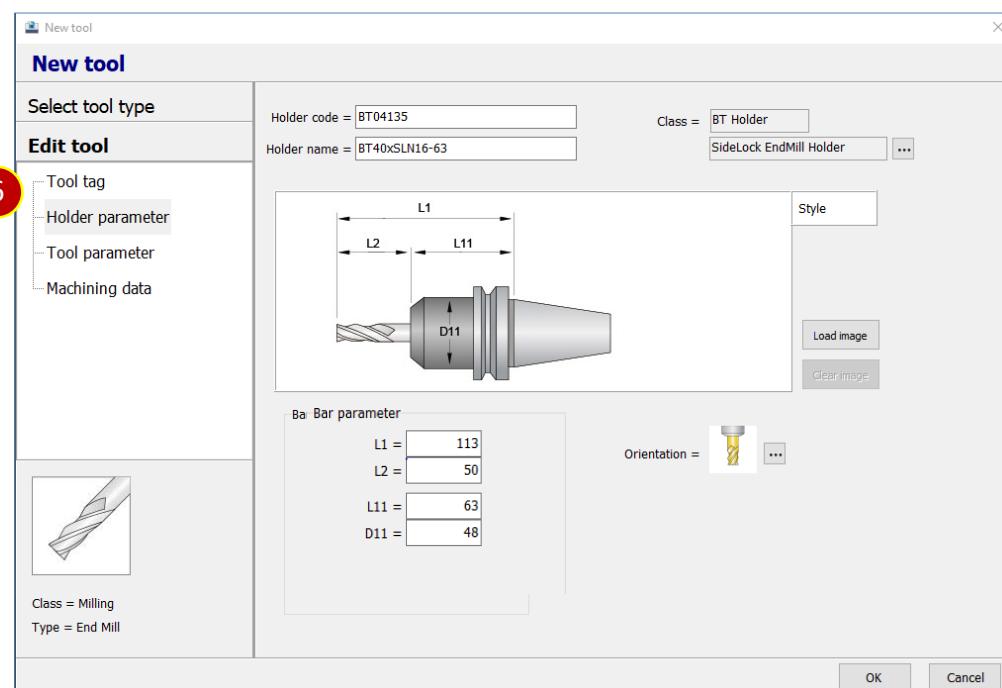
e.g., EndMill-D16



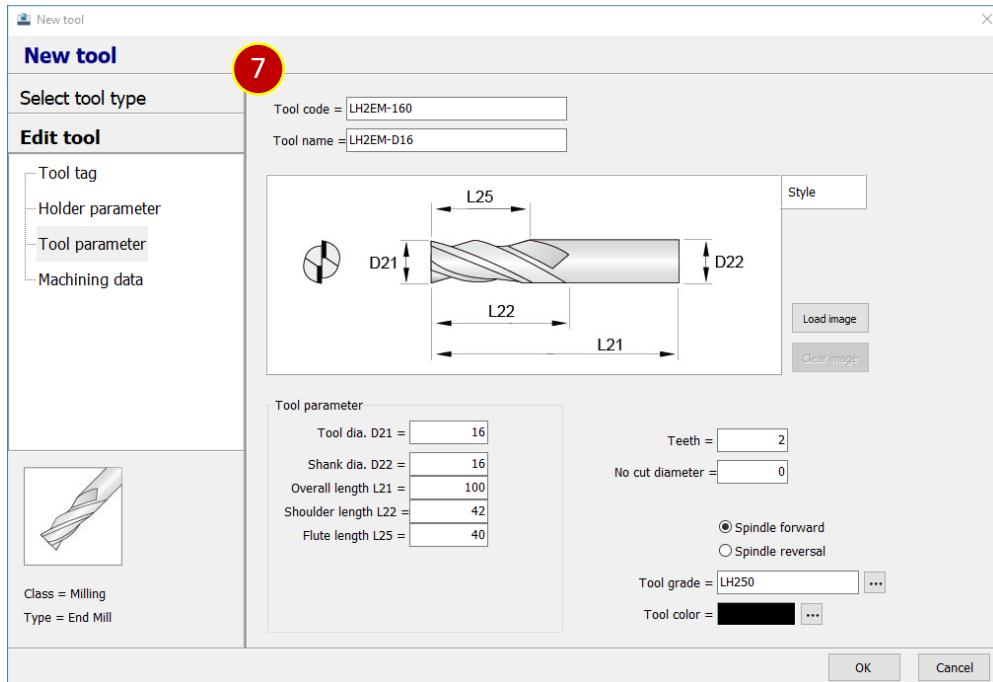
(6) Press [ Holder parameter ] and key in Holder parameter.

e.g., Holder code= BT04135, Holder name= BT40xSLN16-63

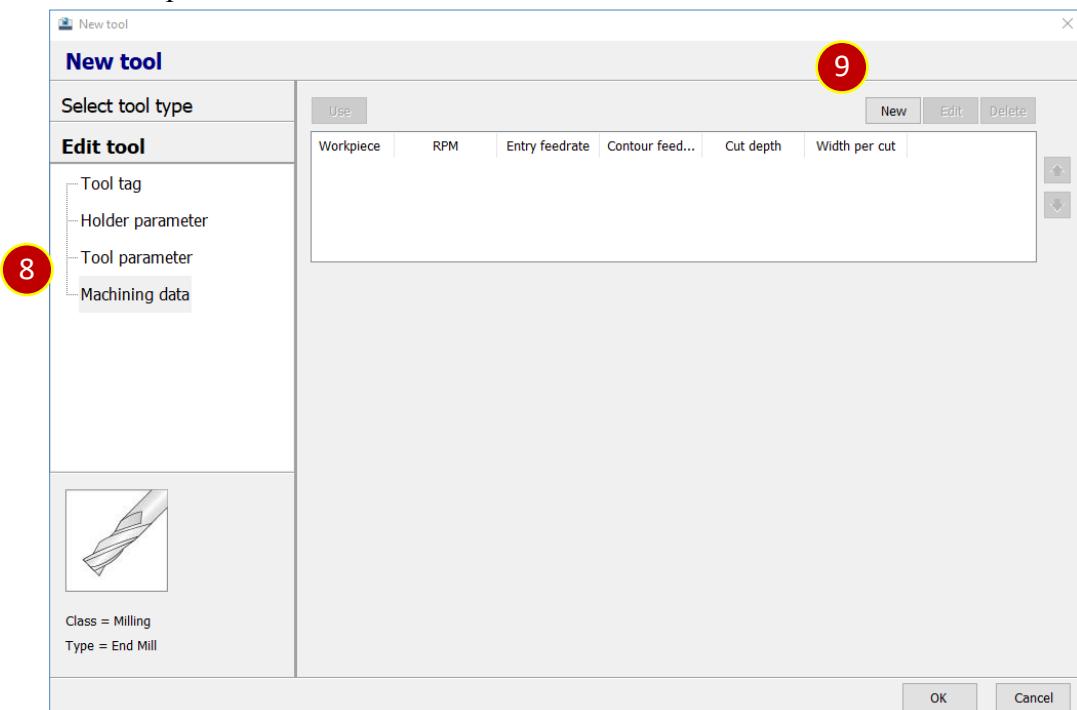
L1=113, L2=50, L11=63, D11=48



- (7) Press [ Tool parameter ] to key in Tool parameter.  
 e.g., Tool code= LH2EM-160, Tool name= LH2EM-D16  
 D21=16, D22=16, L21=100, L22=42, L25=40, Teeth= 2

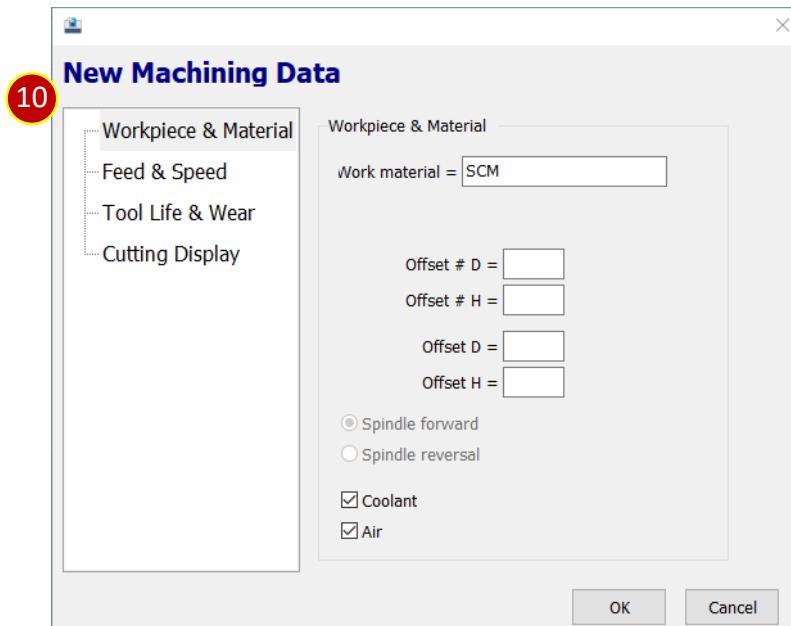


- (8) Press [ Machining data ] to display the Process data.  
 (9) Press [ New ] to add new Machining data, Workpiece, RPM, Entry feedrate, and Cut depth.



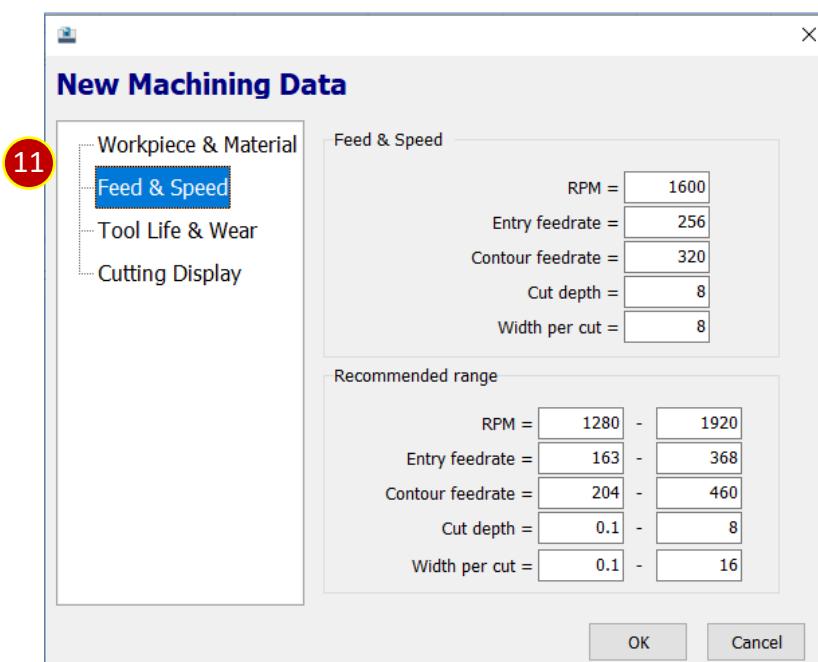
- (10) Press [ Workpiece & Material ] to edit Work material, Offset, Coolant, Spindle forward.

e.g., Work material= SCM



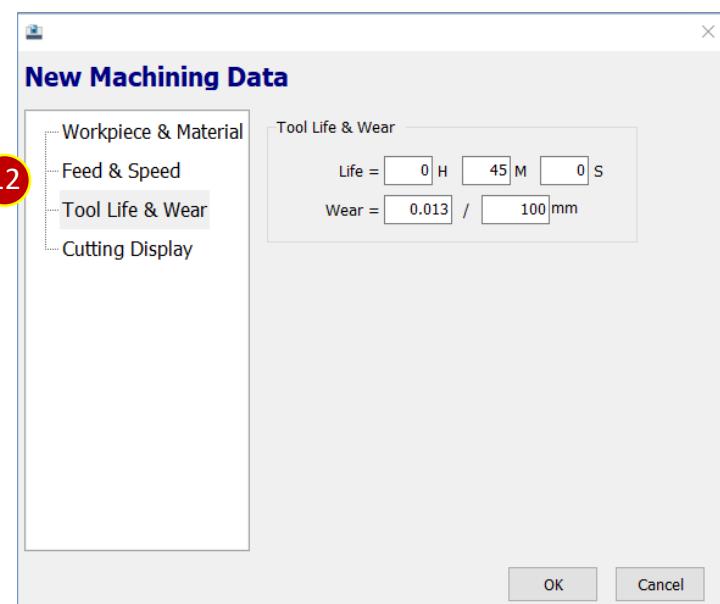
- (11) Press [ Feed & Speed ] to edit Cut condition, Spindle RPM, Feedrate range, and Cut depth.

e.g.

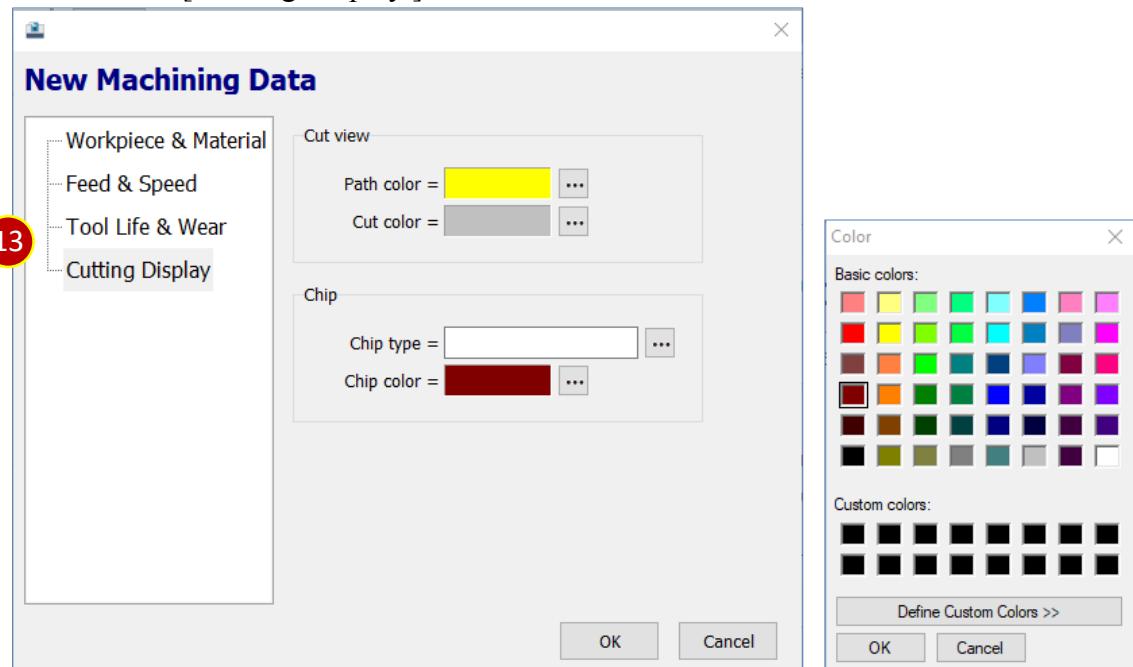


(12) Press [ Tool Life & Wear ] and key in data.

e.g., Life= 0H 45M 0S, Wear= 0.013/100



(13) Press [ Cutting Display ] and select the color.

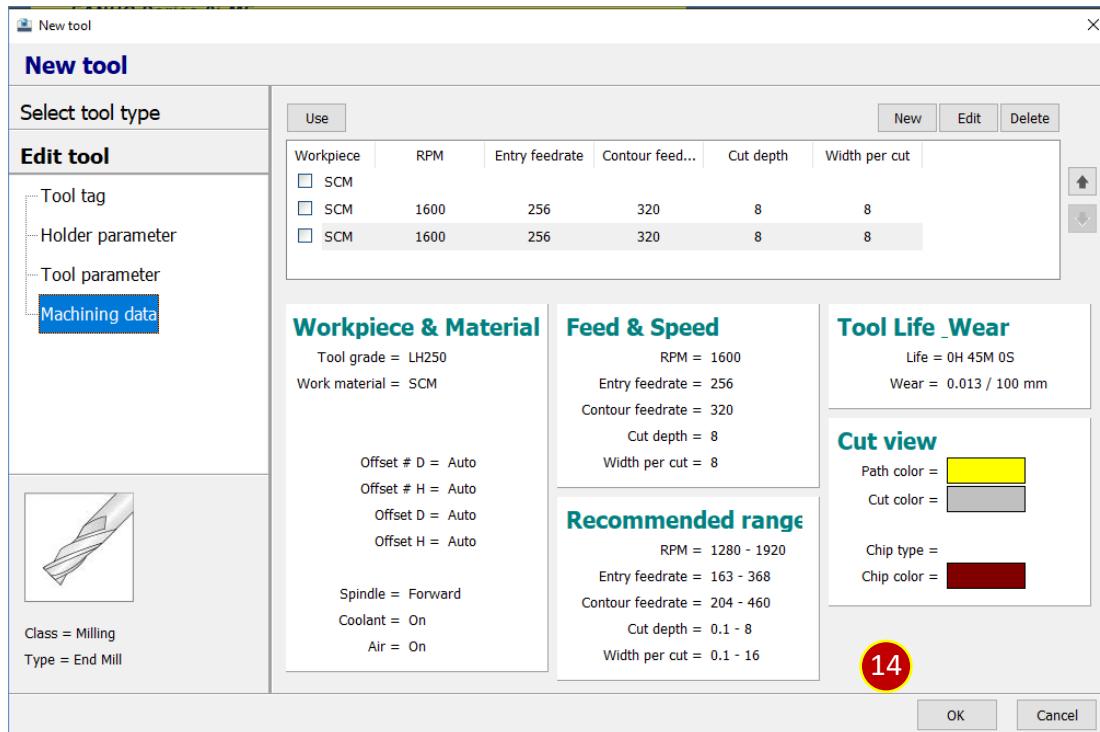


# TM Fanuc Oi-MF Manual



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

(14) Press [ OK ] to finish Tool setting.



### 3. Machine Operation Panel մանաւութեան հարաբեկութեան համար.

#### 3.1 Description of Main Knobs գլուխութեան համար.

(1) MODE SELECT Knob

Համարակալութեան համար.

(2) FEEDRATE OVERRIDE Knob

Համարակալութեան համար.

(3) RAPID OVERRIDE Knob

Համարակալութեան համար.

(4) SPINDLE OVERRIDE Knob

Համարակալութեան համար.

Համարակալութեան համար.

Համարակալութեան համար.

Համարակալութեան համար.

Համարակալութեան համար.

Համարակալութեան համար.



RENAN OPB-387-338-118-084

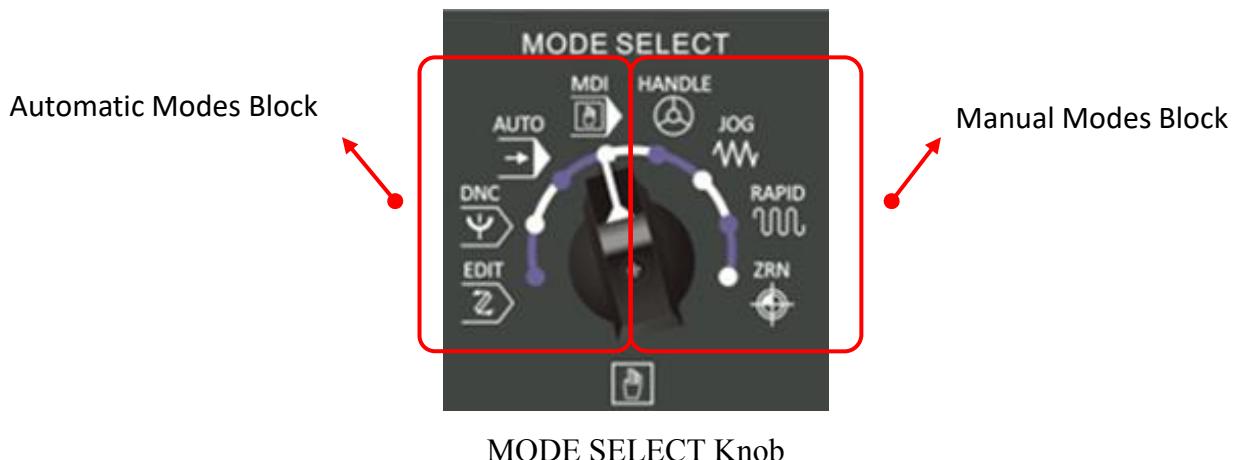
## 3.1.1 Description of each Main Knobs

Override Knob	Name	Description
	MODE SELECT	Operation Mode Selection Knob Use the switch to select the Machine Operation Panel and the current operation mode; other mode except for the selected mode will be disabled.
	FEEDRATE OVERRIDE	Cutting Feedrate Adjustment Override Knob (1) In Automatic mode: Adjust the cutting feed move speed from 0%-200%. (2) In Manual mode: Adjust the cutting feed move speed from 0-7200 mm/min.
	RAPID OVERRIDE	Rapid Feedrate Adjustment Override Knob Adjust the Tool Rapid Move Speed from 0%-100%.
	SPINDLE OVERRIDE	Spindle Speed Adjust Override Knob Adjust the Spindle speed from 50%-120%.

## 3.2 Operation Mode Selection Knob

MODE SELECT Knob includes two separate blocks:

- (1) Automatic Modes Block
- (2) Manual Modes Block



### 3.2.1 Automatic Mode Area Description

Mode	Name	Description
	EDIT Mode	Edit and search the CNC program of Controller.
	DNC Mode	Operation while reading a program from external input/ output device.
	AUTO Mode	Auto-run the CNC program in the Controller.
	MDI Mode (Manual Data Input)	Operation by executing a program manually inserted from the MDI Panel.

### 3.2.2 Manual Mode Area Description

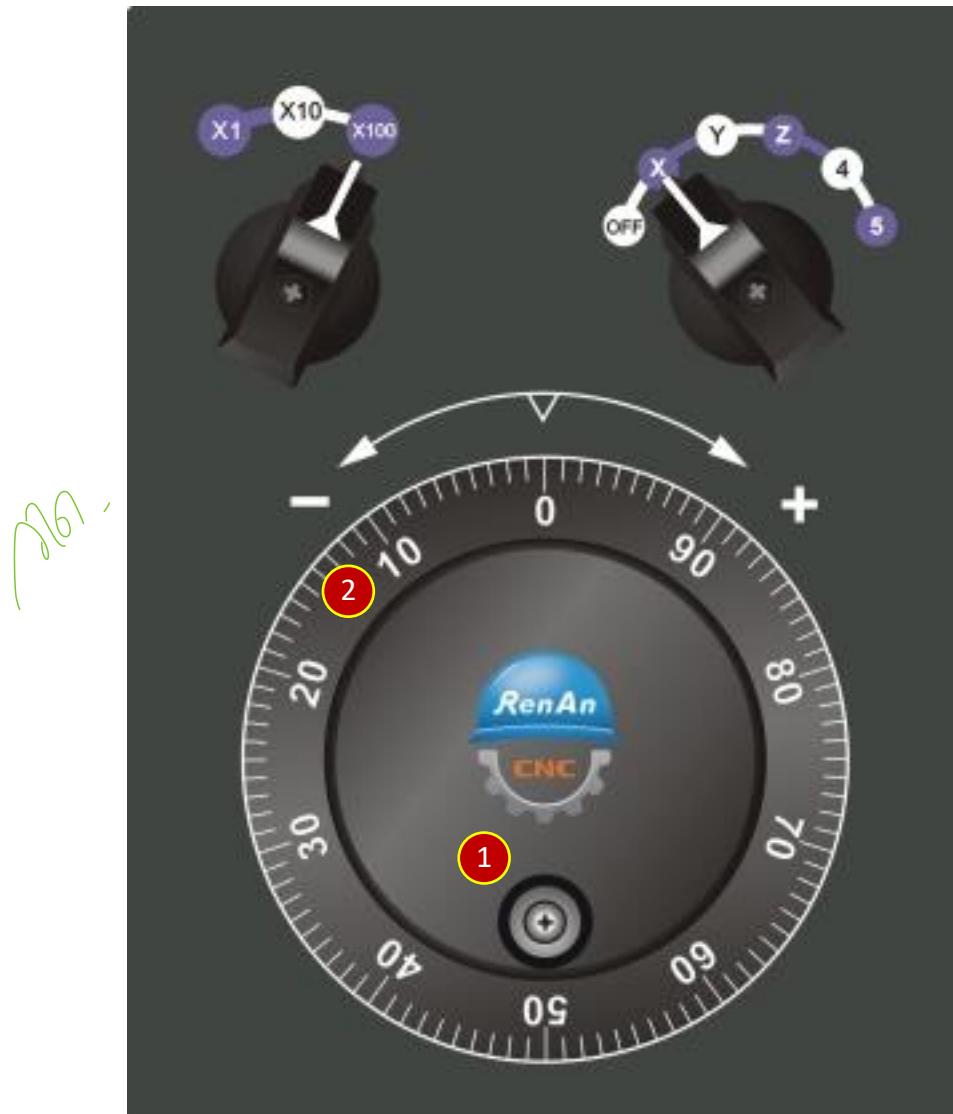
Mode	Name	Description
	HANDLE Mode	Use [( Handwheel )] to operate manual axial Movement.
	JOG Mode	Use [( Axial Movement )] Buttons to manually control continuous feeding.
	RAPID Mode	Use [( Axial Movement )] Buttons to manually control rapid movement of Tool and.
	ZRN Mode (Zero Return Mode)	Return Tool to Machine Zero Point.

## 3.3 HANDLE Operation (Handwheel Feed)

In [( HANDLE )] mode, manually rotate [( Handwheel )] to operate tool movement.

### 3.3.1 HANDLE Override Description:

- (1) Rapid adjustment: Quickly rotate the handle of [( Handwheel )] for rapid movement.
- (2) Accurate adjustment: Hold the outer ring of [( Handwheel )] and rotate slowly for fine adjustment.





### 3.3.2 Description of Handwheel Axis Selection Knob:

Handwheel Axis Selection Knob	Description
	Set [( Handwheel )] movement on X Axis.
	Set [( Handwheel )] movement on Y Axis.
	Set [( Handwheel )] movement on Z Axis.

### 3.3.3 Description of Handwheel Feedrate Knob:

Handwheel Feedrate Knob	Description
	Set the acceleration rate as [( x1 )] and each scale will be 0.001mm. (The basic unit of the Controller is 0.001mm)
	Set the acceleration rate as [( x10 )] and each scale will be 0.01mm.
	Set the acceleration rate as [( x100 )] and each scale will be 0.1mm.

### 3.4 Common Buttons Description

#### 3.4.1 Common Buttons Category

- (1) Auto Run operation function buttons
- (2) Manual operation function buttons
- (3) Other assisted function buttons



#### 3.4.2 Description of Auto Run Operation Function Button:

Button	Name	Description
	S.B.K	When the button light is on, only one block will run at a time then paused.
	M01	When the button light is on, program will be paused at 'M01'.
	B.D.T	When the button light is on, the NC code in the block after '/' will not execute

	CYCLE START	Start the program and run automatically.
	FEED HOLD	Pause tool cutting movement.
	Emergency Stop	Emergency stop button.

### 3.4.3 Description of Manual Operation Function Button Area:

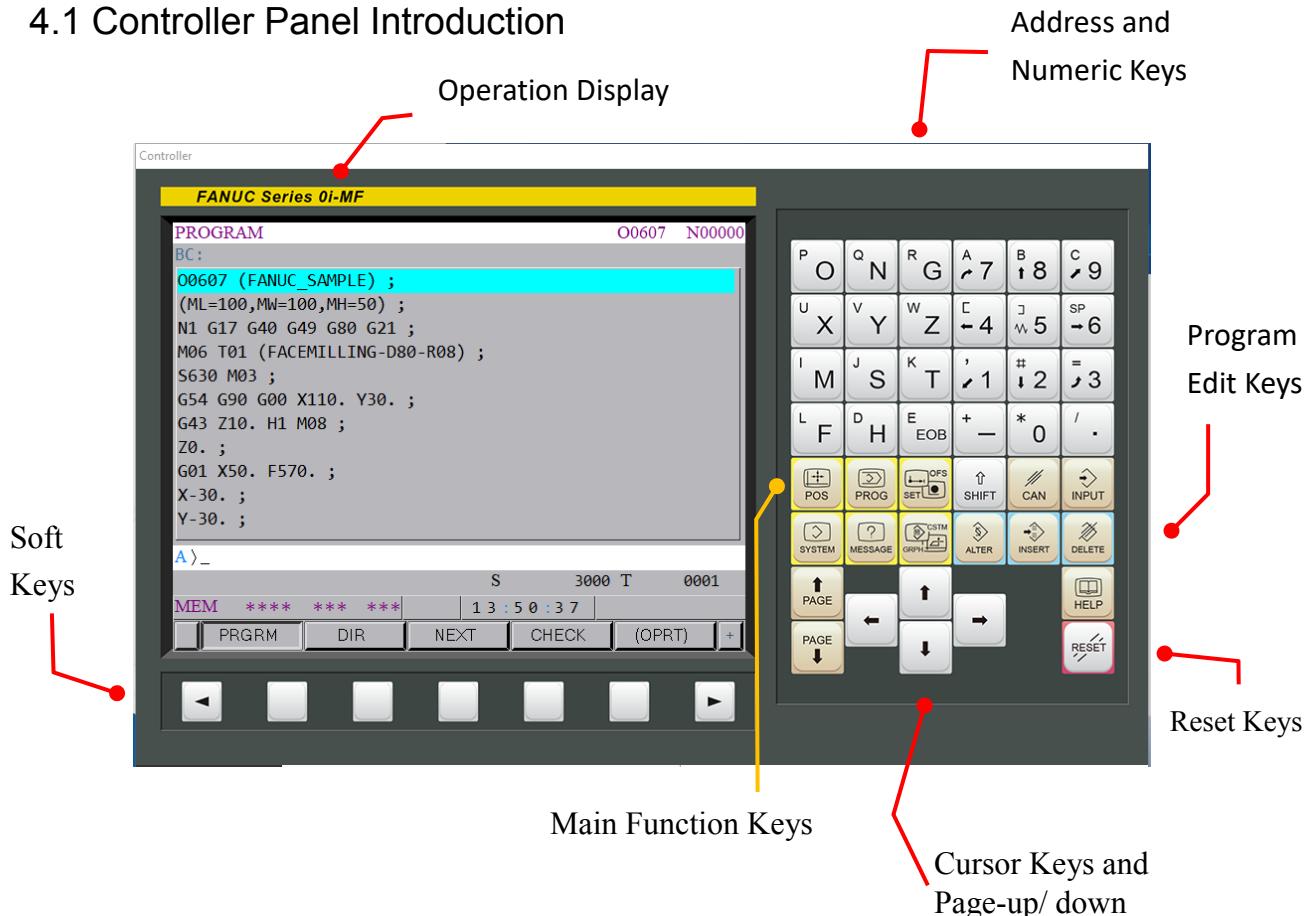
Button	Name	Description
	+X	Tool move toward ‘ +X ’ direction relatively. (Machine Bed move to the Left(-X))
	-X	Tool move toward ‘ -X ’ direction relatively. (Machine Bed move to the Right(+X))
	+Y	Tool move toward ‘ +Y ’ direction relatively. (Machine Bed move to the Front(-Y))
	-Y	Tool move toward ‘ -Y ’ direction relatively. (Machine Bed move to the Back(+Y))
	+Z	Tool move toward ‘ +Z ’ direction.
	-Z	Tool move toward ‘ -Z ’ direction.
	Spindle Forward	Spindle rotate counterclockwise (CCW).
	Spindle Stop	Spindle rotate stop.
	Spindle Reversal	Spindle rotate clockwise (CW).

3.4.4 Other Assisted Function Button:

Button	Name	Description
	COOLANT	Coolant On/Off
	LIGHT	Work light On/Off
	O.T.REL	Release Tool after Tool collision or when unable to retract Tool
	DOOR	Safety Door open/ close
	Program Protect	Program protection, lock and cancel.

## 4. Controller Panel Operation

### 4.1 Controller Panel Introduction



#### 4.1.1 Controller Panel Description

Item	Description
Operation Display Area	Display various of operation data. Operator can inspect various kinds of data.
Soft Keys Area	<p>&lt; PRGSRH N SRH REWIND P TYPE Q TYPE +</p> <p>◀ Return Menu Key, Upper Level Functional when ‘&lt;’ appears in left corner of screen ▶ Continuous Menu Key, Next Page Functional when ‘+’ appears in right corner of screen</p> <p>(OPRT) Operation Key, to display operation options</p>



[ RESET ] Key	 Use to reset data or NC status (1) Clear Alarm (2) Manually pause the auto-run program (3) Move cursor to the beginning of the program in EDIT mode
Main Function Keys	 Display Position Function  Display Program Function  Display Offset and Setting Function  Display System Parameter Function  Display Message/Alarm
Program Edit Keys	 Alteration: Change the Program Content  Insertion: Add NC Program Content  Deletion; Delete Program Content  Cancelation: Clear temporary memory
Address and Numeric Keys	Key-in letters and numbers  Press [ EOB ] to key in ‘ ; ’ to represent the end of a single block.
Cursor Keys Page Up Page Down	Control the cursor

# TM Fanuc 0i-MF Manual



[www.RenAn.com.tw](http://www.RenAn.com.tw)

## 4.2 Program Edit

#### 4.2.1 Enter Program Edit Mode

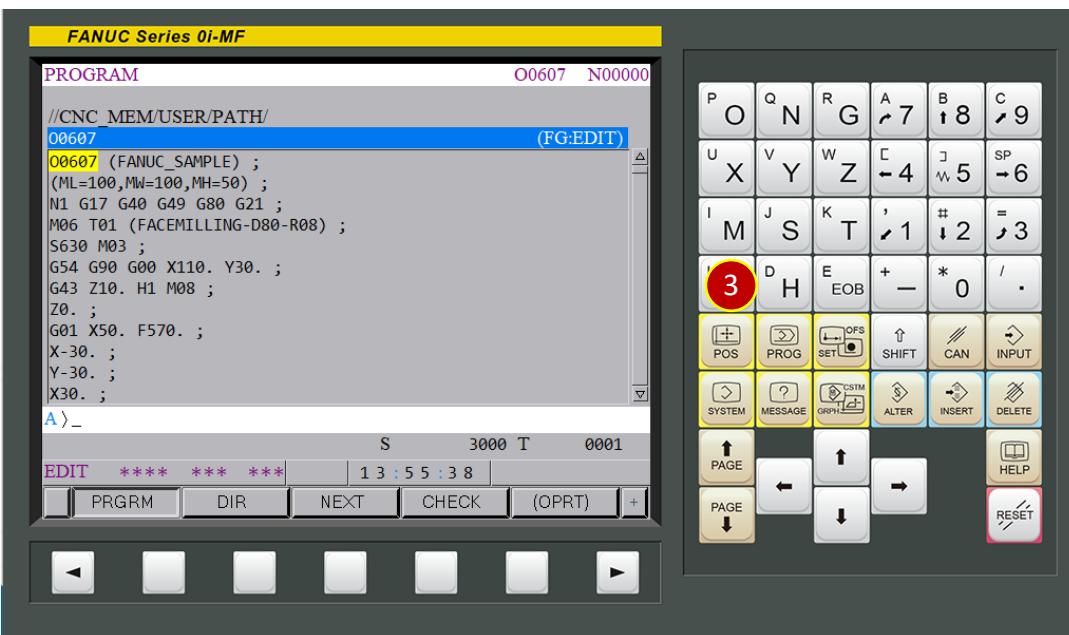
- (1) Switch to [( EDIT )] Mode.



- (2) Switch the program edit key  to editable status



- (3) Press [ PROG ] in Main Function Keys area to open Program Inspection Window.



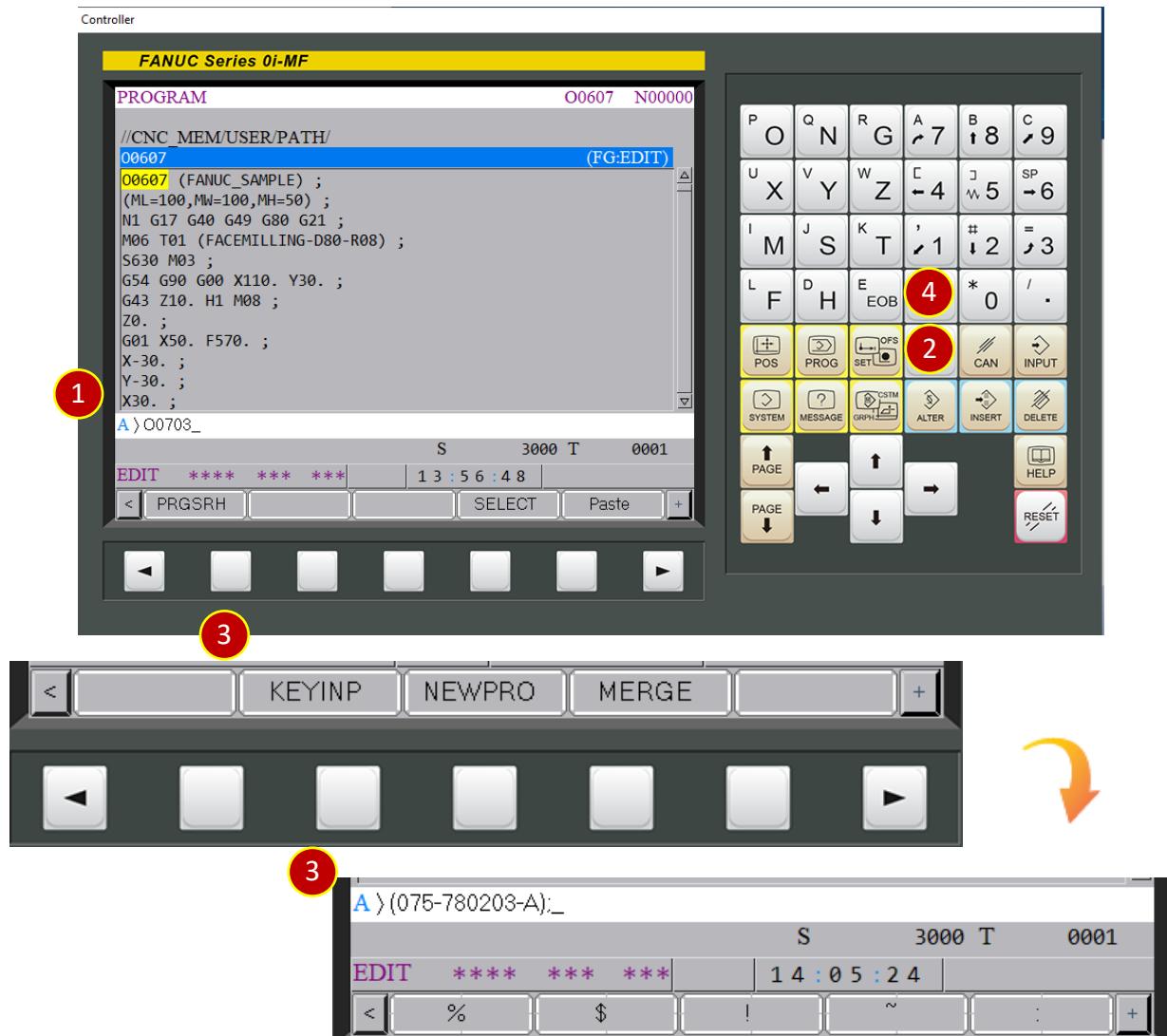
## 4.2.2 New Program Number and Comment

- (1) New program number O0703, Key in “ O ” first and key in number “ 0703 ”.
- (2) Press [ INSERT ] Key, to open new program
- (3) Key in the comment, e.g., “ (075-780203-A) ”

Note: CNC Controller would regard the words in parenthesis as comment and will not execute.

Press [ OPRT ] > Press (twice) > Press [ KEYINP ] to find the parenthesis

- (4) Press [ CAN ] Key to erase characters backwards while keying in program

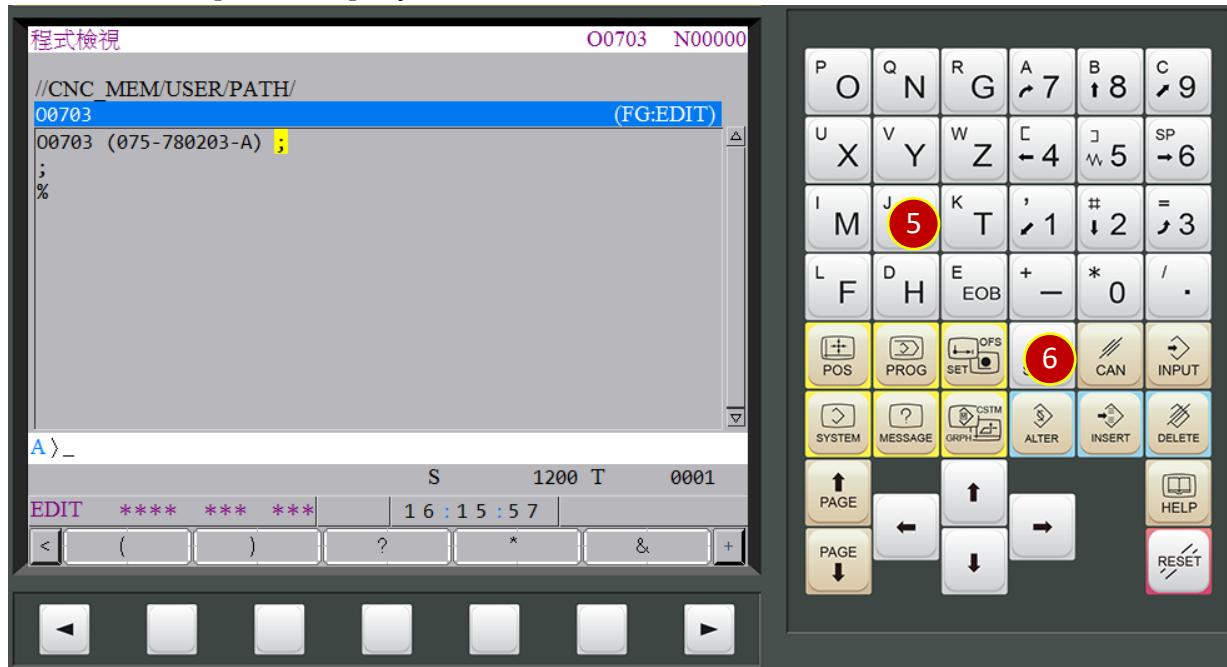


# TM Fanuc Oi-MF Manual



RenAn Information Technology Co., Ltd.  
[www.RenAn.com.tw](http://www.RenAn.com.tw)

- (5) Press [ EOB ] to key in “ ; ”
- (6) Press [ INSERT ] key to insert comment after the cursor



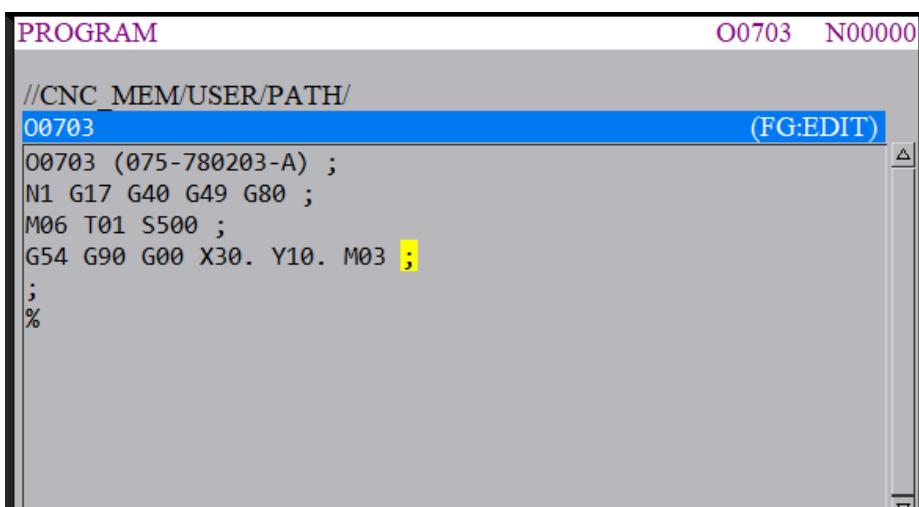
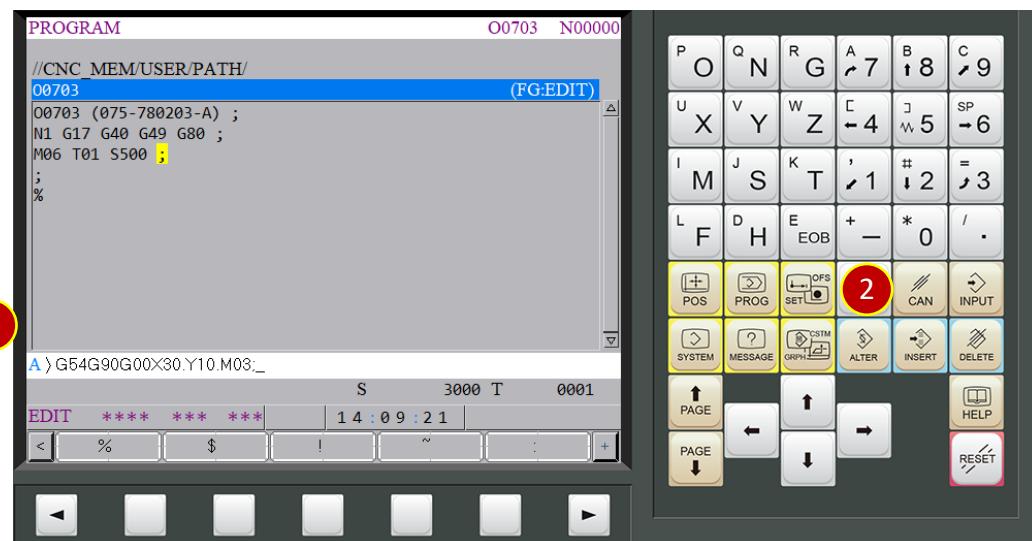
## 4.2.3 Add New Program Content

- (1) Move cursor to key in the NC code in the program content
- (2) e.g., “ N1 G17 G40 G49 G80; M06 T01 S500 ”

```
A ) N1G17G40G49G80;M06T01S500;_
S      3000  T      0001
```

- (3) After key in the NC code above, press [ INSERT ] to insert the NC code
- (4) Continue to key in NC code

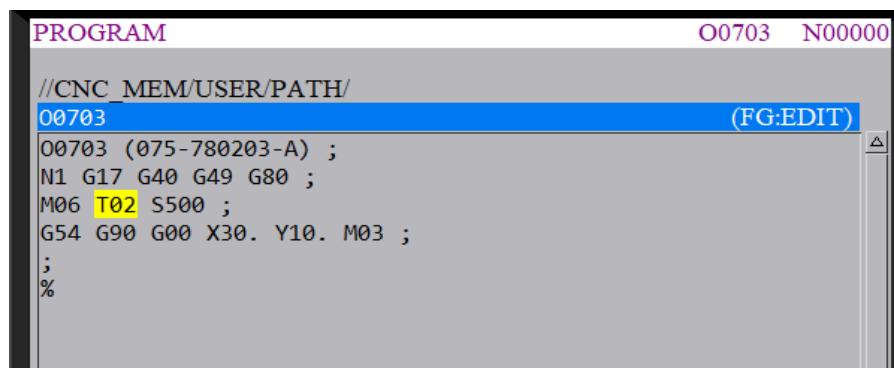
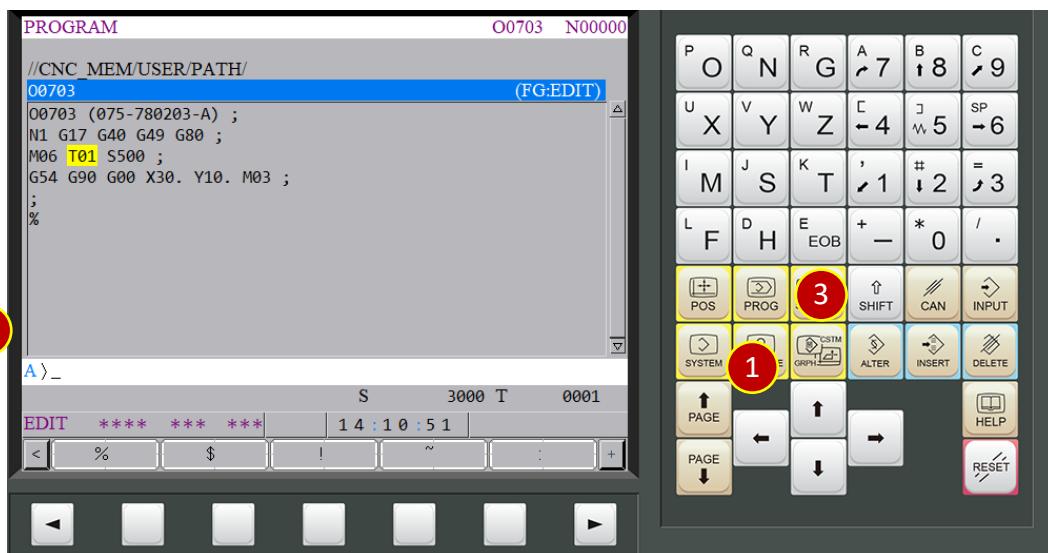
e.g., “ G54 G90 G00 X30. Y10. M03 ”



## 4.2.4 Change Program Content [ ALTER ]

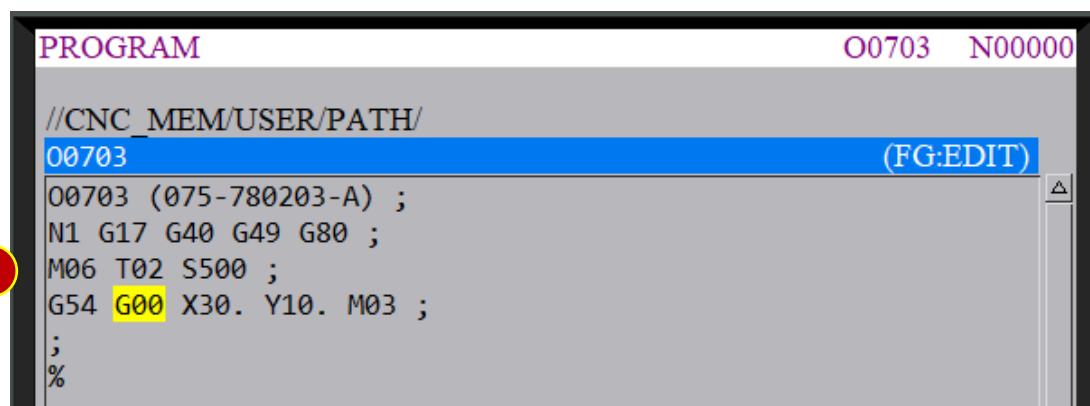
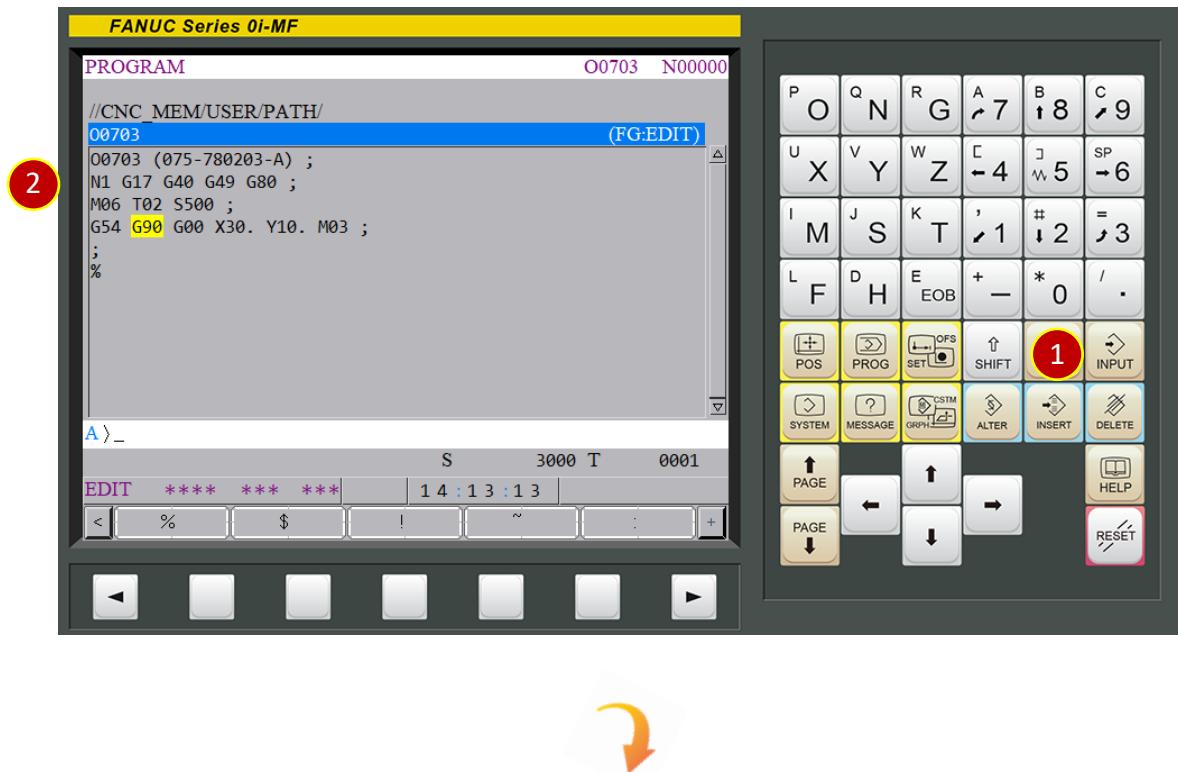
e.g., Change T01 to T02

- (1) When the program content needs to be changed, move cursor to the intended NC code.
- (2) Key in new NC code “ T02 ”
- (3) Press [ ALTER ] key and NC code T01 will be replaced by T02



## 4.2.5 Delete Program Content [ DELETE ]

- (1) Press [ DELETE ] to delete the program content
- (2) Delete the code that is selected by cursor



## 5. Transmission Parameter Setting and Operation

We offer RS232 in the Training Machine(25PIN) that can connect with external device (Such as PC) and transmit data.



### 5.1 RS232 COM Setting

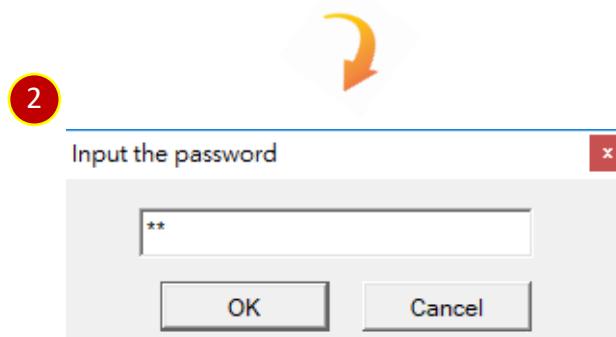
Designate the COM Port in Training Machine to the Controller.

RS232 connection port COM3 is the default setting used in data transmission.

Please follow the steps below when the connection port needs to be changed.

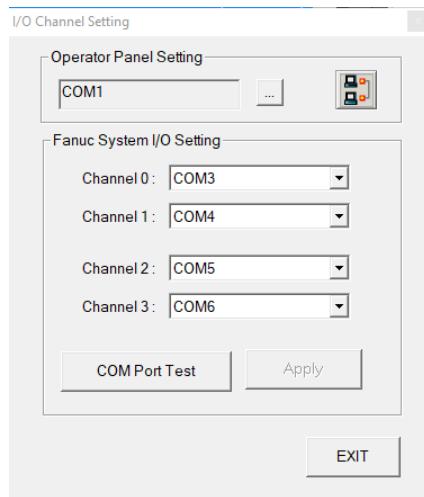
#### 5.1.1 Open the setting interface of RS232

- (1) Press [ I/O Setting ]
- (2) Key in the password aa and press [ OK ]





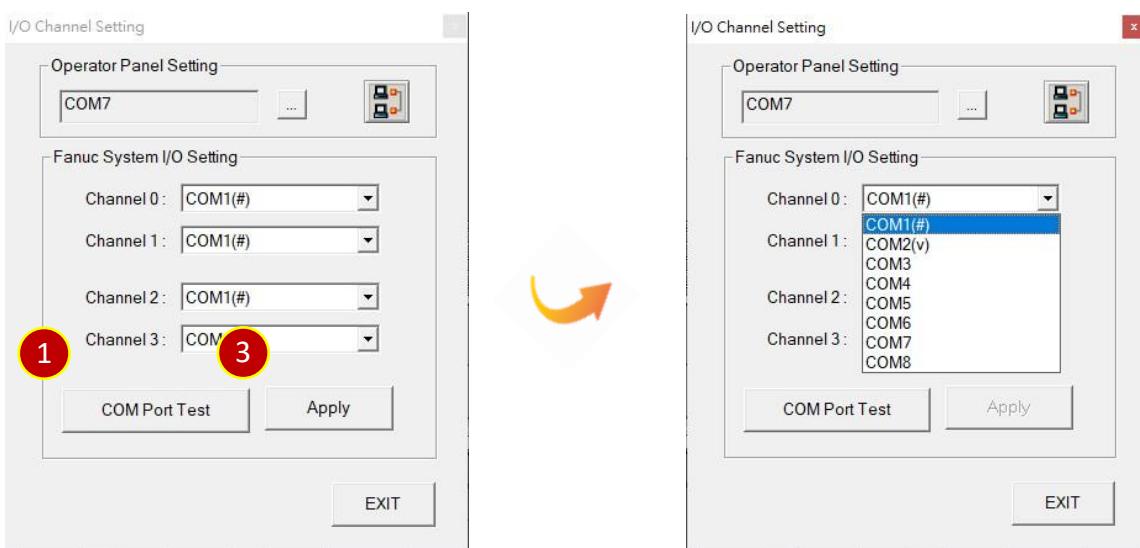
## 5.1.2 RS232 Connection Interface Setting



## 5.1.3 RS232 Connection Setting Procedure

- (1) Press [ COM Port Test ], auto-inspect the condition of each COM Port and the result will show on the list
 

COM1(V), ( V )= Available Port, not connected  
 COM2 , ( )= Not available (or being used by other devices)  
 COM3(#), ( # )= Available Port, connected
- (2) Select the available and connected COM Port  
 e.g., < CHANNEL 0 > , select [ COM1(#) ]  
 It means COM3 is used and is available for CHANNEL 0 to transmit
- (3) Press [ Apply ]
- (4) Press [ Exit ]



## 5.2 Transmission Parameter Setting

Set CHANNEL 0 as a default channel and set transmission parameter as

Code Type = ISO	Parity Check = None
Baud Rate = 4800	
Data bits = 8	Stop Bit = 1

### 5.2.1 Cancellation of Parameter Input Protection

After Cancellation of Parameter Input Protection,  
transmission parameter can be modified.

Steps:

(1) Switch to [( MDI )] Mode

(2) Switch [( Program Edit Lock )] to the editable status

(3) Press [ OFS/SET ]

(4) Press [ SETTING ]

(5) Select [ PARAMETER WRITE ]

(6) Key in “ 1 ”

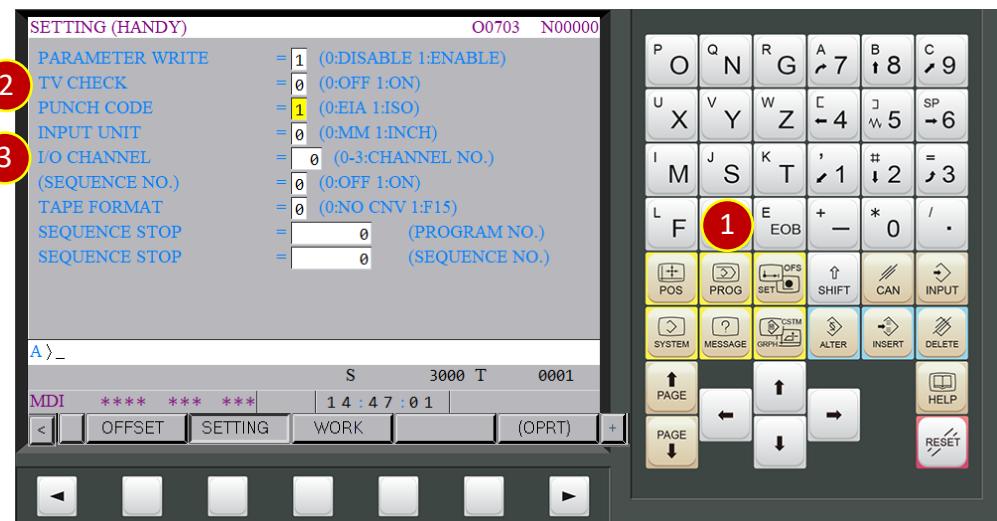
(7) Press [ INPUT ]



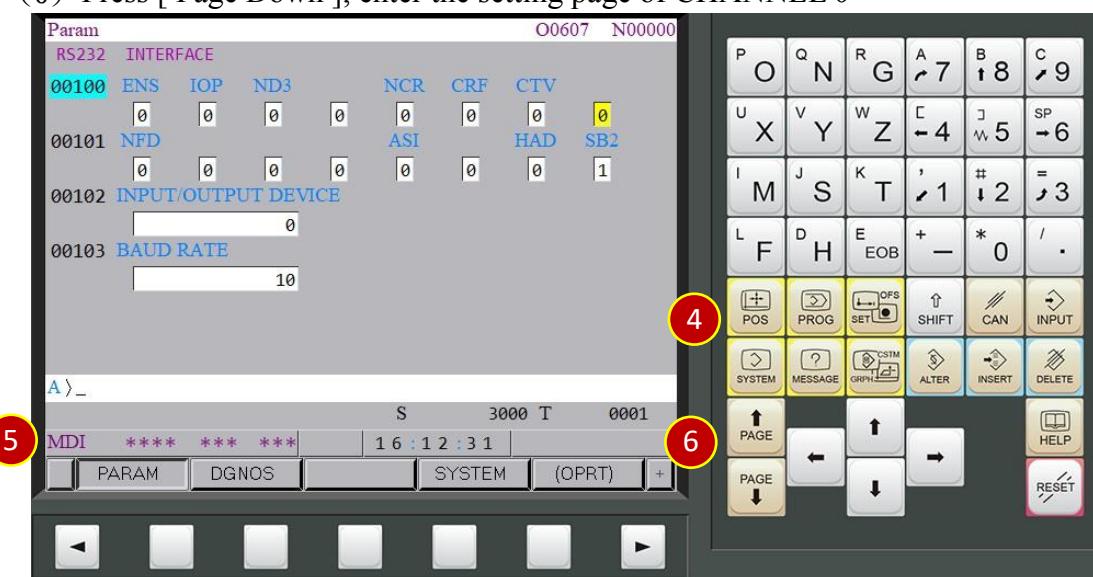
## 5.2.2 Parameter Modification

Make the transmission parameter in the Controller and the transmission parameter in the external device the same to transmit the programs

- (1) Press [ OFS/SET ] to show the interface of parameter setting
- (2) PUNCH CODE: assign transmission code type  
e.g., PUNCH CODE= 1. (0: EIA, 1: ISO)
- (3) [ I/O CHANNEL ]: assign the channel when transmitting  
e.g., I/O CHANNEL= 0. (0: CHANNEL 0, 1: CHANNEL 1)



- (4) Press [ SYSTEM ], show the system parameter
- (5) Press [ PARAM ]
- (6) Press [ Page Down ], enter the setting page of CHANNEL 0



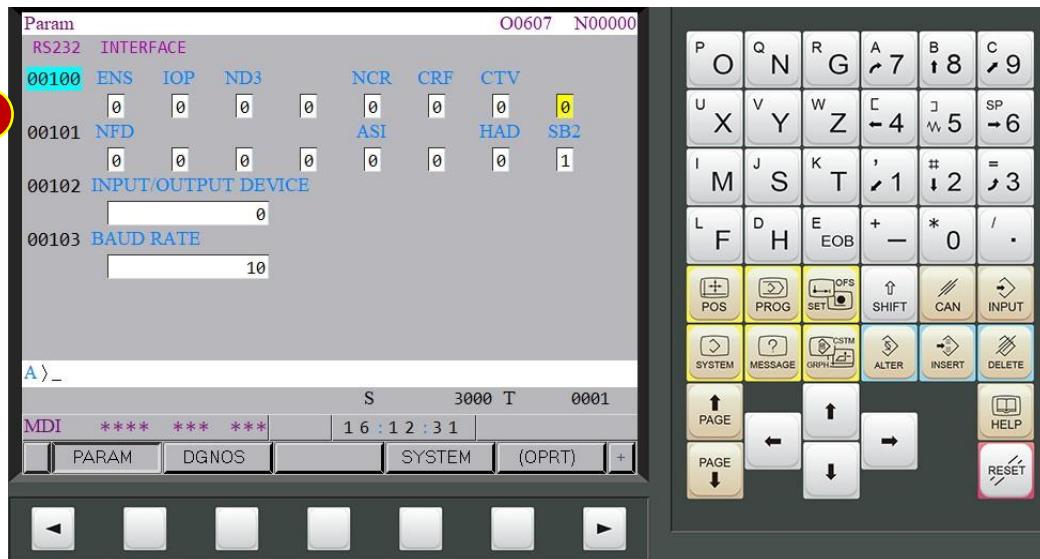
## (7) Set the transmission parameter of CHANNEL 0

The description of transmission parameter 00101:

NFD				ASI		HAD	SB2
0	0	0	0	0	0	0	0

SB2, specify the stop bit. (0: stop bit 1, 1: stop bit 2)

ASI, specify the transmission coding type. (0: Encoding type specified by PUNCH CODE, 1: ASCII)



00103 BAUD RATE. Transmission Baud Rate of CHANNEL 0

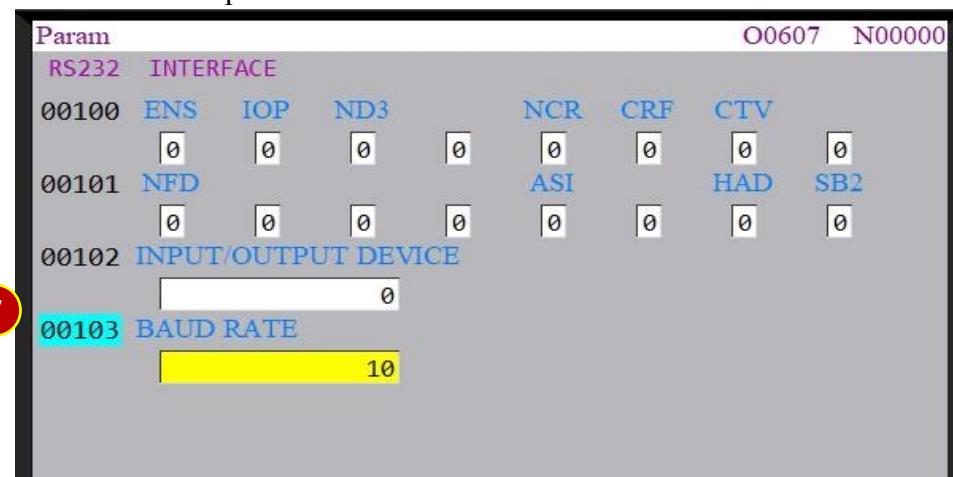
e.g.: BAUD RATE = 10

09: 2400 bps transmission

10: 4800 bps transmission,

11: 9600 bps transmission

12: 19200 bps transmission



# TM Fanuc Oi-MF Manual



- (8) Press [ OFS/SET ] after finishing the parameter setting
- (9) Press [ SETTING ]
- (10) Change [ PARAMETER WRITE ] to “ 0 ” (DISABLE)



## 5.3 Receive Program from the Controller

Open the software NcEditor by PC and send the program to training machine

### 5.3.1 Operation Steps

(1) Open up NcEditor to wait for the transmission

(2) Switch to [( EDIT )] Mode



(3) Switch the [( Program Edit Lock )] to the editable status



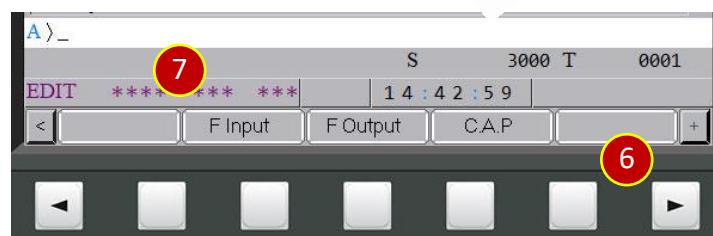
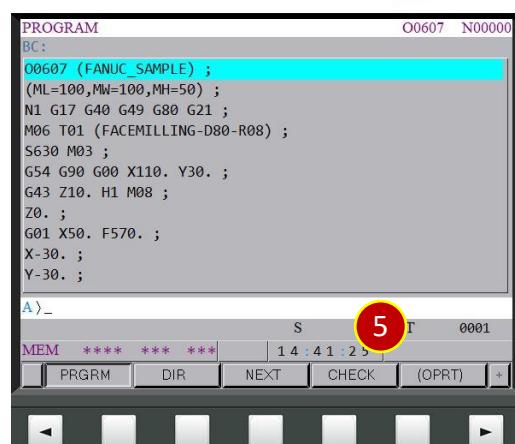
(4) Press [ PROG ]

(5) Press [ (OPRT) ]

(6) Press [ ] Button 4 times to switch menu.

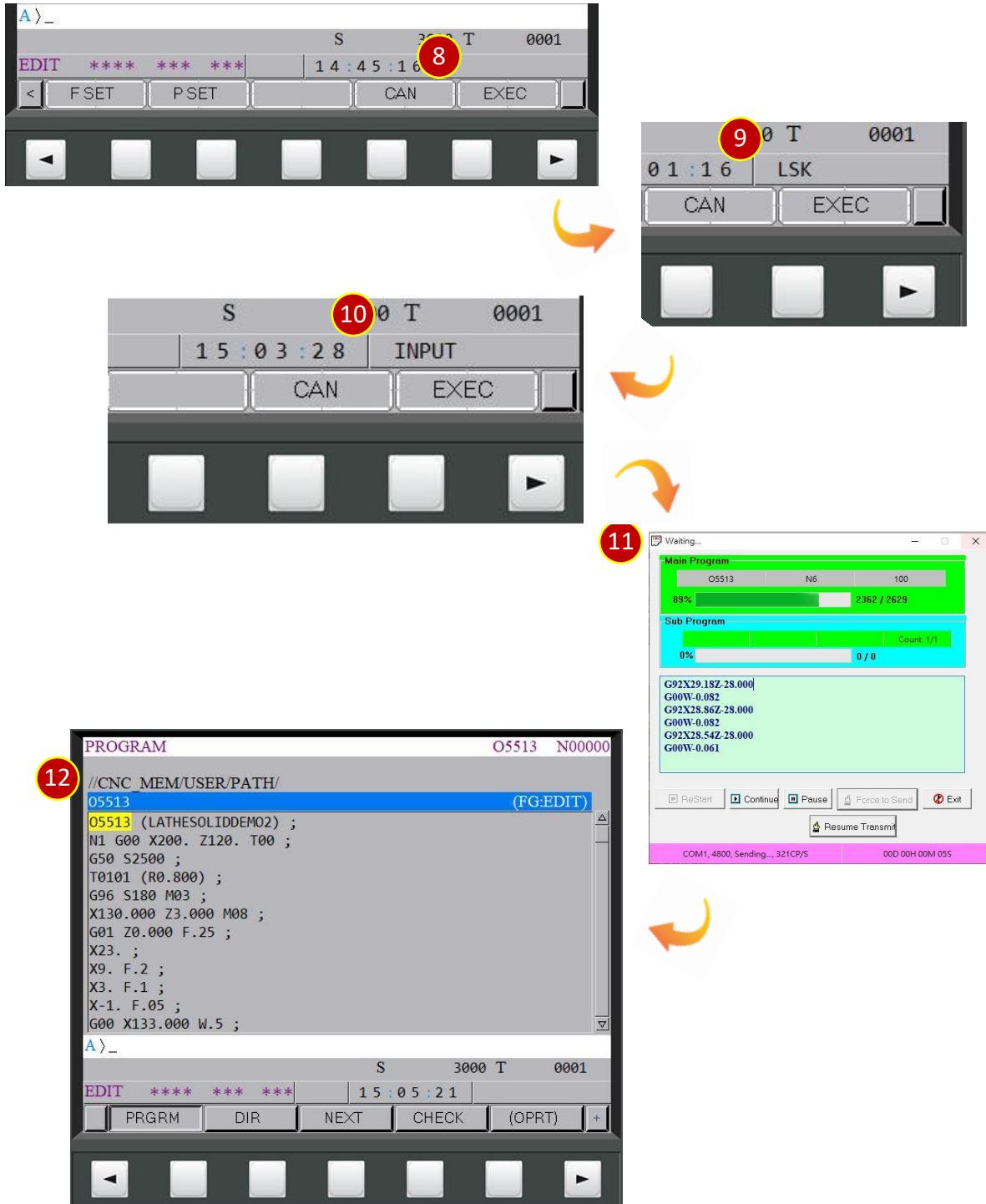
(7) Press [ F Input ] to receive the program

1



6

- (8) Press [ EXEC ], receive the program and NcEditor will send the program by auto notification
- (9) ‘ LSK ’ indicates waiting to receive the program
- (10) ‘ INPUT ’ means receiving the program
- (11) NcEditor will show the transmission process
- (12) The program shows automatically after finished receiving



# TM Fanuc 0i-MF Manual



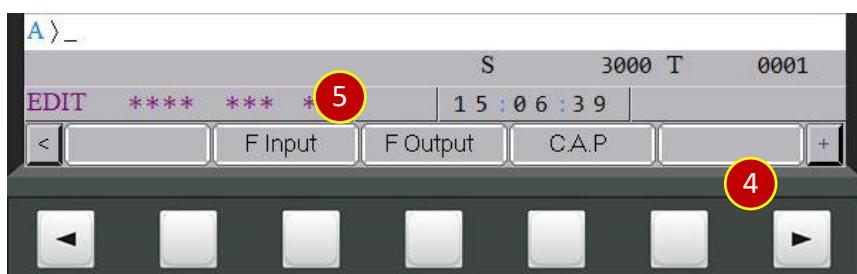
## 5.4 Send Program from the Controller

Send the program in the Controller to PC

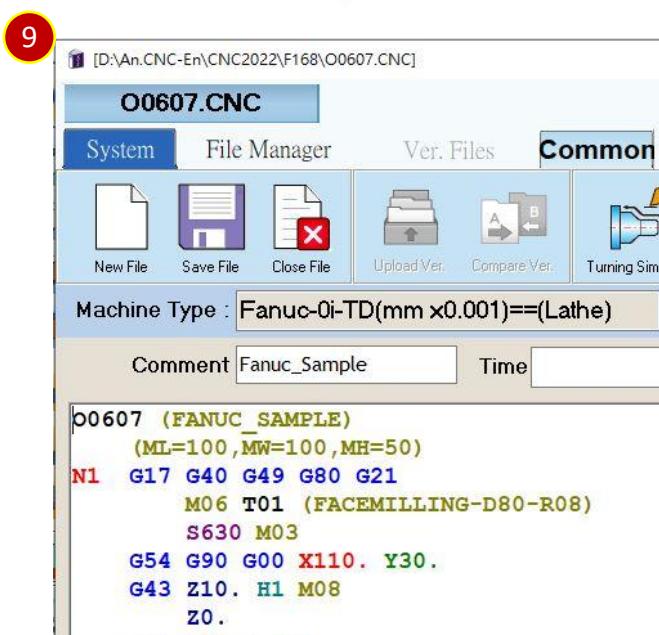
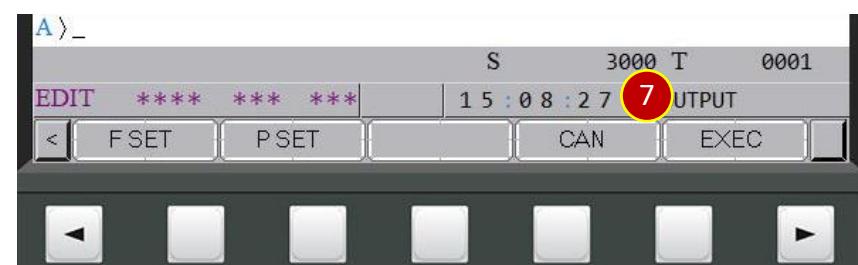
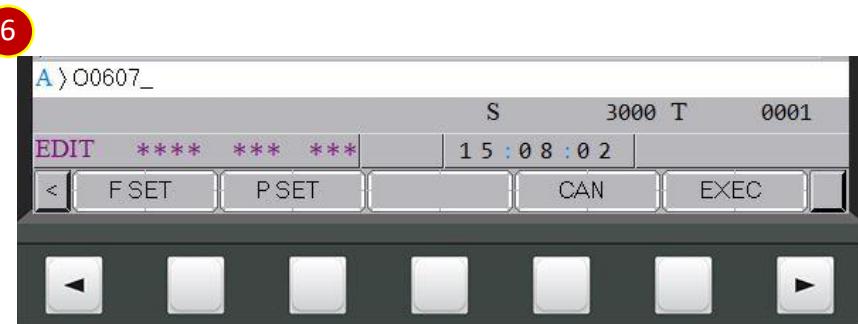
#### 5.4.1 Operation Steps

- (1) Switch to [( EDIT )] Mode
  - (2) Press [ PROG ] 
  - (3) Press[ (OPRT) ] 
  - (4) Press [  ] Button 4 time
  - (5) Press [ F Output ] 

PROGRAM O5513 N00000  
//CNC\_MEM/USER/PATH/  
05513 (FG:EDIT)  
05513 (LATHE SOLID DEMO 02) ;  
N1 G00 X200. Z120. T00 ;  
G50 S2500 ;  
T0101 (R0.800) ;  
G96 S180 M03 ;  
X130.000 Z3.000 M08 ;  
G01 Z0.000 F.25 ;  
X23. ;  
X9. F.2 ;  
X3. F.1 ;  
X-1. F.05 ;  
G00 X133.000 W.5 ;

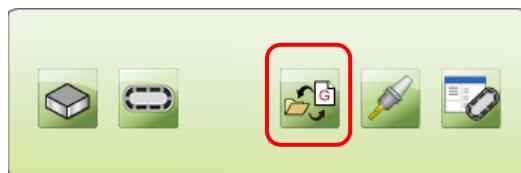


- (6) Key in the program O code to be uploaded  
e.g.: "O0607"
- (7) Press [ EXEC ] to start transmission
- (8) 'OUTPUT' shows up, indicating the program is sending and NcEditor is receiving
- (9) Program transmission complete. NcEditor shows the program content

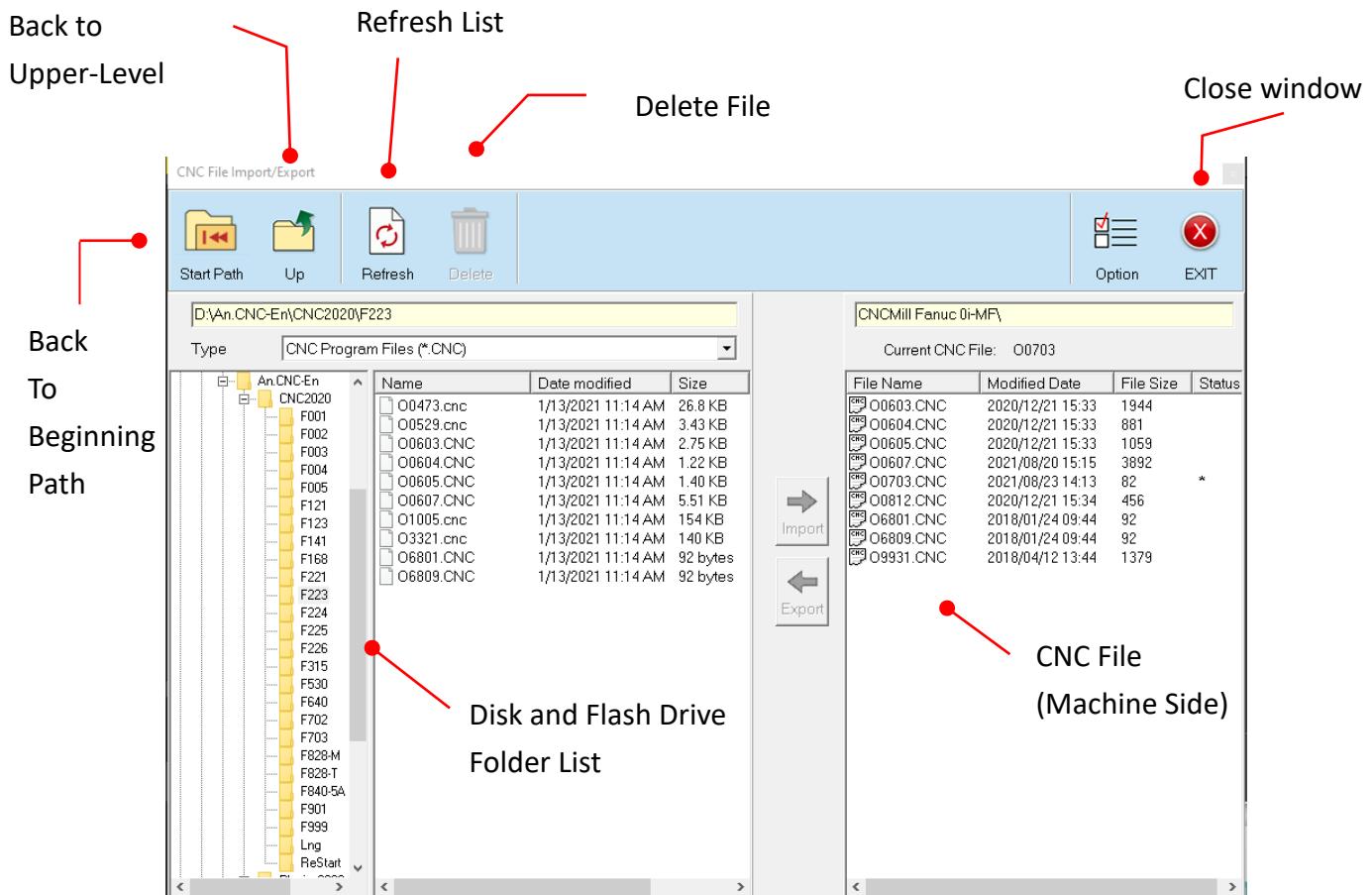


## 5.5 CNC File Import/Export Management

Export the NC files from training machine to disk or USB  
or import the file of disk or USB to training machine

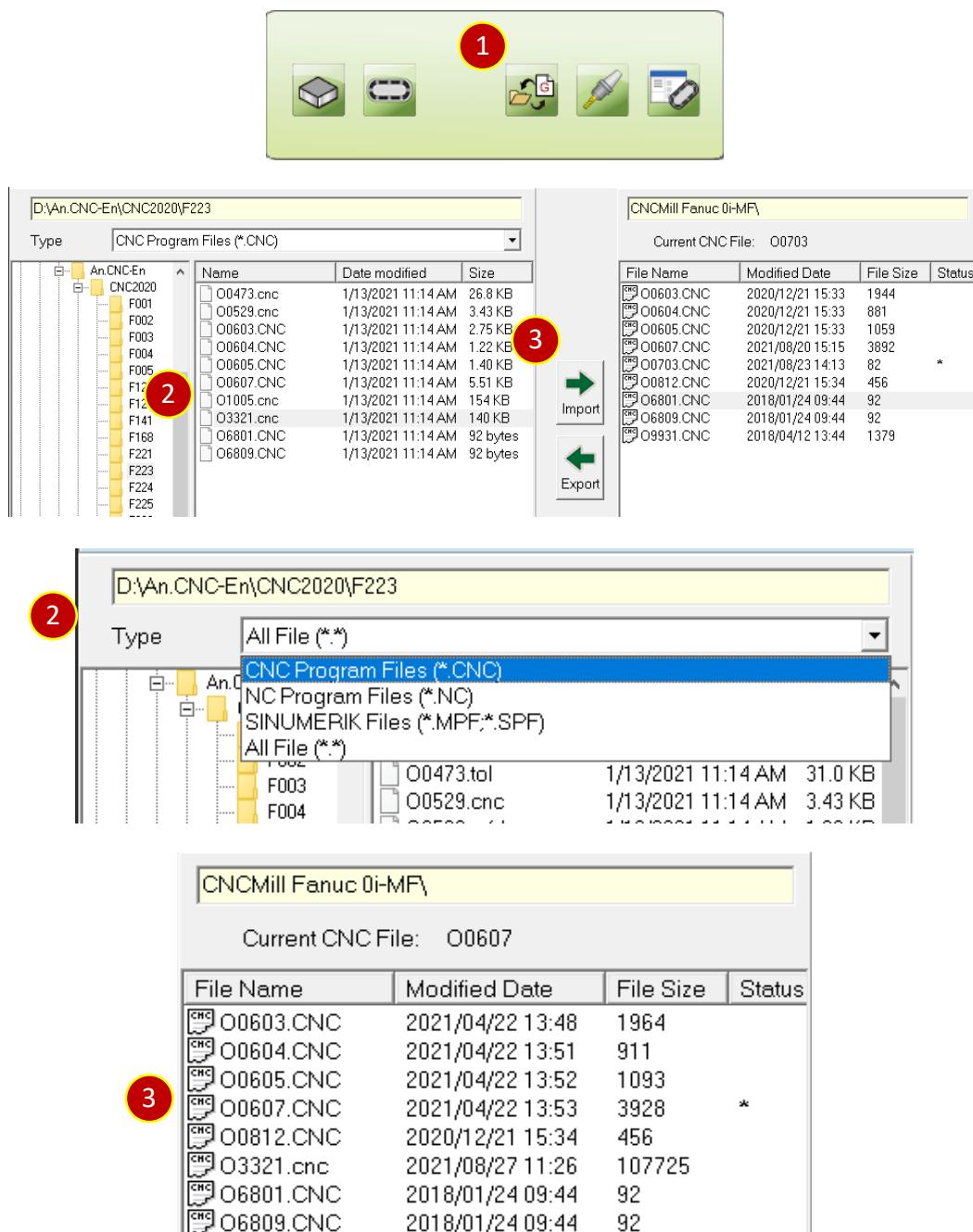


### 5.5.1 Open CNC File Import/ Export Function



## 5.5.2 Import CNC File

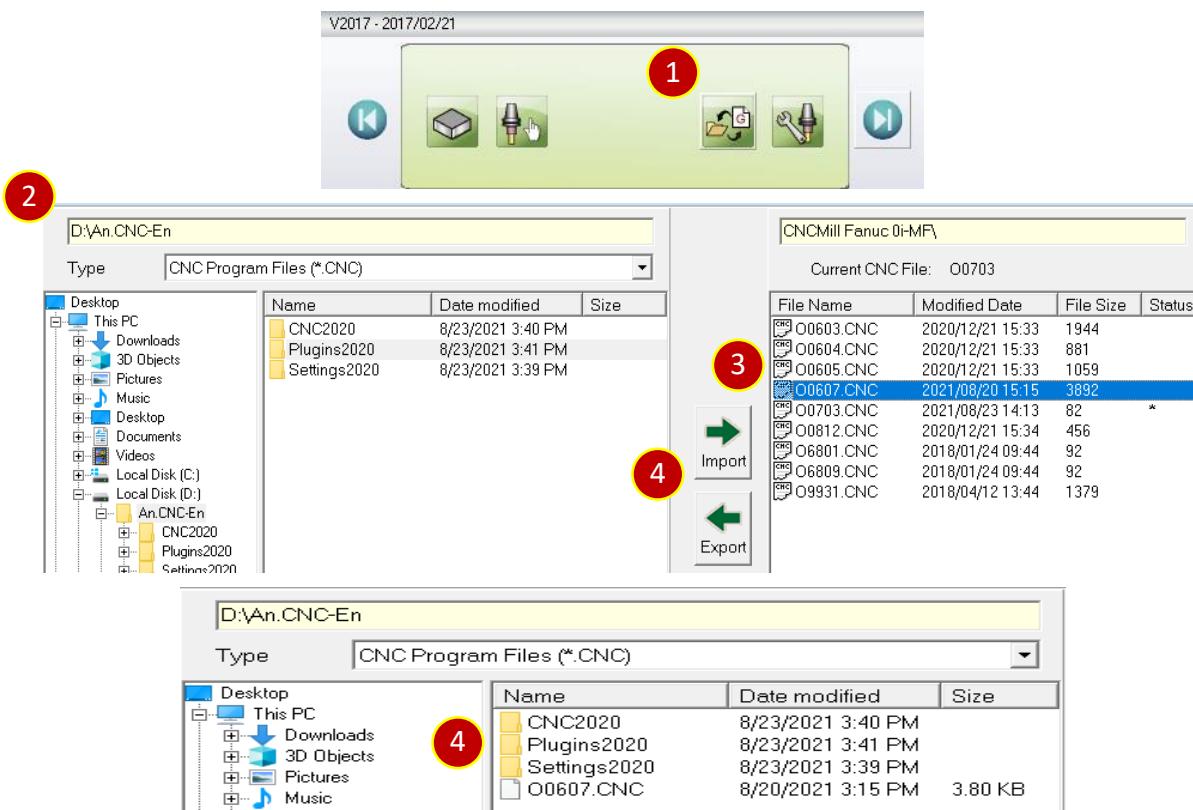
- (1) Press [ CNC File Import/ Export ]
- (2) Choose the CNC file from PC side, e.g., O3321.cnc  
Must select file type as [ CNC Program File(\*.CNC) ]
- (3) Press to import CNC file to machine side



## 5.5.3 Export CNC File

- (1) Press [ CNC File Import/Export ]
- (2) Select the exporting CNC folder from the menu of left side  
e.g., C:\CNCprogram
- (3) Select the exporting CNC file from the menu of right side e.g.: O0607

(4) Press to export the CNC file



## 6. Basic Machine Operation

## 6.1 ZRN Operation

After turning on the machine, return each axis to Reference Point to set the actual reference position for each axis as basis for the following machining and Auto Run.

### 6.1.1 Operation Steps

- (1) Press [ POS ] to display the coordinate function button.
  - (2) Press [ ALL ] to display all the coordinate values.
  - (3) Check machine coordinates values:

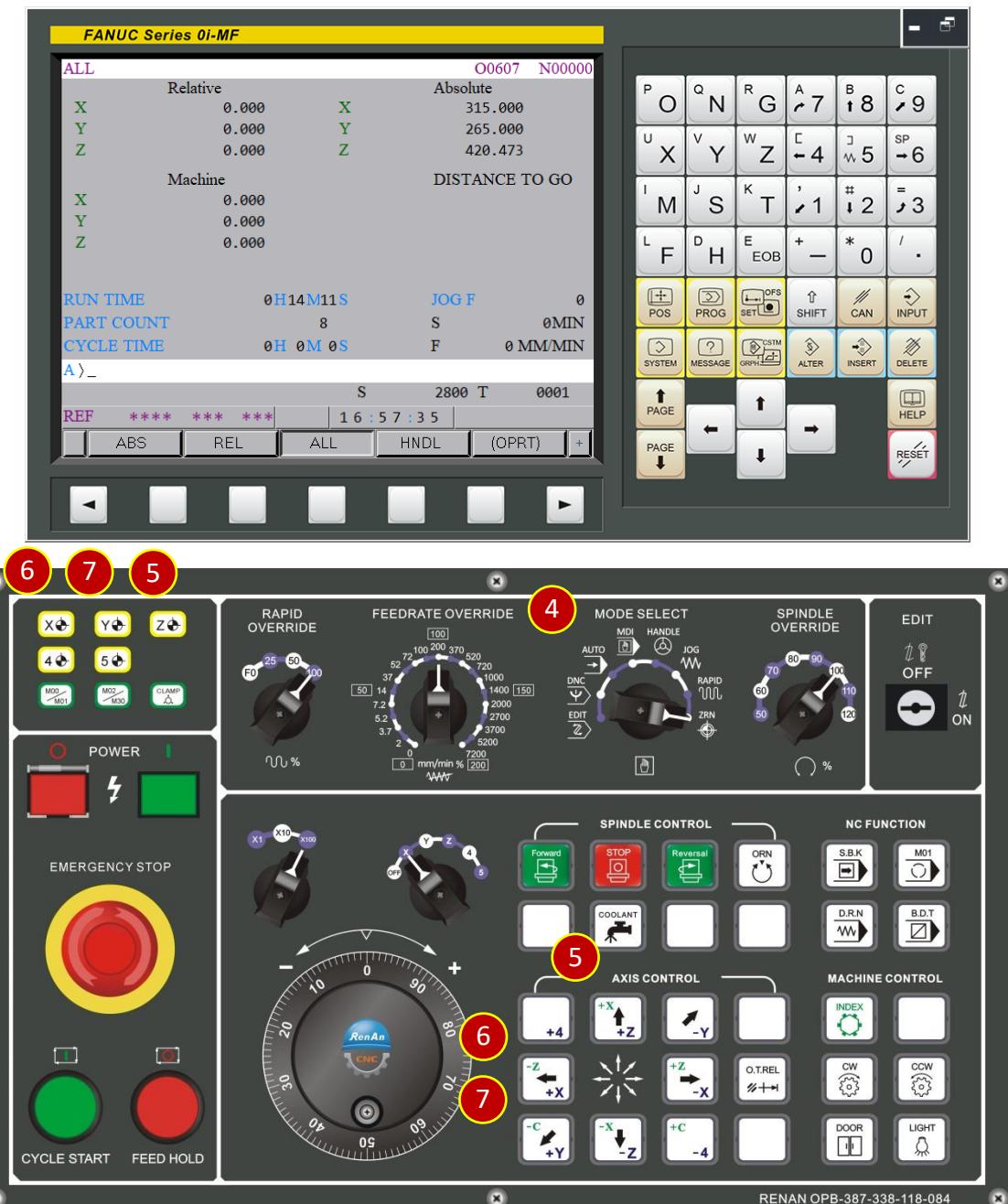
**RELATIVE:** The coordinate values based on the assigned position

#### **ABSOLUTE:** The current position in Work Coordinates

**MACHINE:** The coordinate values based on machine Reference Point as Zero Point reference



- (4) Switch MODE SELECT Knob to [( ZRN )] (Zero Return)
- (5) Press [( +Z )] in the Axial Movement Panel. The Z Axis starts to move toward Reference Point rapidly. When the Z Axis Machine Coordinate is 0 and the Z Axis ZRN is complete, the Z Axis ZRN indicator will light up.
- (6) Press [( X )]. X Axis starts to move toward Reference Point rapidly. When the X Axis ZRN is complete, the X Axis ZRN indicator will light up.
- (7) Press [( +Y )]. Y Axis starts to move toward Reference Point rapidly. When the Y Axis ZRN is complete, the Y Axis ZRN indicator will light up.

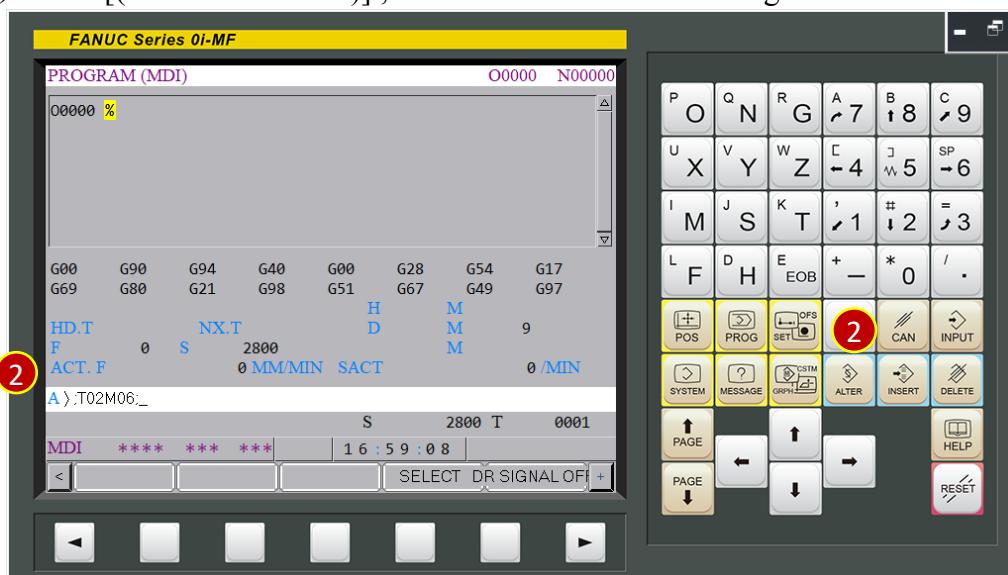


## 6.2 Manual Tool Change

Example: Change to Tool No.2 manually

### 6.2.1 Operation Steps

- (1) Switch MODE SELECT Knob to [(MDI)]
- (2) Key in “;T02 M06;”, press [ INSERT ]
  - (a) ‘ T02 ’ represents Tool No.2
  - (b) ‘ ; ’ represents the line break, press [ EOB ] to key in
- (3) Key in “ G43 H02; ” and press [ INSERT ] to use Tool Offset of Tool No.2
- (4) Press [( CYCLE START )] , the current Tool will be changed to Tool No.2



## 6.3 Edge Finder Operating Standard Procedures

### 6.3.1 Move the Edge Finder rapidly to the Workpiece

- (1) Change the Tool to the Edge Finder ( Tool No.99 ).
- (2) In [( RAPID )] Mode, use [( Axial Movement )] Buttons, move the Edge Finder to about 50mm from the left side of Workpiece.
- (3) Use [( Handwheel )] and switch Handwheel Feedrate to [( x100 )].
- (4) Rotate the [( Handwheel )] rapidly to approach the Workpiece until about 3mm away.

### 6.3.2 Steps of ‘ Precise Fine-Tune Edge Finding ’

- (1) Switch to [( HANDLE )] Mode and set Handwheel Feedrate to [( x100 )].
- (2) Place your fingers on the outer ring of the [( Handwheel )] and rotate to touch the Workpiece.
- (3) Rotate slowly one scale by one, stop moving forward when the light is on
- (4) When the [( Handwheel )] retracts one scale and the light is off, adjust the Handwheel Feedrate to [( x10 )].
- (5) Touch the Workpiece with Edge Finder again, stop when the light is on.
- (6) Continue adjusting [( Handwheel )] until they meet both of the following requirements:

The light turns off when the [( Handwheel )] retracts one scale

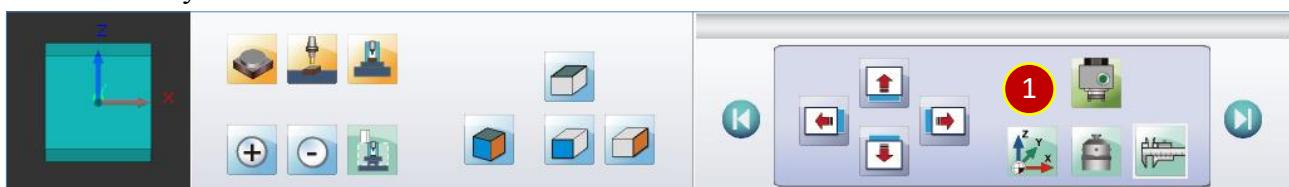
The light turns on when the [( Handwheel )] approaches one scale.



## 6.4 Z Axial Gauge Operating Standard Procedures

### 6.4.1 Move the Tool rapidly to the Z Axial Gauge

- (1) In Application Toolbar, press [ Z Axial Gauge ] to set the Z Axial Gauge onto the Workpiece.
- (2) In [( RAPID )] Mode, use [( Axial Movement )] Buttons, move the Tool to about 50mm above the Z Axial Gauge.
- (3) Use [( Handwheel )] and switch Handwheel Feedrate to [( x100 )].
- (4) Rotate [( Handwheel)] rapidly to approach the Z Axial Gauge until about 3mm away.



### 6.4.2 Steps of ' Precise Fine-Tune to Touch Z Axial Gauge '

- (1) Switch to [( HANDLE )] Mode and set Handwheel Feedrate to [( x100 )].
- (2) Place your fingers on the outer ring of the [( Handwheel )] and rotate to touch the Workpiece.
- (3) Rotate slowly one scale by one, stop moving forward when the light is on
- (4) When the [( Handwheel )] retracts one scale and the light is off, adjust the Handwheel Feedrate to [( x10 )].
- (5) Touch the Z Axial Gauge with the Tool again, stop when the light is on.
- (6) Continue adjusting [( Handwheel )] until they meet both of the following requirements:

The light turns off when the [( Handwheel )] retracts one scale

The light turns on when the [( Handwheel )] approaches one scale.

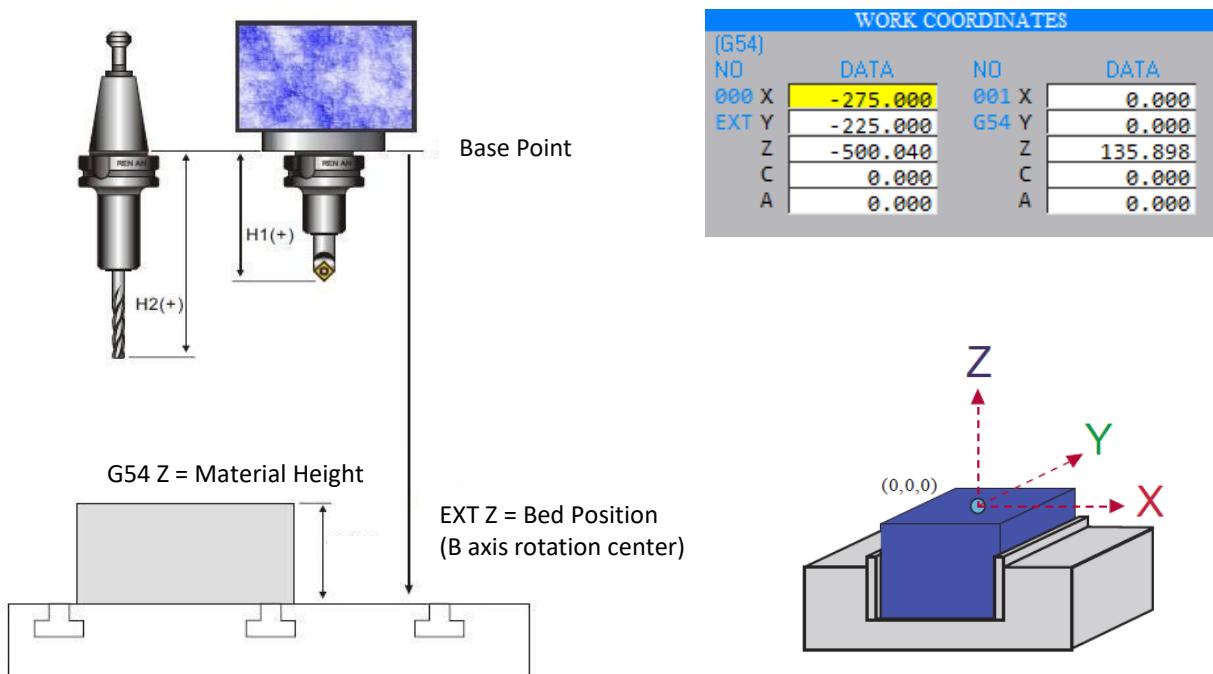


## 6.5 Work Coordinate Setting

### 6.5.1 Definition of Work Coordinate

It is necessary to set the correct Zero Point of Work Coordinate system to execute precise cutting while running CNC program. For milling, the Zero Point of Work Coordinate is normally at the top center point of Workpiece.

- (1) After changing to new Workpiece, the correct Zero Point of Work Coordinate system must be reset according to Workpiece dimensions.
- (2) After changing to new Tool, the Tool Geometry (Length) Offset value must be reset.  
(The figure which takes the spindle conical degrees major diameter as the reference Tool length)



EXT X = X Axis Machine Coordinate from Machine Zero Point to Machine Bed center  
 EXT Y = Y Axis Machine Coordinate from Machine Zero Point to Machine Bed center  
 EXT Z = The Machine Coordinate value from the bottom of Spindle Head to the Face of Machine Bed (B axis rotation center)

**(Note: The X, Y, Z of EXT are usually set as default. Do not change unless necessary)**

G54 X = Zero Point of the Work Coordinate system. The Machine Coordinate value of X Axis (usually minus)

G54 Y = Zero Point of the Work Coordinate system. The Machine Coordinate value of Y Axis (usually minus)

G54 Z = Material height, from Bed to Zero Point of Workpiece Geometry offset

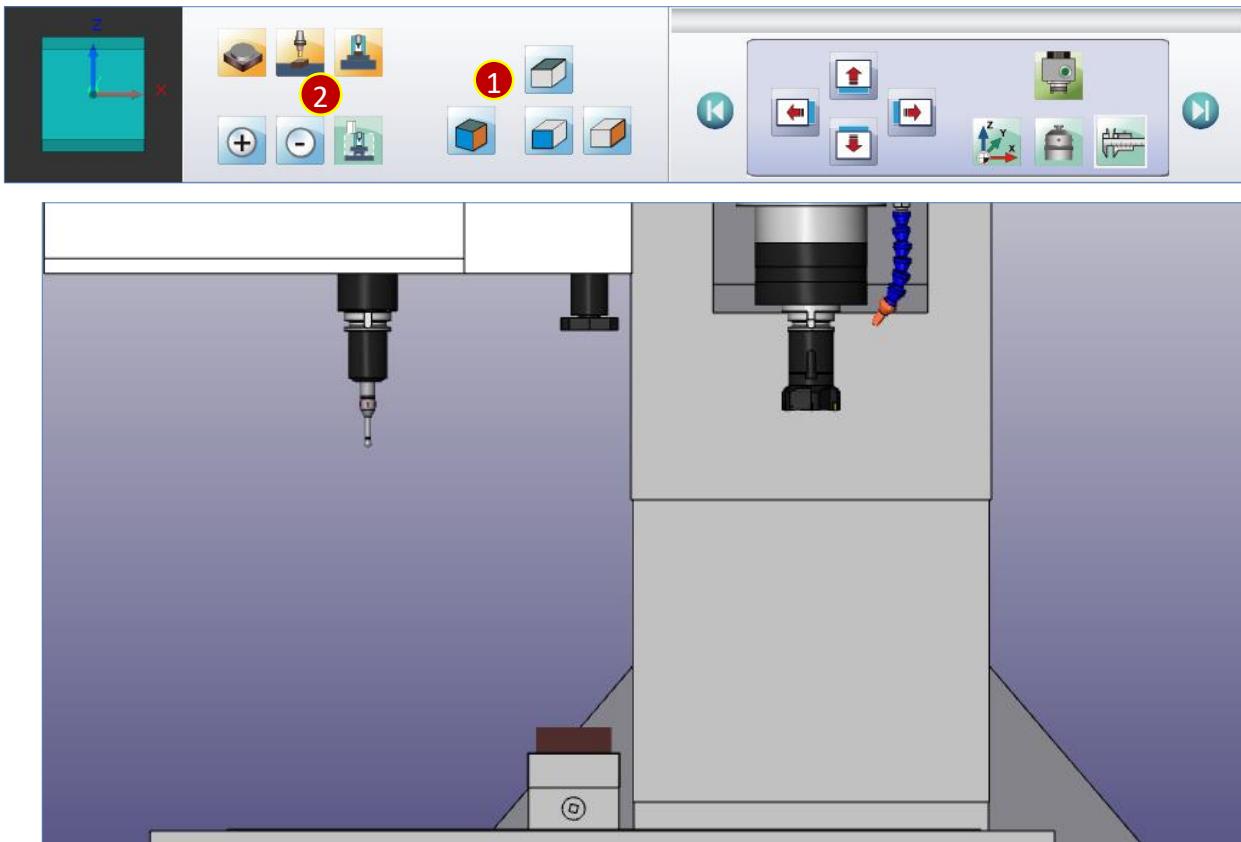
(H) = Tool length of each Tool

NO	LEN GEOM(H)
001	131.000
002	205.000
003	155.000
004	175.000
005	145.000
006	0.000
007	0.000
008	0.000

## 6.5.2 View Adjustment

Adjust view to proper angle and size for Tool Offset.

- (1) At the View Toolbar, press [ Front View ].
- (2) Press [ Show/Hide Case ] to hide the machine case.



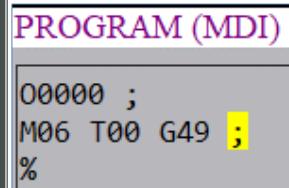
### 6.5.3 Work Coordinate Setting of Axis (Z Axis)

Use the Z Axial Gauge to measure the distance from the Machine Bed to the top of Workpiece to set the Z Axis Work Coordinate of G54.

Set Machine Bed as the Relative Zero Point, pick any Tool for measurement

Take Empty Tool T00 for example:

- (1) Switch MODE SELECT Knob to [(MDI)]
- (2) Press [ PROG ]
- (3) Key in “;M06 T00 G49;” switch to T00 and G49 clear Tool Offset
- (4) Press [ INSERT ] °
- (5) Press [( CYCLE START )] , the current Tool will be switched to T00, and Tool Offset cleared



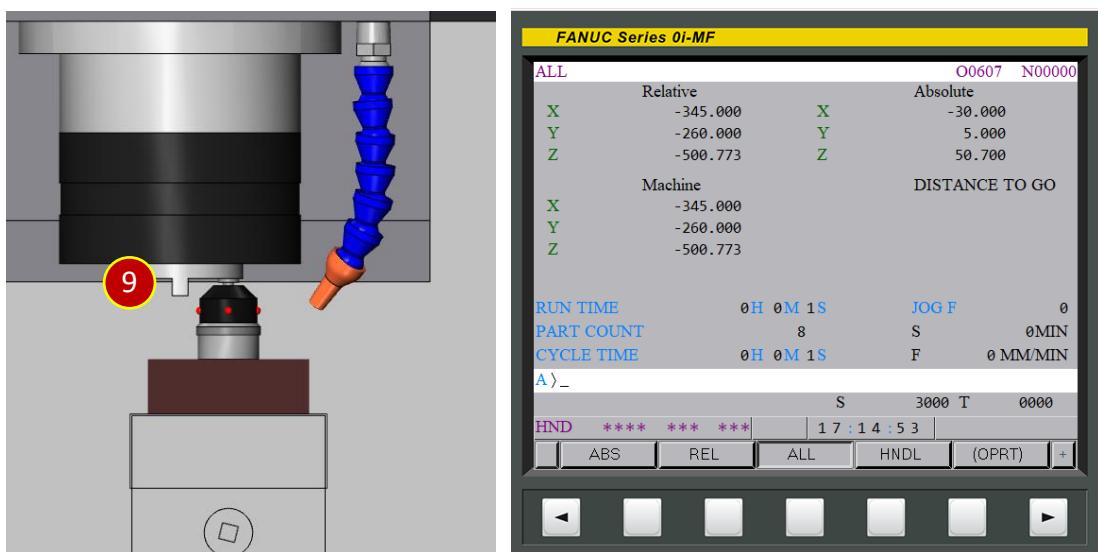
## (6) Switch MODE SELECT Knob to [( HANDLE )] Mode



(7) Press [ Z Axial Gauge ] button and place the Z Axial Gauge onto the Workpiece

(8) Use [( Handwheel )] to move Tool above the Z Axial Gauge.

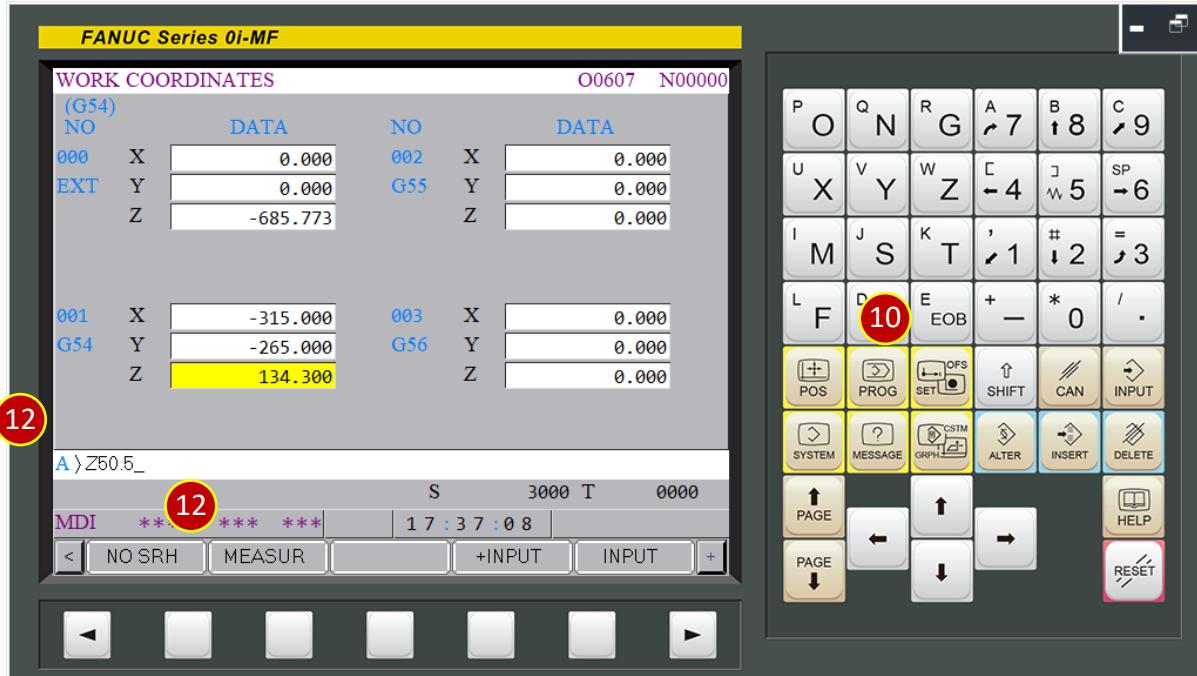
(9) Measure position according to ‘ Precise Fine-Tune to Touch Z Axial Gauge ’



# TM Fanuc Oi-MF Manual



- (10) Press [ OFFSET/SETTING ].
- (11) Select [ (OPRT) ] > [ Work ].
- (12) Move cursor to G54 Z column, type in “ Z50.5 ”. Press [ (OPRT) ] > [ MEASUR ].  
(Z Axial Gauge height = 50mm, reserved height = 0.5mm)
- (13) G54 Z Axis Work Coordinate setting complete when 134.500 appears in G54 Z Axis.



## 6.6 Tool Offset (Tool Gauge)

Based on the Spindle Nose, the distance between the Spindle Nose and the Tool Nose is Tool Geometry Offset value.

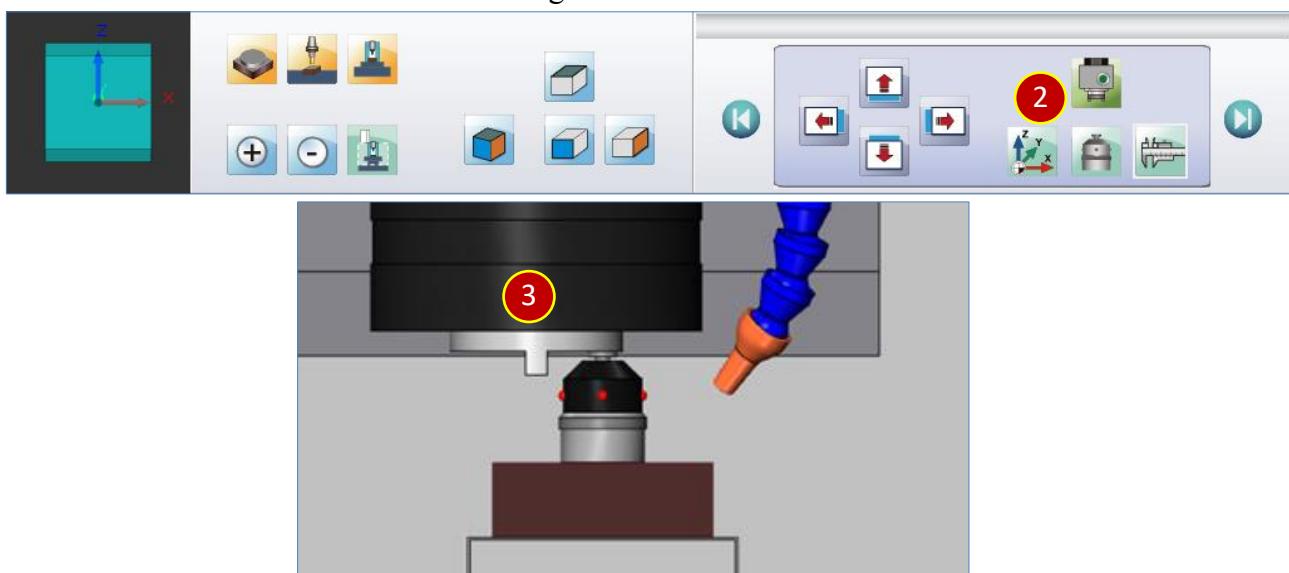
### 6.6.1 Tool Geometry Offset (Set Base Point) (Relative Zero Point)

- (1) Switch to Empty Tool T00.

E.g., In [(MDI)] Mode, input “ ; M06 T00 G49 ; ” to remove Tool.

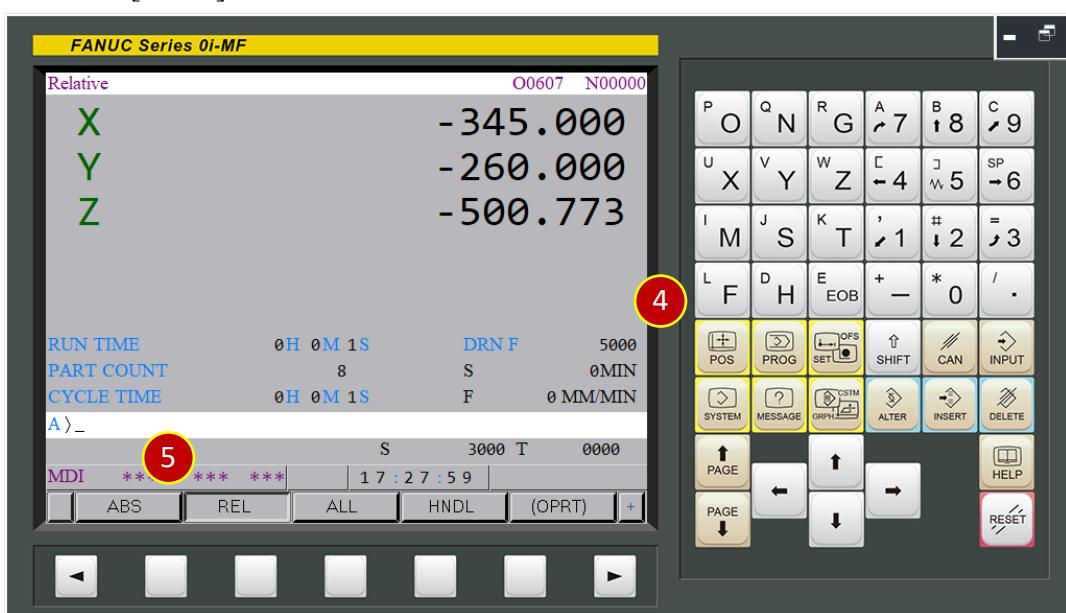
- (2) Press [ Z Axial Gauge ] and place the Z Axial Gauge onto the Workpiece.

- (3) Touch the Z Axial Gauge with Spindle Nose and operate according to ‘ Precise Fine-Tune to Touch Z Axial Gauge ’.



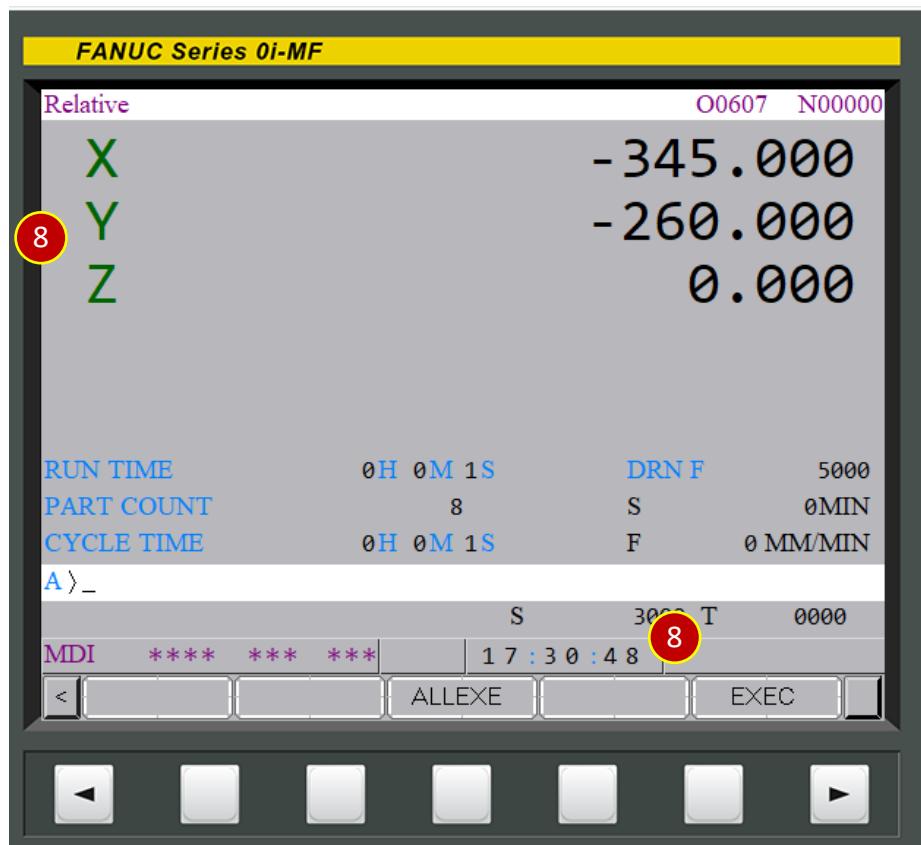
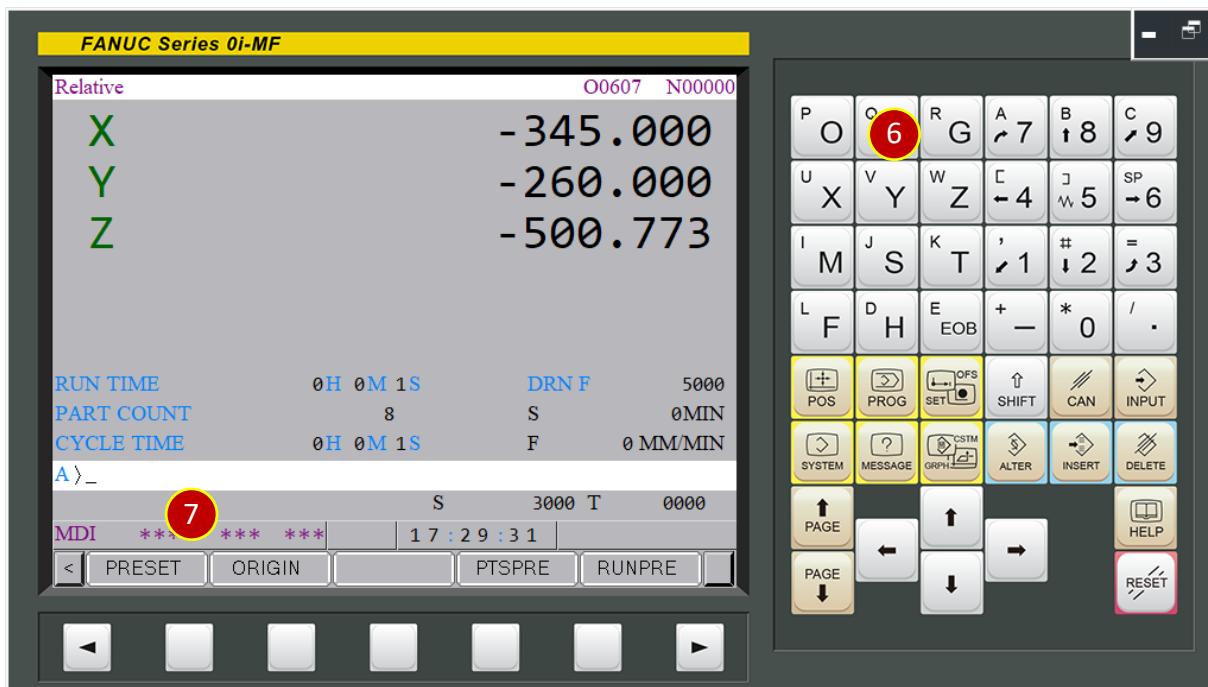
- (4) Press [ POS ].

- (5) Press [ REL ].



# TM Fanuc 0i-MF Manual

- (6) Press [ Z ], Z Axis status on Controller will start to flash
  - (7) Press [ (OPRT) ] > [ ORIGIN ]
  - (8) Press [ EXEC ], Z Axis coordinate becomes 0, setting current Z Axis position as base point (Relative Zero Point).

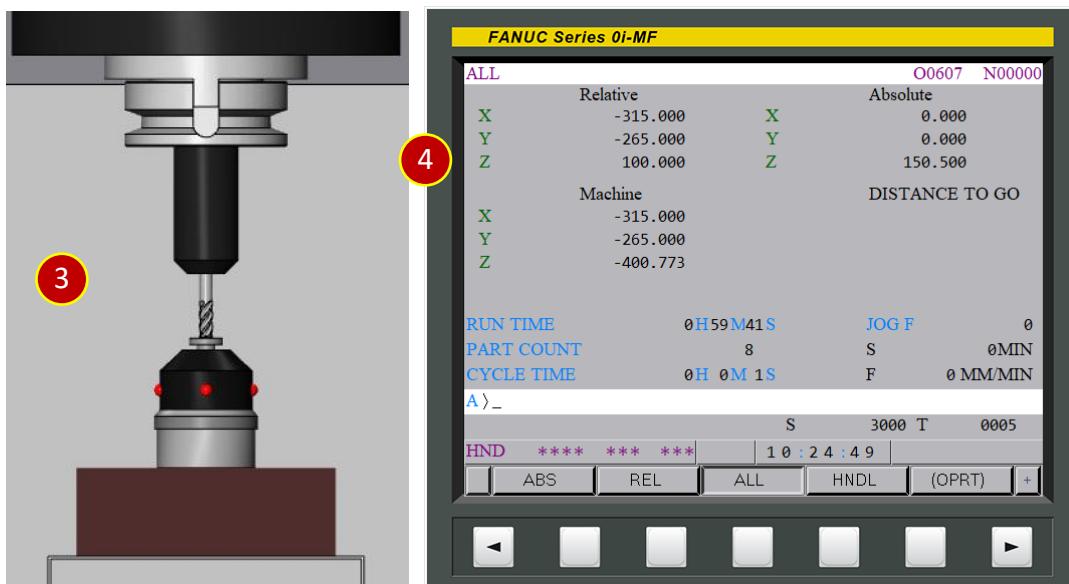


## 6.6.2 Tool Geometry Offset (Tool length)

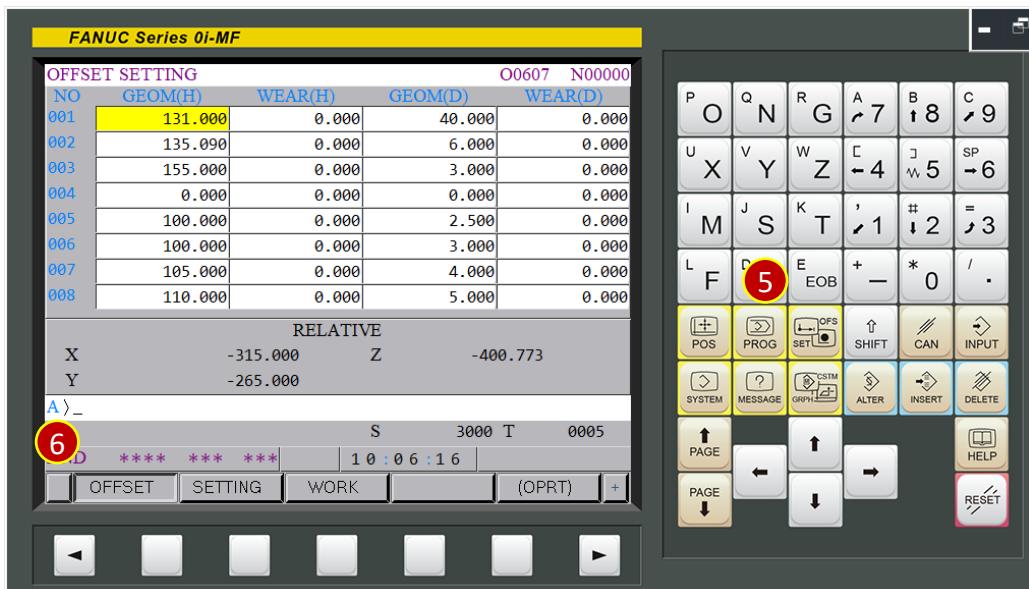
Use Z Axial Gauge to measure the length from the Spindle Nose to the Tool Nose.

Take Tool No.5 geometry offset for example:

- (1) Change Tool manually to Tool No.5
- (2) Press [ Z Axial Gauge ] to place the Z Axial Gauge onto the Workpiece
- (3) Touch the Z Axial Gauge with the Tool and operate according to ‘ Precise Fine-Tune to Touch Z Axial Gauge ’.
- (4) The current Z Axis value of Relative Coordinate will be the length of Tool No.5  
e.g., 100.000



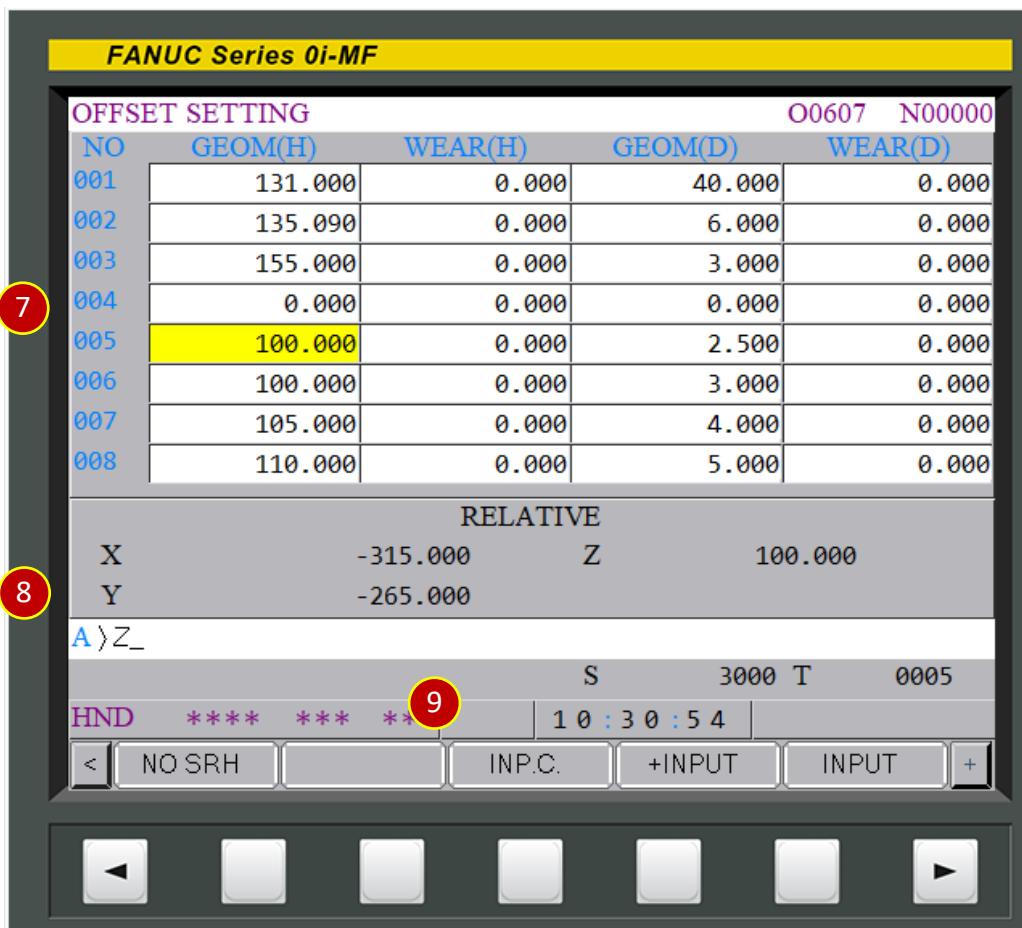
- (5) Press [ OFFSET/SETTING ]
- (6) Press [ OFFSET ]



# TM Fanuc Oi-MF Manual



- (7) Move cursor to No. 005 GEOM(H) column
- (8) Key in “ Z ”.
- (9) Press [ (OPRT) ] > [ INP.C ] to retrieve the value of Z Axis (127.001) and the No.32 Tool Length Offset is complete.



- (10) Finish all the Geometry Offset setting of all the needed Tool with the method above

10

OFFSET SETTING					O0607 N00000
NO	GEOM(H)	WEAR(H)	GEOM(D)	WEAR(D)	
001	131.000	0.000	40.000	0.000	
002	135.090	0.000	6.000	0.000	
003	155.000	0.000	3.000	0.000	
004	0.000	0.000	0.000	0.000	
005	100.000	0.000	2.500	0.000	
006	100.000	0.000	3.000	0.000	
007	105.000	0.000	4.000	0.000	
008	110.000	0.000	5.000	0.000	

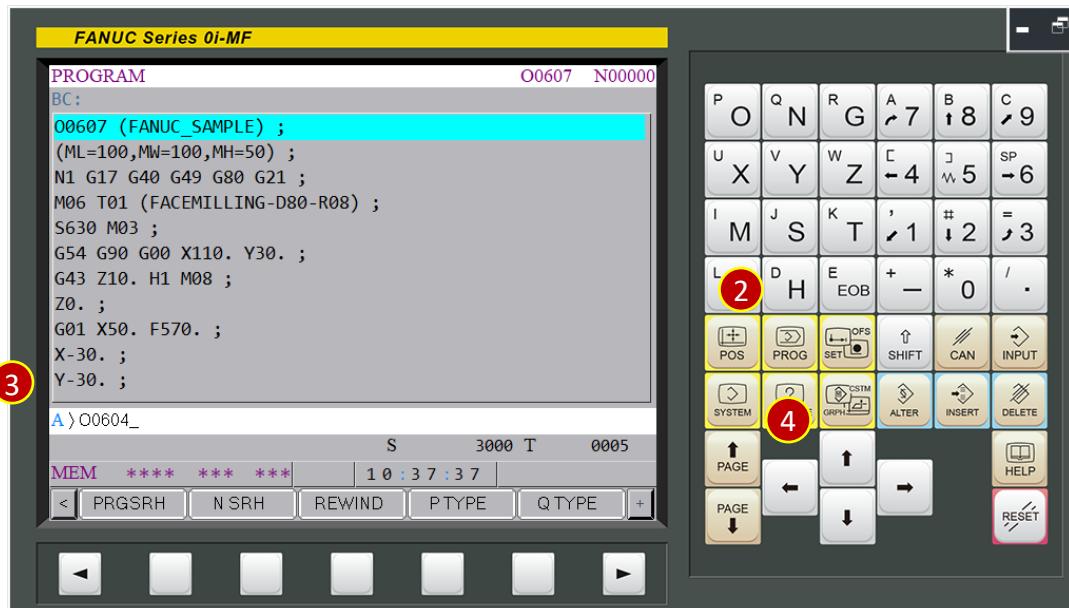
## 6.7 Auto Run

Auto-run the CNC program in the Controller

### 6.7.1 Operation Steps

- (1) Switch to [(AUTO)] Mode
- (2) Press [ PROG ] Button
- (3) Key in the program number to be simulated  
e.g., O0502

- (4) Press [ ] to call and open the program



(5) Before auto-running the program, switch RAPID OVERRIDE Knob to [( 25% )]

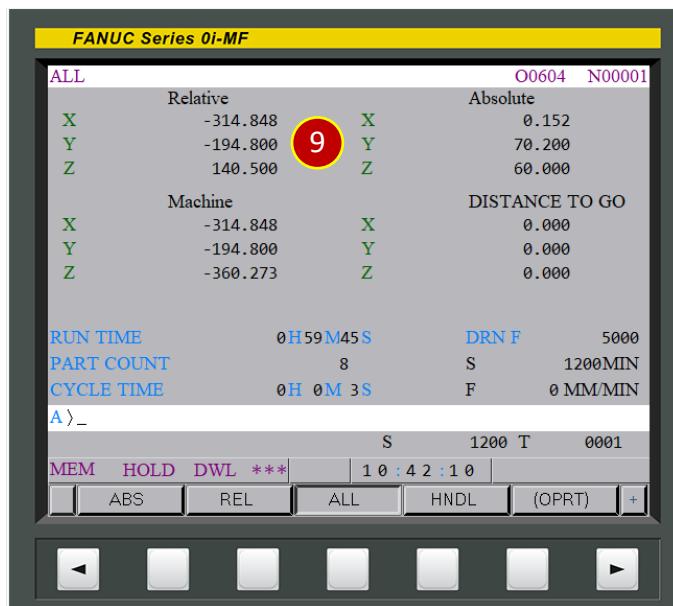
(6) Press [( S.B.K )] (lights up when enabled)

(7) Press [( CYCLE START )] to run the program

(8) Watch the Tool move. When the Tool becomes close to the Workpiece (about 60mm), press [( FEED HOLD )] immediately to pause the Tool cutting movement

(9) Check if the position of Tool, Workpiece and coordinates are identical to the dimensions

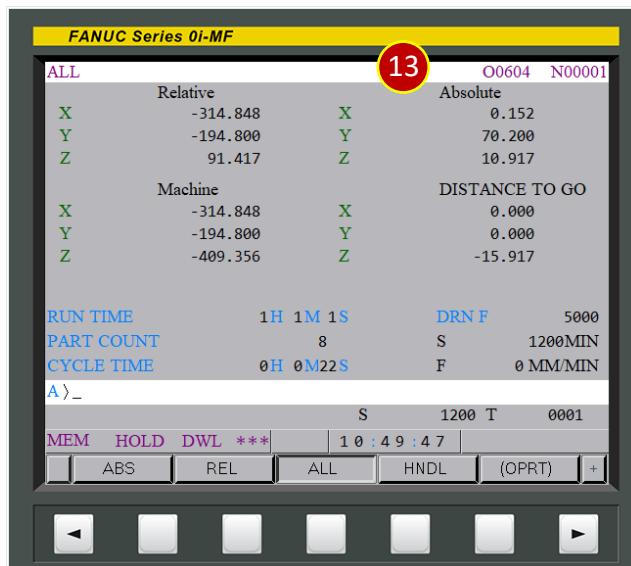
e.g., Absolute Coordinates Z =60.000. The Tool Nose is about 60mm away from Workpiece by visual inspection.



- (10) Press [( CYCLE START )] to continue running the program
- (11) When Tool becomes closer to the Workpiece (about 10mm), press [( FEED HOLD )]
- (12) Switch RAPID OVERRIDE Knob to [( F0 )]
- (13) Make sure the position of Tool and Workpiece are identical to program coordinates

## 6.7.2 Caution:

- (1) When commissioning machining, keep the Rapid Feedrate at F0 position if the Tool is close to the Workpiece
- (2) Always press [( FEED HOLD )] Button immediately to pause the machine whenever something wrong is sensed, then check the status.

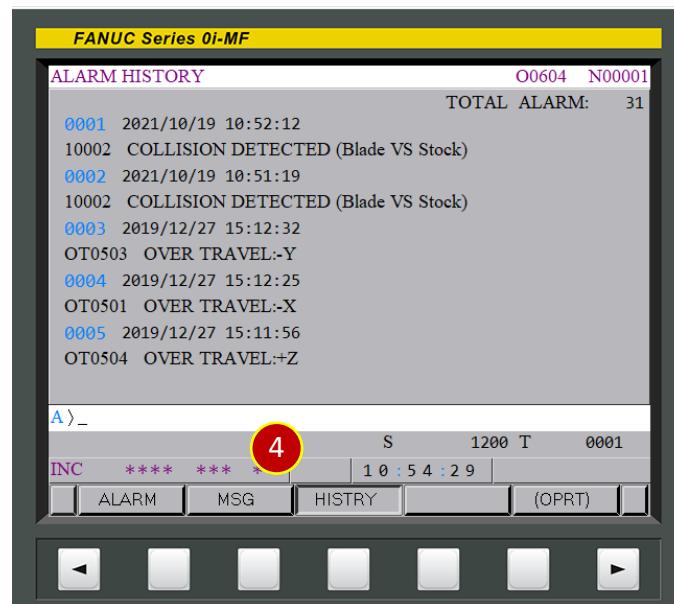


## 6.8 Machine Alarm

When errors occur during machine operation, ALARM notification will appear on the Controller.

### 6.8.1 Clear the Alarm

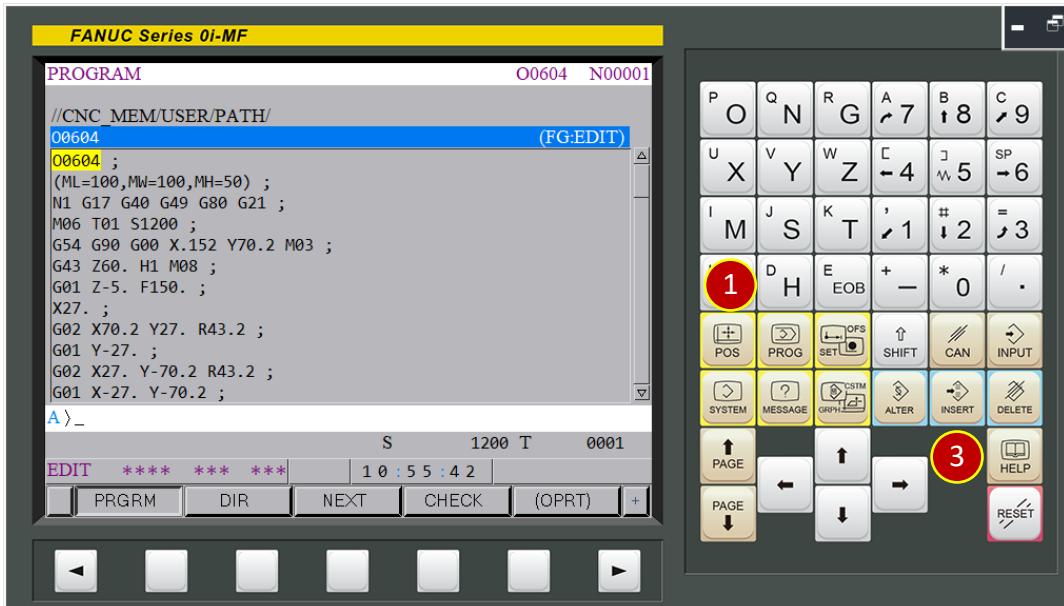
- (1) When the ‘ ALM ’ status on the Controller is flashing.
- (2) Check the alarm number and content and deal according to it.
- (3) Press [ RESET ] to clear the alarm after acknowledging the alarm detail.
- (4) Press [ HISTRY ] Button to check alarm history



## 6.8.2 Return to Auto Run Mode

To return to Auto Run Mode when the program is interrupted or when the alarm notification shows up:

- (1) Press [ PROG ] to return to the Program Display Window.
- (2) Switch to [( EDIT )] Mode.
- (3) Press [ RESET ] to set the cursor back to the beginning of the program.



- (4) Switch to [( AUTO )] Mode
- (5) Make sure the cursor is at the beginning of the program again, and press [( CYCLE START )] to auto-run



## 7. Appendix 1, System Parameters

### 7.1 System Parameters

Press [ SYSTEM ], switch to System Parameters Window.



### 7.2 I/O input and output device parameters

#### 7.2.1 #0000 Bit [ 0 ] TVC: Perform TV check

= 0: No (default)

= 1: Conduct

(With the TV CHECK of OFF/SET [ SETTING ])

#### 7.2.2 #0000 Bit [ 1 ] ISO: Code format for data output

= 0: EIA code

= 1: ISO code (Default)

(With the PUNCH CODE of OFF/SET [ SETTING ])





## 7.2.3 #0020 I/O CHANNEL: Specify input/output device number

= 0: Channel 1 (Default)

= 1: Channel 1

= 2: Channel 2

(With the IO CHANNEL of OFF/SET [ SETTING ])

00020	I/O CHANNEL
<input type="text" value="0"/>	

## 7.2.4 #0101/#0111 Bit [ 0 ] SB2 stop bit setting

= 0: 1 bit (Default)

= 1: 2 bits

## 7.2.5 #0101#/0111 Bit [ 3 ] ASI data input code

= 0: EIA or ISO code (Automatic recognition) (default)

= 1: ASCII code

00101	NFD	ASI	HAD	SB2
<input type="text" value="0"/>				
				<input type="text" value="1"/>

## 7.2.6 #0103/#0113: Baud Rate (Default 10)

Set Value	Baud Rate
1	50
2	100
3	110
4	150
5	200
6	300

Set Value	Baud Rate
7	600
8	1200
9	2400
10	4800
11	9600
12	19200

00103	BAUD RATE
<input type="text" value="10"/>	



## 7.3 About programming parameters

### 7.3.1 #3401 Bit [ 0 ] DPI: Address words that can use a decimal point when the decimal point is omitted

- = 0: Regard as the minimum setting unit (0.001) (Default)
- = 1: Treated as mm, inch, sec unit

03401	ABS	MAB	DPI
[0]	[0]	[0]	[0]
[0]	[0]	[0]	[0]

### 7.3.2 #5101 Bit [ 4 ] RD1 Bit [ 5 ] RD2: The direction of the Tool retracts axis of the canned cycle G76 or G87

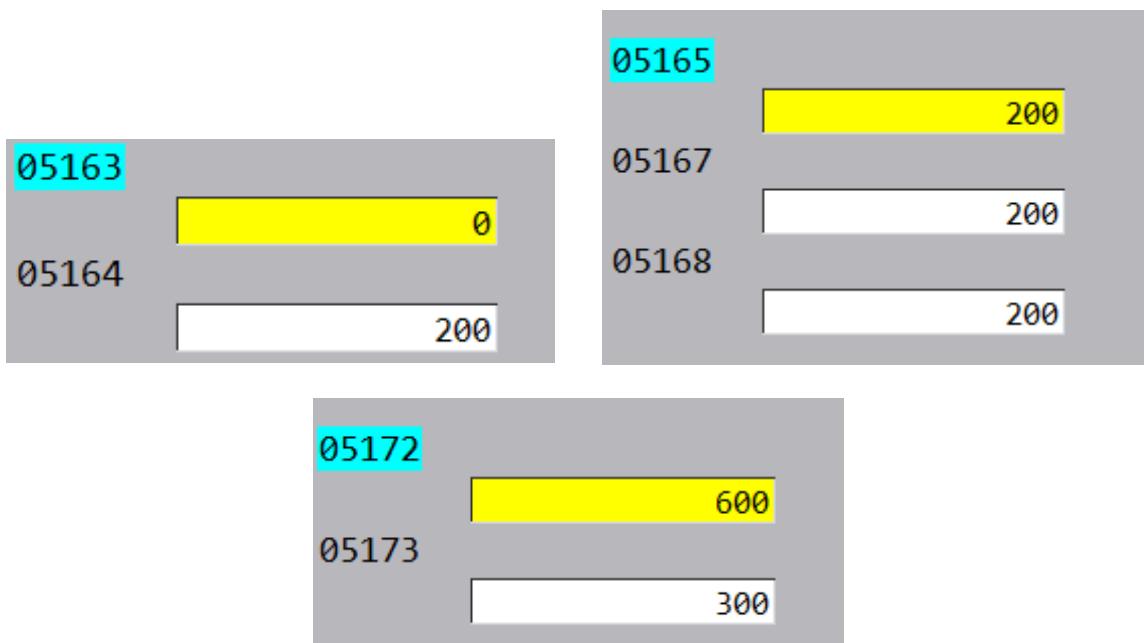
The settings are as follows according to the plane selection:

RD2	RD1	G17	G18	G19
0	0	+X	+Z	+Y
0	1	-X	-Z	-Y
1	0	+Y	+X	+Z
1	1	-Y	-X	-Z

05101	M5B	M5T	RD2	RD1	EXC	FXY
[0]	[0]	[0]	[0]	[0]	[0]	[0]

## 7.3.3 G83 uses small hole chip removal drilling cycle setting

- (1) #5163: M code setting, Default 0
- (2) #5167, #5168, #5164, #5165: Retraction speed ratio, Default 200%
- (3) #5172: When there is no I, return to the R point movement speed, the default value is 600 mm/min
- (4) #5173: Move to the bottom of the hole when there is no I, the default value is 300 mm/min

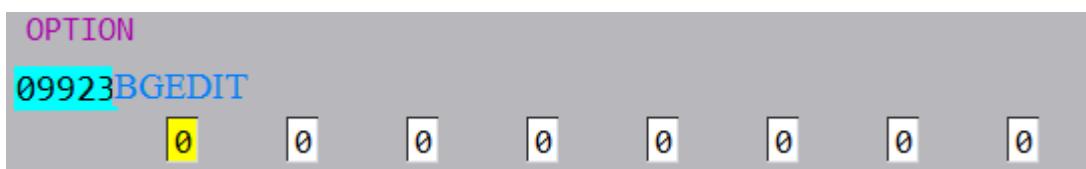


## 7.4 Background editing function setting

### 7.4.1 #9923 Bit [ 7 ] BGEDIT: Background editing function

=0: Close (VM Default)

=1: Open (TM Default)



CNC Simulator  
CAD/CAM System  
Training Equipment



**RenAn Information Technology Co., Ltd.**  
5F-3, No.1 Keji Road, Dali District, Taichung City 412028, Taiwan

+886(4)24918777  
[service@renan.com.tw](mailto:service@renan.com.tw)  
[www.renan.com.tw](http://www.renan.com.tw)



RenAn Website