

# CNC Operation Training Simulator

Horizontal Turning

Fanuc 0i-TF

Operation Manual



**RenAn**



# ***Ren An Information Technology Co., Ltd***

## **Product Guarantee**

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

Guarantee items instruction:

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

Software product: As from purchase date, the customer can change version within 90 days.

Hardware product: As from purchase date, free maintenance within a year.

***During the free maintenance period,  
the company will ask for cost or the service fee when:***

The company is unable to show product guarantee.

The content in the product guarantee doesn't match with the product, or the content has been modified, or cannot be identified.

After purchasing, the hardware has been damaged or has broken down during transportation.

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, equipments comes back to our company or the reseller.

***During the free maintenance period,*** please contact our resellers for machine repair.

When transporting the machines, please use the original Ren An package, otherwise, Ren An won't be responsible for the damage during transportation.

Any questions regarding the transportation fee, please contact Ren An or our resellers.

***Beyond the free maintenance period:***

the repair fee will differ according to different conditions.

- Limited flaw guarantee:**
- (A)Ren An assures all the functions mentioned in the instruction manual within 90 days.
  - (B)During one year after delivery, under normal using condition, Ren An will be responsible for any flaw.

**Customer compensation:**

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

Ren An won't be responsible for any lost files, stolen parts, and damage caused by purpose.

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

***The contract is under the Law of Taiwan, Republic Of China***

# ***Ren An Information Technology Co., Ltd***

## **Software License Agreement**

THE FOLLOWING ARE THE TERMS AND CONDITIONS UNDER WHICH REN AN 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.

【You, as the Customer, agree as follows:】

### ***1. Definitions***

"Designated Computer" shall mean the one computer upon which Customer shall run the Licensed Software. "Licensed Software" shall mean the software, in executable form only, supplied by Ren An to this Agreement.

### ***2. License***

- You may not distribute copies of the licensed Software or its accompanying written materials to others.
- You may not solely copy disks, unless under agreement.  
No other copies shall be made, sold, rent, loaned, published or exchanged unless authorized by Ren An.
- You may not modify disk's codes, reverse-engineer the software, or decompile codes.
- Ren An 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.

### ***3. General***

- This License Agreement is a legal agreement between you and Ren An Information Technology Co., Ltd for the software Product(s) you have purchased from Ren An.
- The license has been granted the authority to sub-license the Software to you the Customer.
- 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 Ren An, and supercedes any prior discussions or agreements relating to the software.

This license may be modified only by the written agreement of both parties.

I AGREE: \_\_\_\_\_

-

## ***Proprietary Notice***

This document contains propriety information of Ren An 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 Ren An licensed software.

### **REN AN Information Technology Co., Ltd Trademark Proclamation.**

### ***REN AN 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 Ren An. 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 maybe 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.

REN AN is a registered trademark.

All other trademark belong to their respective owners.

# Content

1. Basic Introduction .....	1-1
1.1 Start CNC Training Machine .....	1-1
1.2 Introduction .....	1-2
1.3 View and Application Tool Bar .....	1-3
1.4 View Tool Bar .....	1-5
2. Tool and Material .....	2-1
2.1 Material Setting .....	2-1
2.2 Tool Manager .....	2-2
2.3 Turret Install .....	2-5
3. Jaw Installation and Operation .....	3-1
3.1 Uninstall Jaw .....	3-1
3.2 Material Setting .....	3-4
3.3 Jaw Install .....	3-5
3.4 Soft Jaw Modify .....	3-12
3.5 Workpiece Clamping .....	3-25
4. Controller Panel .....	4-1
4.1 Controller Panel Introduction .....	4-1
4.2 Machine Operation Mode Select Buttons .....	4-3
4.3 Select Tool Manually .....	4-4
4.4 Handwheel Feed Operation .....	4-5
4.5 Common Buttons Description .....	4-7
5. Controller Panel Operation .....	5-1
5.1 Controller Panel Introduction .....	5-1
5.2 Program Edit .....	5-3
6. Transmission Parameter Setting and Operation .....	6-1
6.1 RS232 COM Setting .....	6-1
6.2 Transmission Parameter Setting .....	6-3
6.3 Receive Program from the Controller .....	6-6
6.4 Send Program from the Controller .....	6-8
6.5 CNC File Import/Export Management .....	6-10
7. Machine Basic Operation .....	7-1
7.1 ZRN Operation .....	7-1
7.2 Work Coordinates Setting (OFFSET) .....	7-3
7.3 Z Axis Tool Offset (Offset) .....	7-20
7.4 Auto-Run .....	7-25
7.5 Machine Alarm .....	7-28

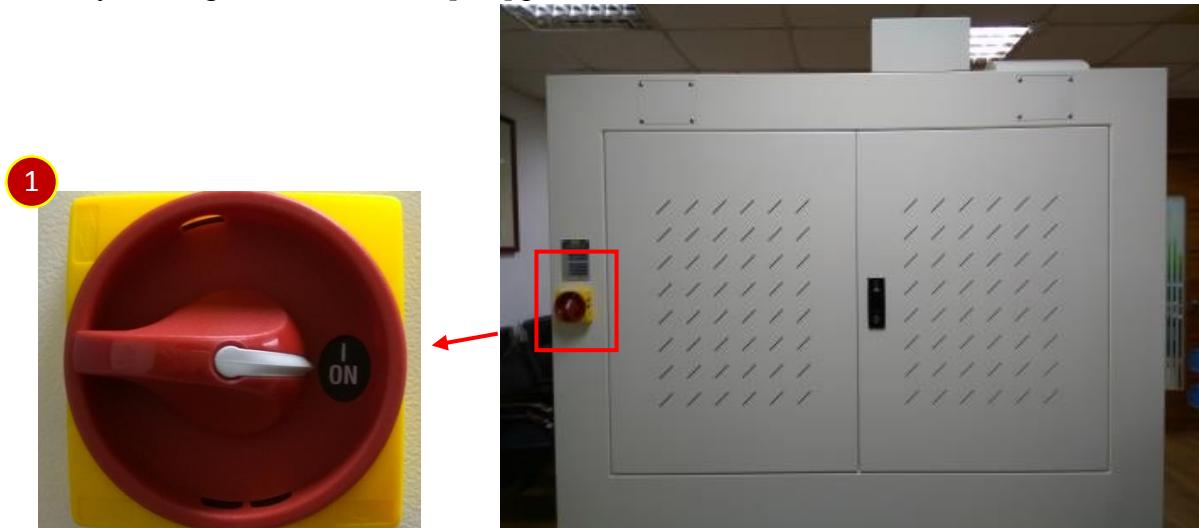
# 1. Basic Introduction

## 1.1 Start CNC Training Machine

### 1.1.1 How to Activate the System

(1) Power On the Training Machine 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 Training Machine



## 1.2 Introduction

The workspace of the training system includes:

Machine Simulation Area, Controller and Tool Bar, Operation Panel



### 1.2.1 Workspace Description

Workspace Components	Description
Machine Simulation	Includes 3D View Display, View Function Buttons and Application Function Buttons
Controller Function Panel	Controller Panel and Touch Screen Tool Bar
Machine Operation Panel	Display Machine Operation Panel

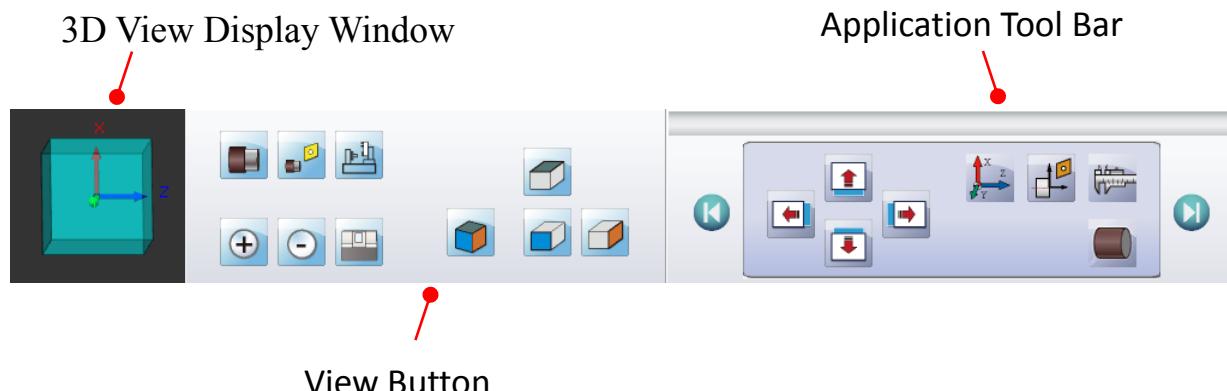


## Manual

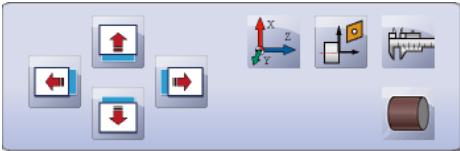
## 1.3 View and Application Tool Bar

## 1.3.1 View and Application Tool Bar includes:

- (1) 3D View Display Window: To display the current XYZ angels,  
touch and drag to switch the angle
- (2) View Button: Switch to different common angle
- (3) Application Tool Bar: Sliding touch interface.  
Switch to different function button group  
 : Switch to previous categories of function  
 : Switch to next categories of function



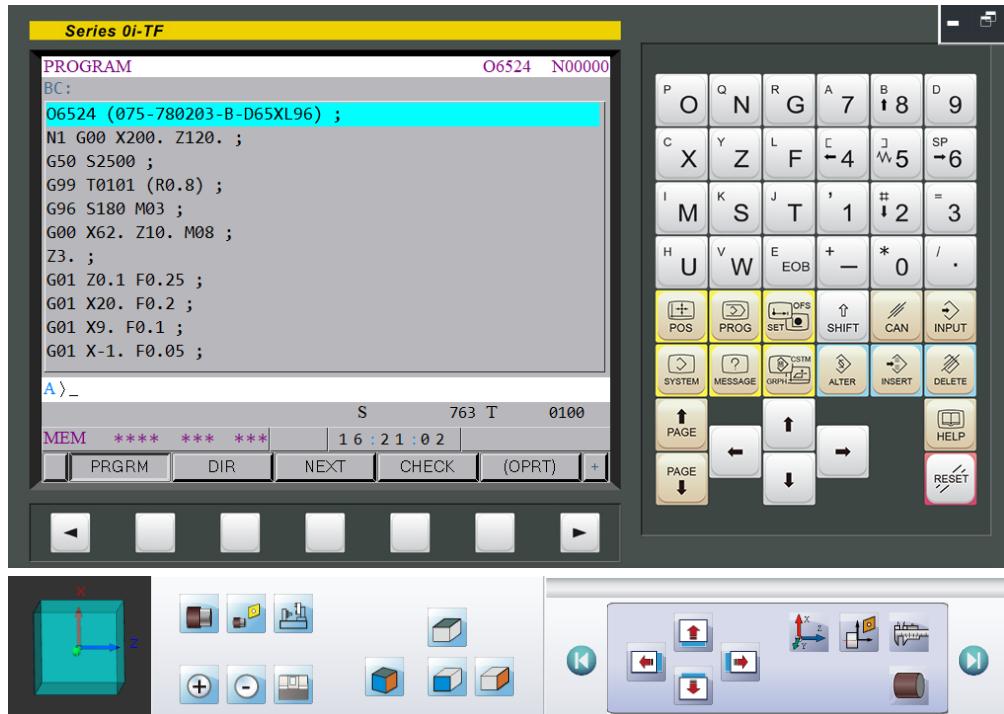
## 1.3.2 Application Function Button Description

Application Function Button	Description
	View Shift, Workpiece Dimensional Measurement, Workpiece Section
	Material, Tool Turret, CNC File Import/ Export, Tool Manager, Turret Data
	Simulation Speed Control
	Operation Configuration Import/ Export Resume to default Product and Material Import/ Export
	Software/ Hardware Setting
	Examine System

## Manual

## 1.4 View Tool Bar

【View Tool Bar】 can adjust the viewing angle and zoom in/ out.



## 1.4.1 Buttons Description

Button	Name	Description
	Machine View	Zoom in/out with machine size
	Table View	Adjust to the specific view of material and tool
	Material View	Zoom in/out with materialize
	Hide Case	Display or hide the shell
	Zoom out	Zoom out the view
	Zoom in	Zoom in the view

Button	Name	Description
	Top view	Switch angle to top view (XY)
	Front view	Switch angle to front view (XZ)
	Side view	Switch angle to side view (YZ)
	ISO view	Switch angle to ISO view (XYZ)
	UP	Hold the button to continuously move the picture upward
	DOWN	Hold the button to continuously move the picture downward
	LEFT	Hold the button to continuously move the picture leftward
	RIGHT	Hold the button to continuously move the picture rightward
	Tool Setting Display State	Activate the Tool Setting Display State to enhance the hint effect (In Manual mode only)
	Measure	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) Press [Application Tool Bar] > 【Material Setting】  
proceed with the setting of the material dimension
- (2) Enter the window of [Material Setting], input the diameter and length of the material  
E.g.: Diameter (D)=65.0, Length (L)=96.0, (L1)=30.0, (D2)=0.0, (L2)=0.0
- (3) After the setting is completed, press 【OK】



#### 2.1.2 Material Settings Buttons Description

Button	Description
	Numeric keys from 0~9 and decimal point
	Backspace
	Clear the buffer
	Confirm the settings of the material
	Exit from the material setting window

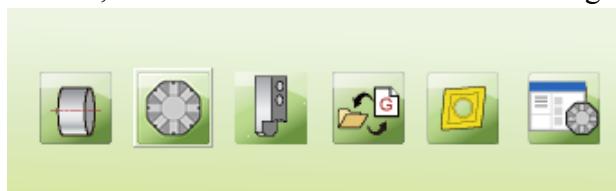
## Manual

### 2.2 Tool Manager

Via 【Application Tool Bar】 , press 【Tool Manager】 ,

To set up the tool data includes Tool tag, Holder parameter,

Tool parameter, Process data, Cut condition as the basic data setting before simulation



#### 2.2.1 [Tool Manager] Function Description

**Function Buttons**

**Tool Data**

**Tool List**

Workpiece	Cutting Range	Max RPM S	Surface (m/min)	Spindle (rpm)	Feed rate F	Depth per cut
S45C	0 - 300	2500	180	1000	0.3	2.4

**Machining Data**

**Workpiece & Material**

- Tool grade = CA225
- Work material = S45C
- Workpiece cutting range :
  - Min. dia. = 0
  - Max. dia. = 300
  - Offset # = Auto
  - Offset = Auto
- Spindle = Forward
- Coolant = On
- Air = Off

**Feed & Speed**

- Max RPM S = 2500
- Surface (m/min) S = 180
- Spindle (rpm) S = 1000
- Feed rate F = 0.3
- Depth per cut = 2.4

**Tool Life Wear**

- Life = 3H M S
- Wear = 0.001 / 400 mm

**Cut view**

- Path color =
- Cut color =
- Chip type =
- Chip color =

**Recommended range**

- Surface (m/min) S = 100 - 250
- Spindle (rpm) S = 800 - 1200
- Feed rate F = 0.1 - 0.45
- Depth per cut = 1 - 5

## Manual

### 2.2.2 New Tool

e.g. Add tool “ExternalRough(R)-A80-R0.8”

(1) Select tool type via the tool list

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

(2) Press similar tool number

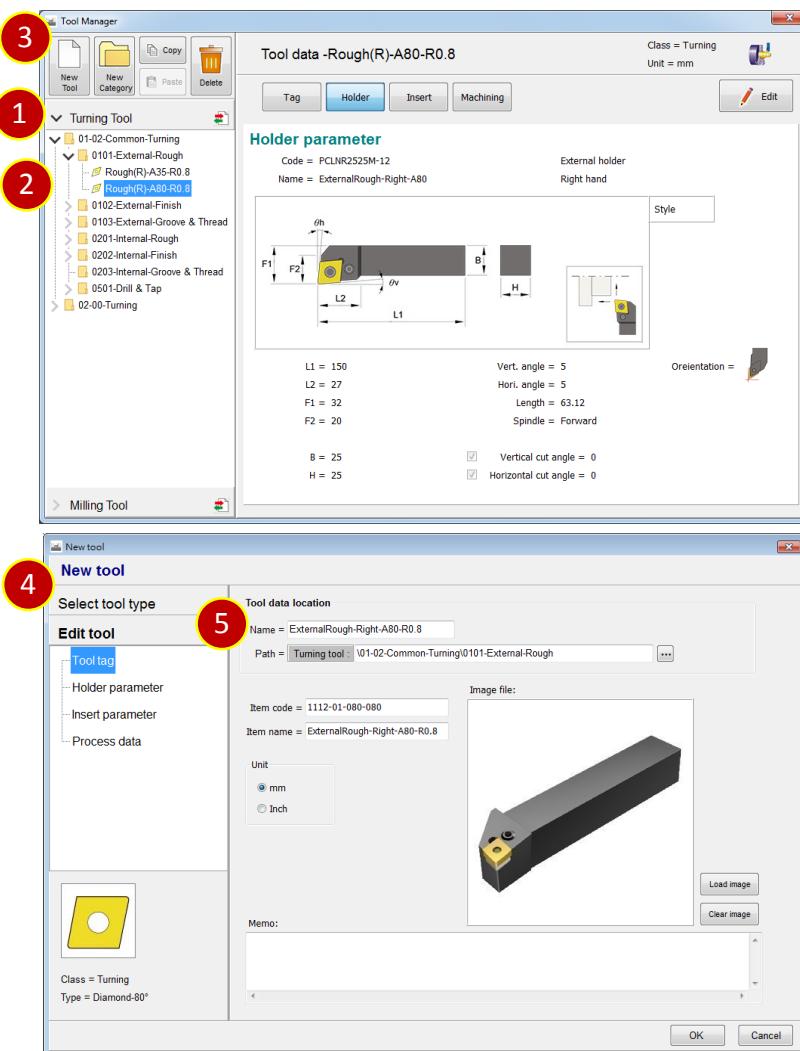
e.g. Rough(R)-A80-R0.8

(3) Press 【New Tool】 to enter New tool window

(4) Press 【Select Tool Type】 > 【Turning】 > 【Diamond-80】 to enter Edit tool page

(5) Key in Tool Location Name

e.g. ExternalRough(R)-A80-R0.8



## Manual

(6) Press 【Holder Parameter】 , to set tool parameter

e.g. L1=150, L2=27, F1=32, F2=20,B=25, H=25, Vertical Angle=5

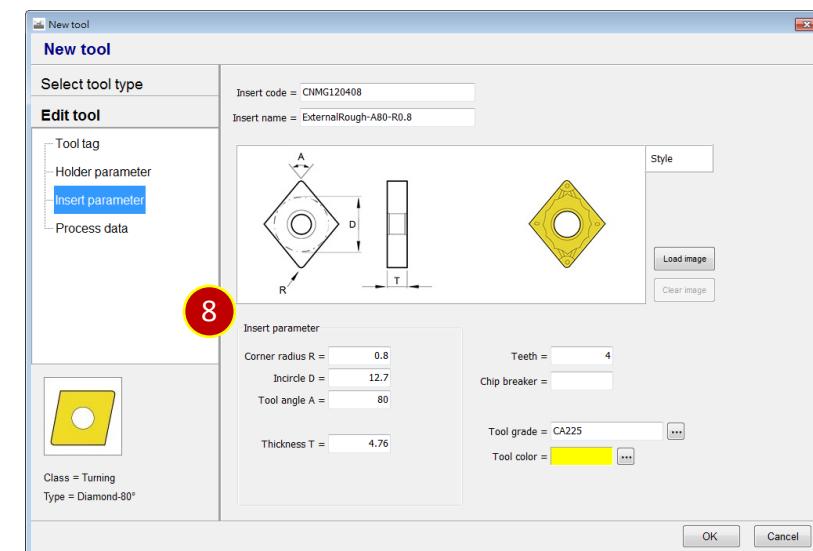
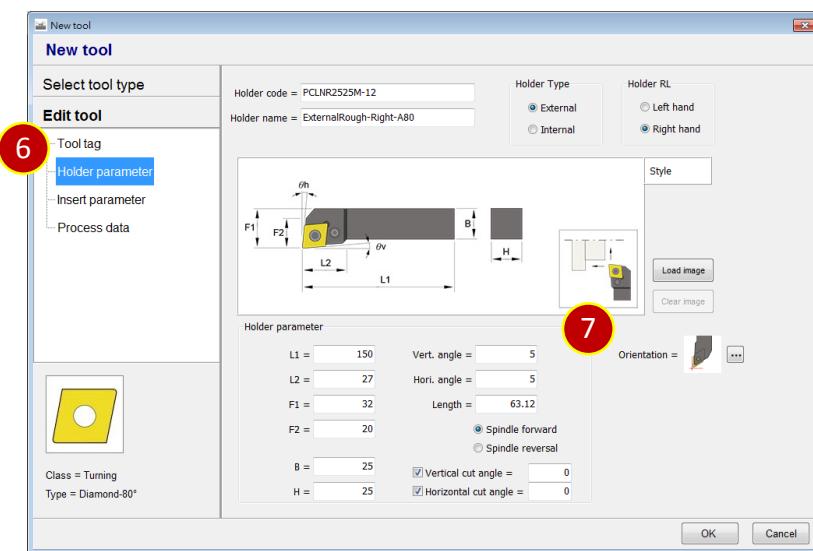
Horizontal Angle=5, Length= 63.12, Vertical cut angle=0,

Horizontal cut angle=0

(7) Set up the Orientation

(8) Press 【Insert Parameter】 Key in the parameter.

e.g. Corner Radius R=0.8, Incircle D =12.7, Tool angle A=80, Thickness T=4.76

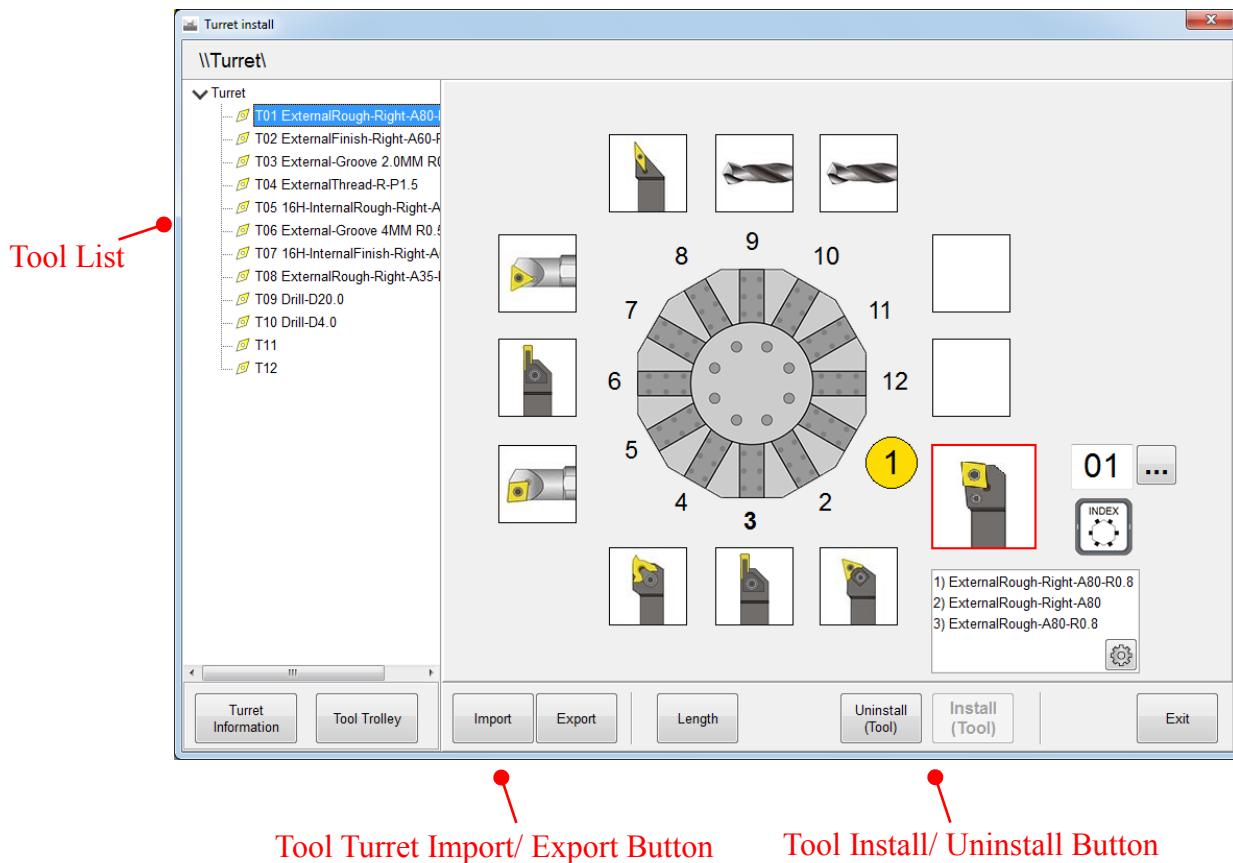


## Manual

### 2.3 Turret Install

To install the tool turret on the machine before simulation

#### 2.3.1 Tool Turret Install Function Description



## Manual

## 2.3.2 Tool Install

e.g. Install “Rough(R)-A80-R0.8” on the No.11



(1) Press 【Application Tool bar】 > 【Turret】

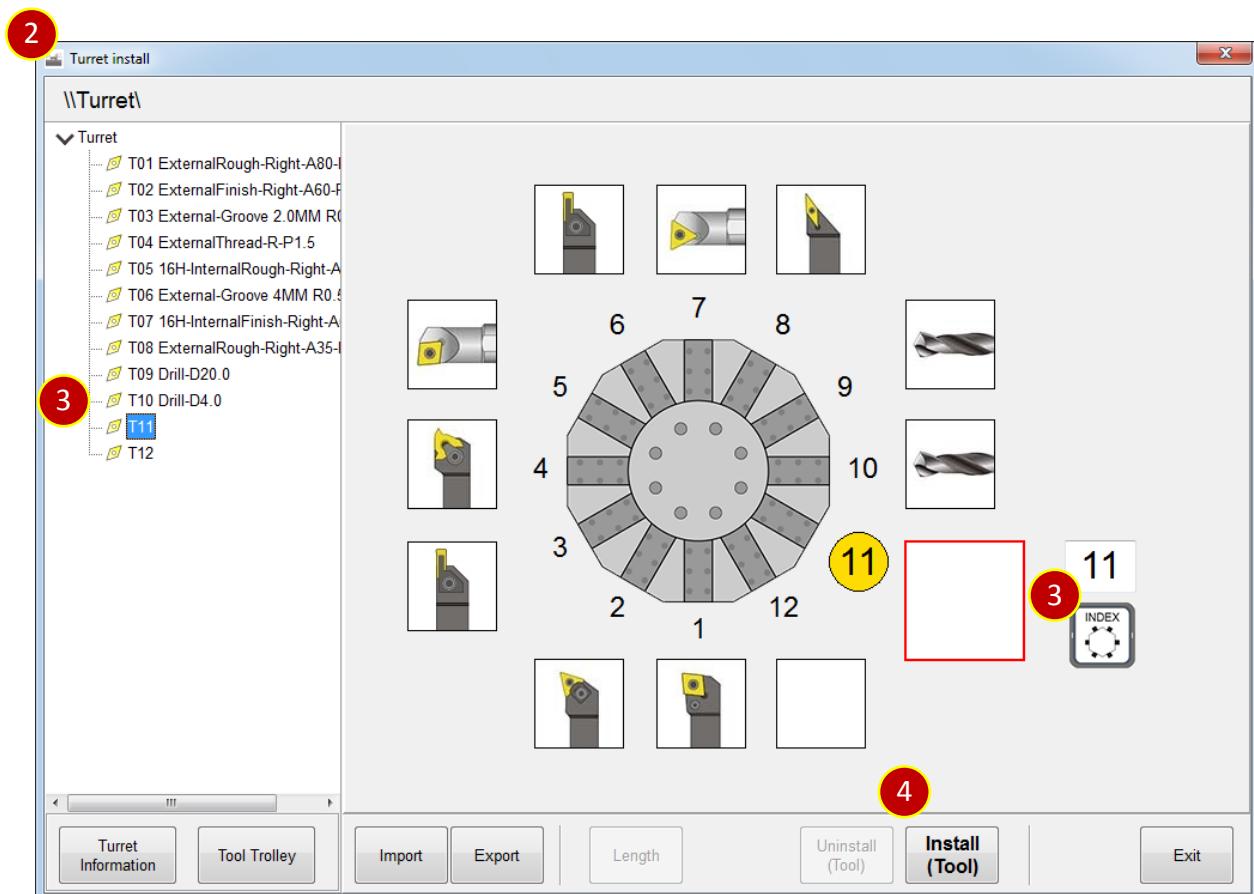
(2) Open Turret Install window



(3) Select the Empty tool No.11, and Press [T11] in the Tool List,

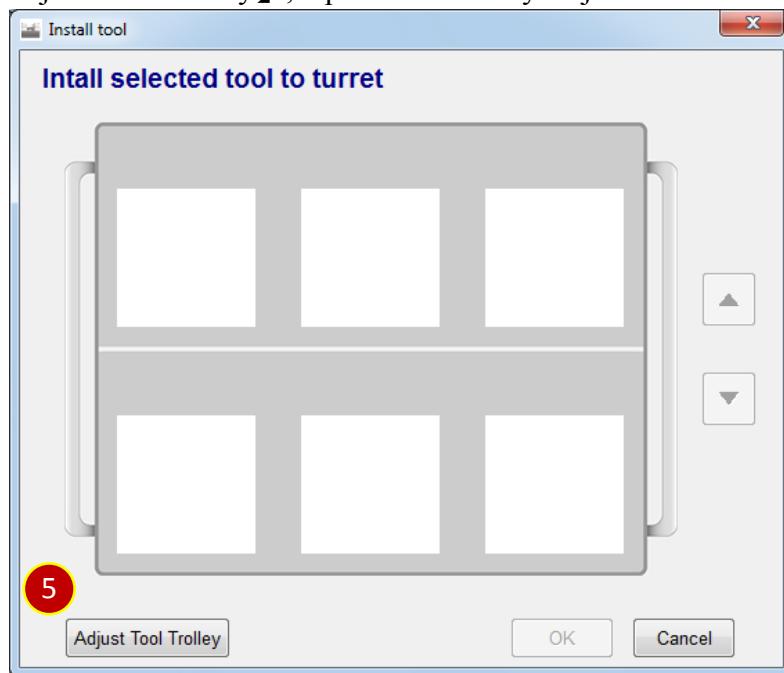
then Rotate to preparatory tool position

(4) Press 【Install (Tool)】 , open the Tool Install window

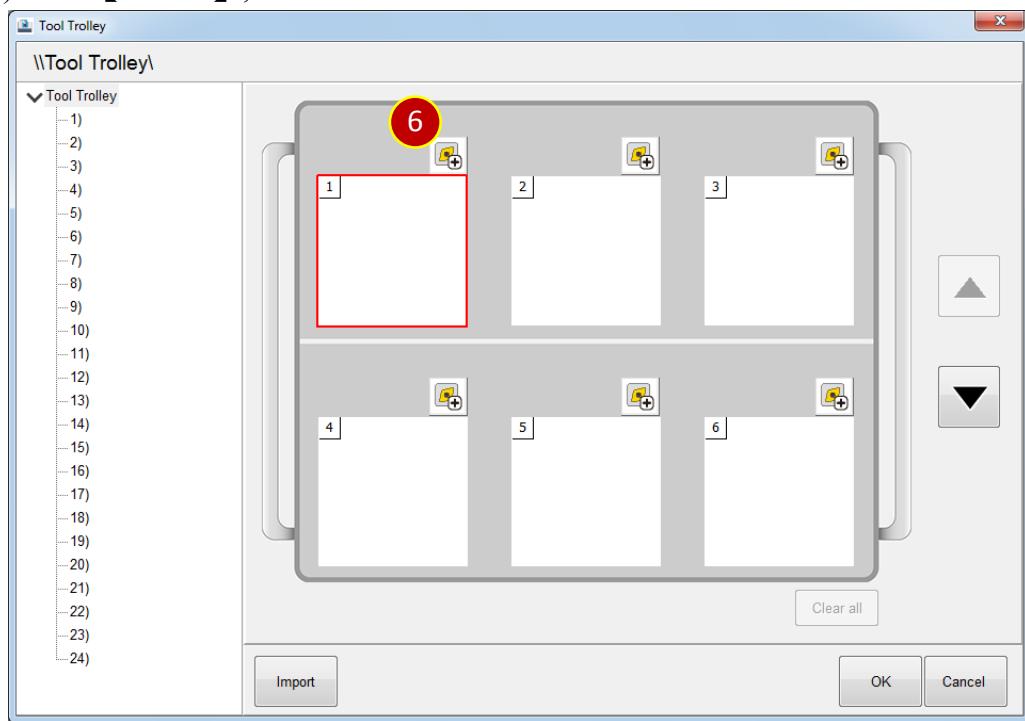


## Manual

(5) Press 【Adjust Tool Trolley】 , Open Tool Trolley Adjust window



(6) Press 【】 , enter Select Tool window

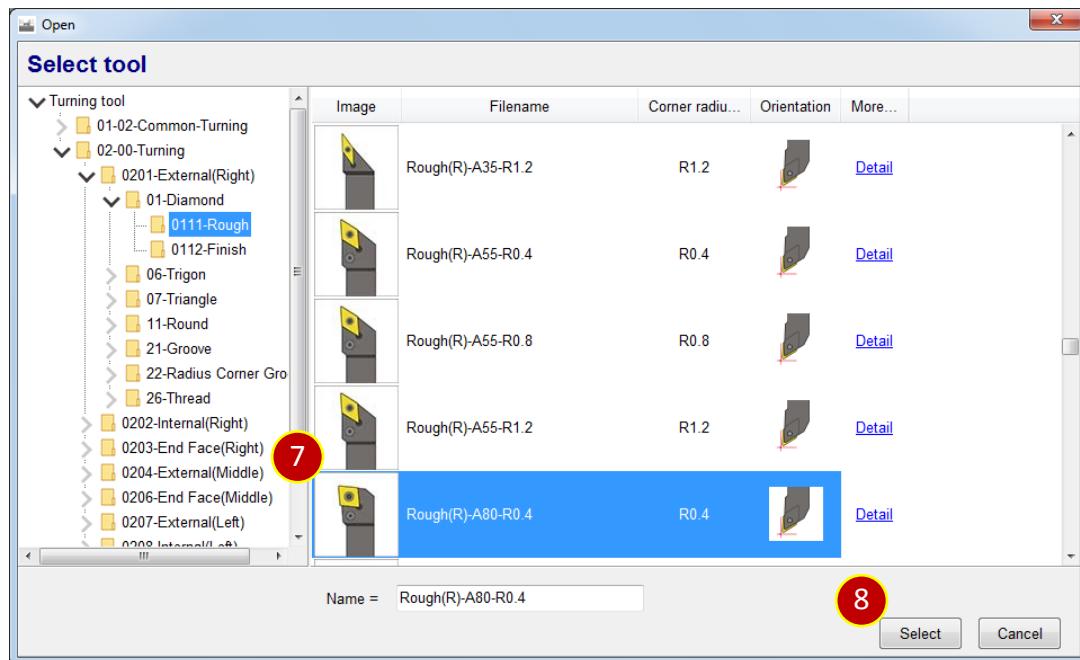


## Manual

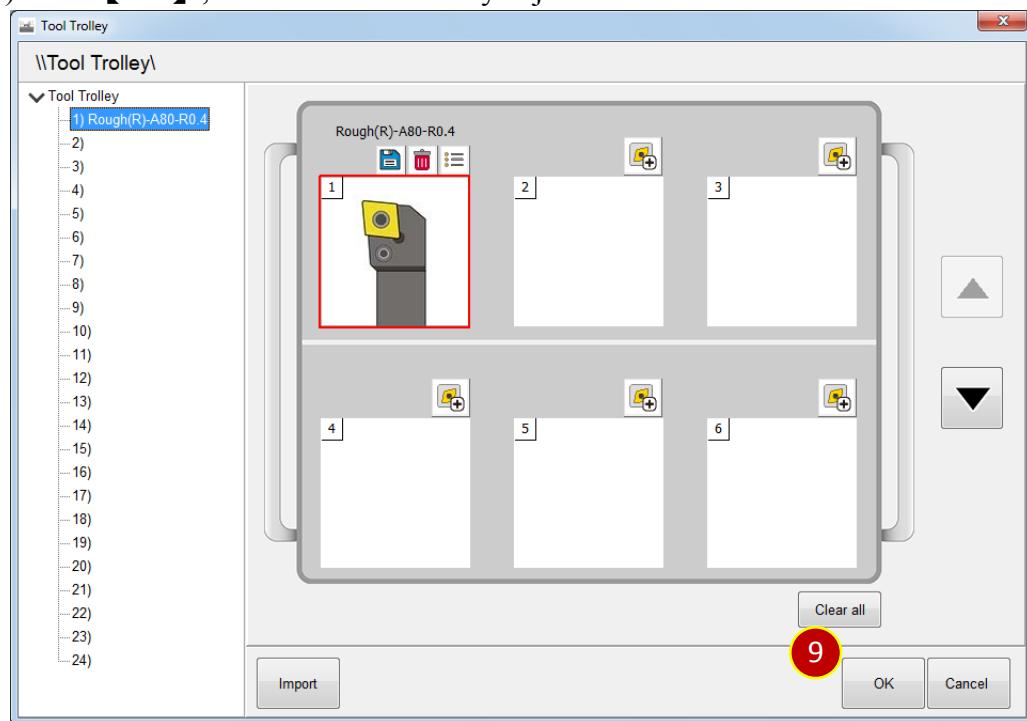
(7) Select the tool in Tools data

e.g. [01-02-Turning] &gt; [Rough(R)-A80-R0.8]

(8) Press 【Select】 , bring tool back to Tool trolley

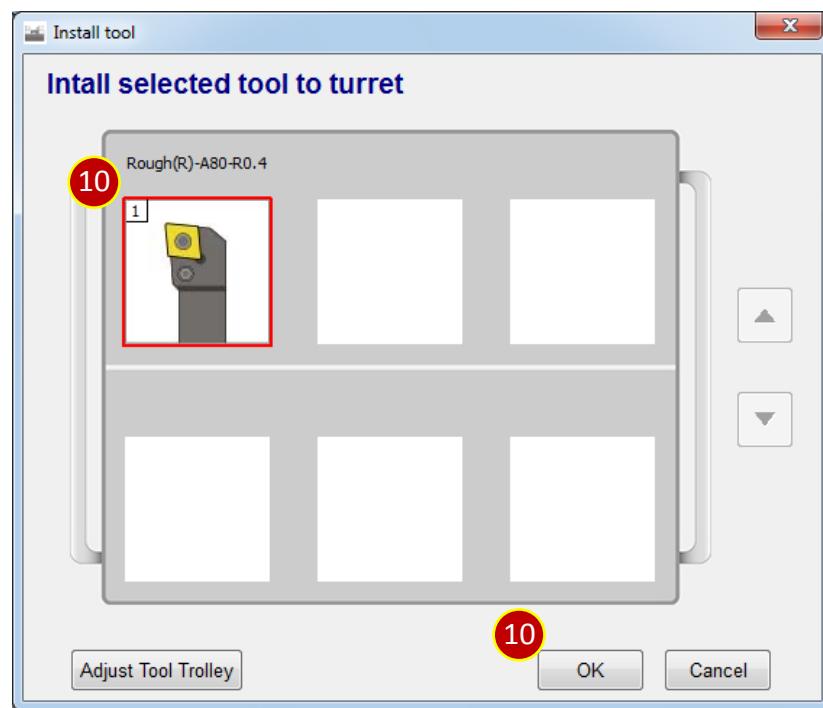


(9) Press 【OK】 , close the Tool Trolley adjust window

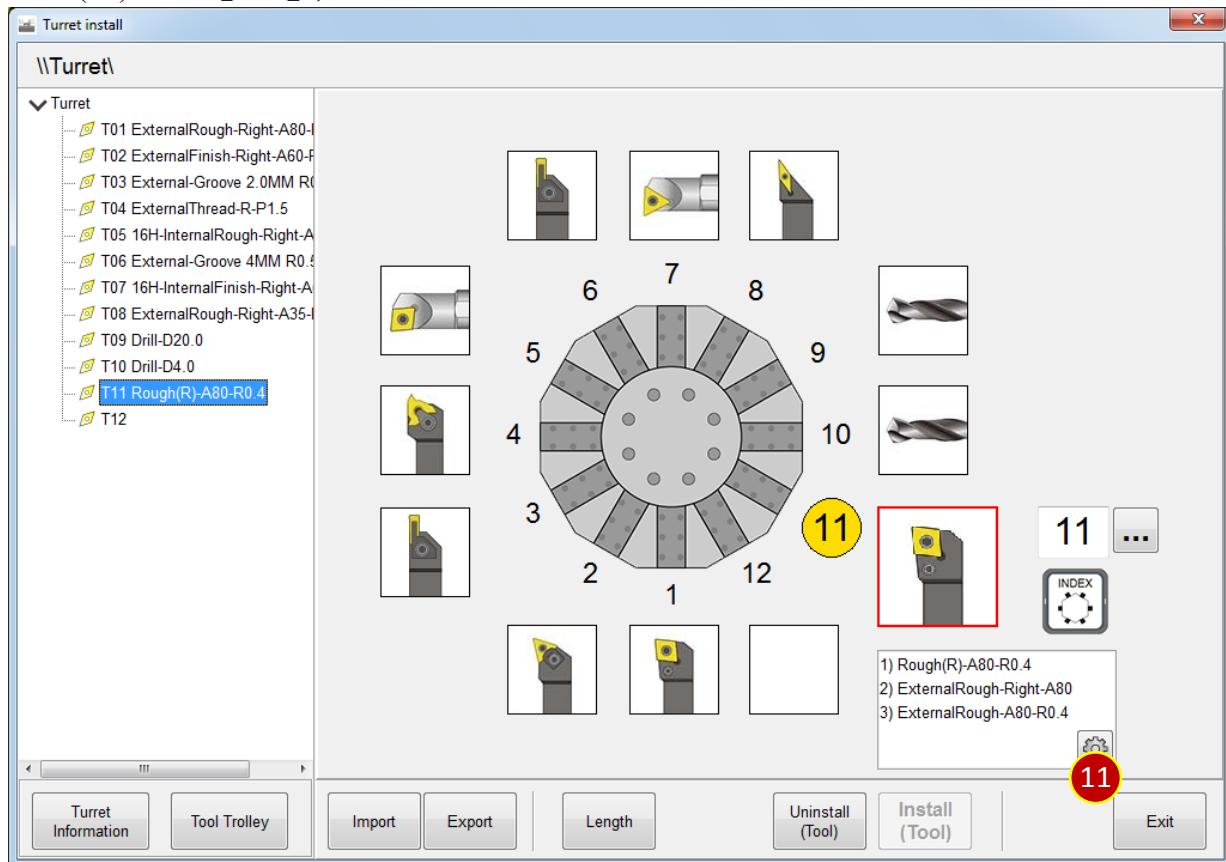


## Manual

(10) Select No.1 [Rough(R)-A80-R0.4] in the Trolley



(11) Press 【OK】 , the tool will be installed to Turret



### 3. Jaw Installation and Operation

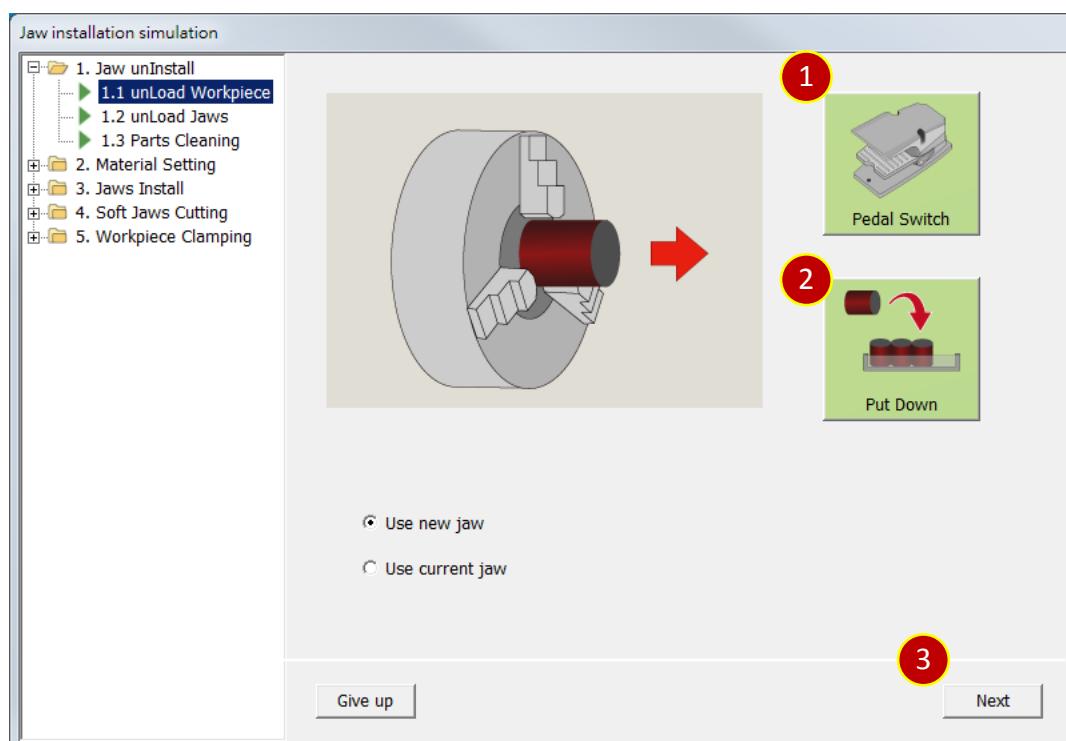
Press **Jaw Install** button to execute soft jaws cutting movement

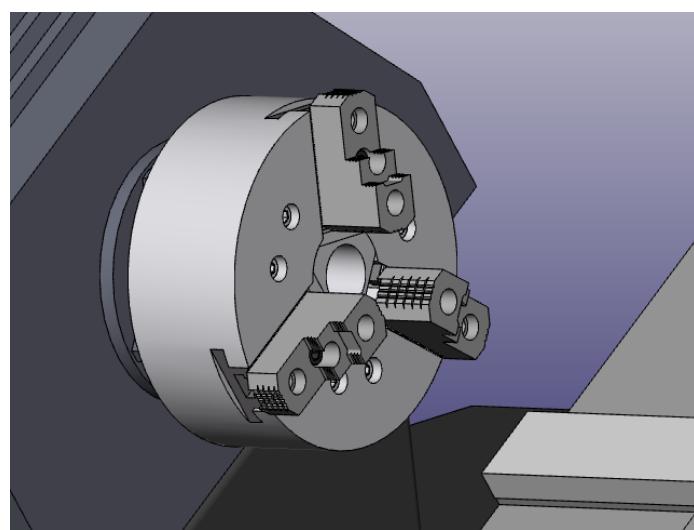


#### 3.1 Uninstall Jaw

##### 3.1.1 Unload Workpiece

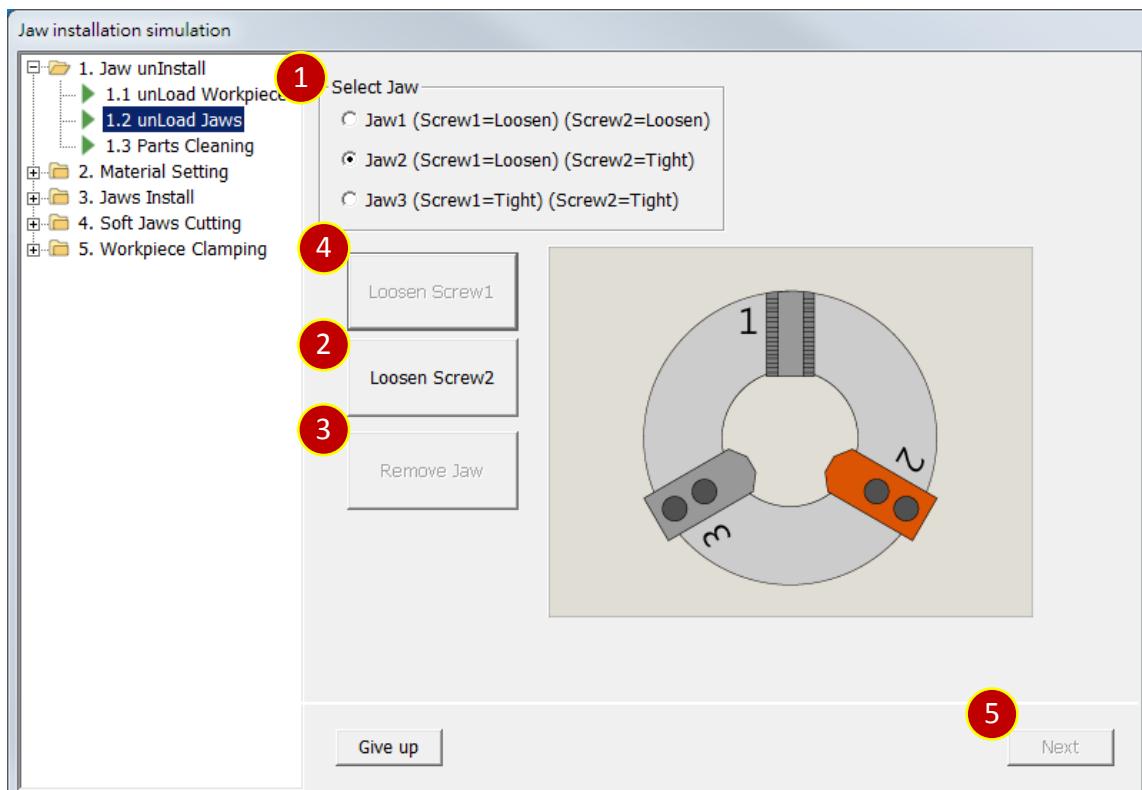
- (1) Press **Pedal Switch** to open the jaw
- (2) Press **Put Down** to put down the workpiece
- (3) Press **Next**

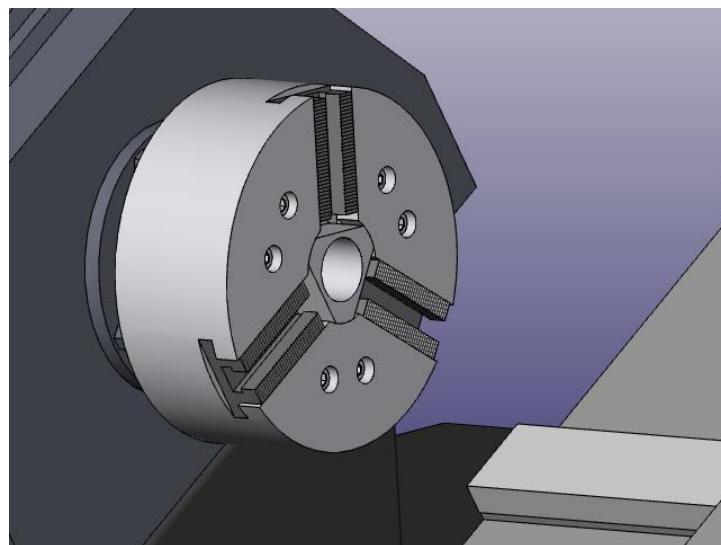




### 3.1.2 Unload Jaws

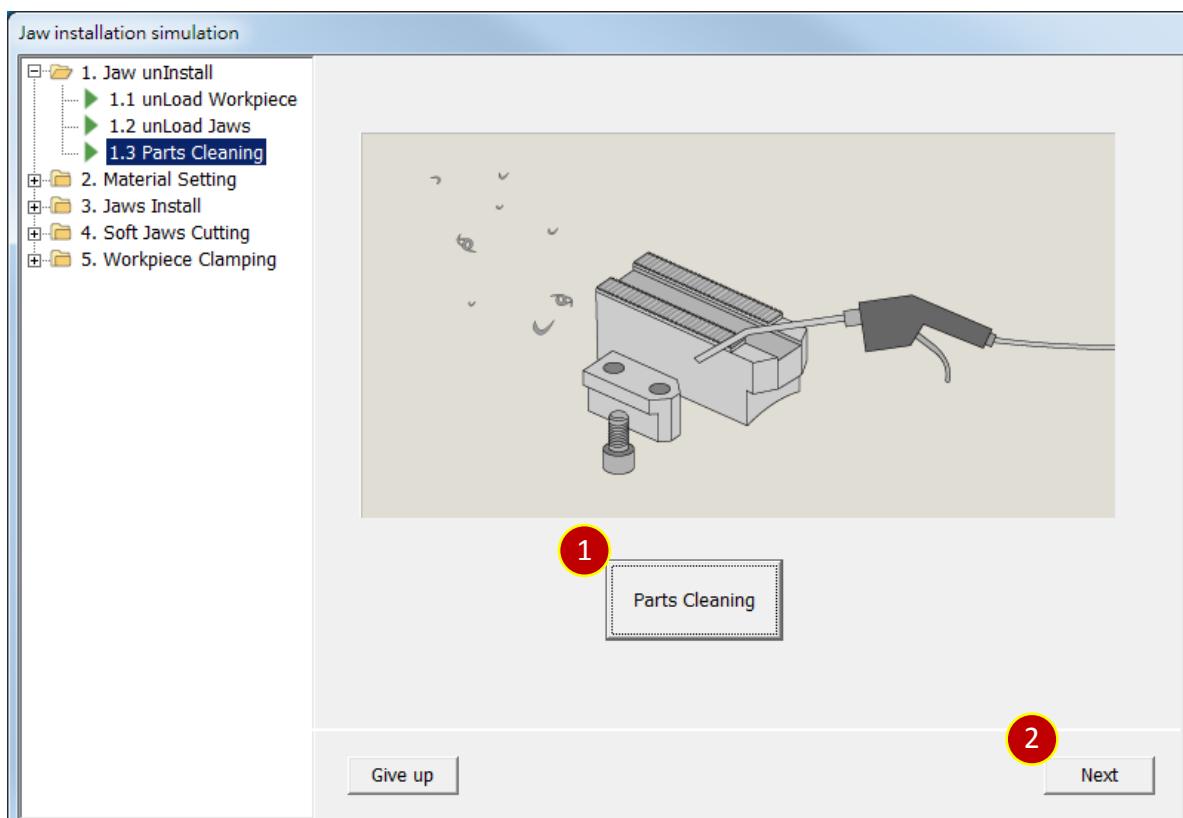
- (1) Select **【Jaw1】**
- (2) Press **【Loosen Screw 1】** and **【Loosen Screw 2】**
- (3) Press **【Remove Jaws】**
- (4) Select **【Jaw 2】** and **【Jaw 3】**, loosen screw 1 and screw 2 to unload jaw
- (5) Press **【Next】** after removing all jaws





### 3.1.3 Parts Cleaning

- (1) Press 【Parts Cleaning】 to clean the chips of parts
- (2) Press 【Next】 to enter material setting operation



## Manual

## 3.2 Material Setting

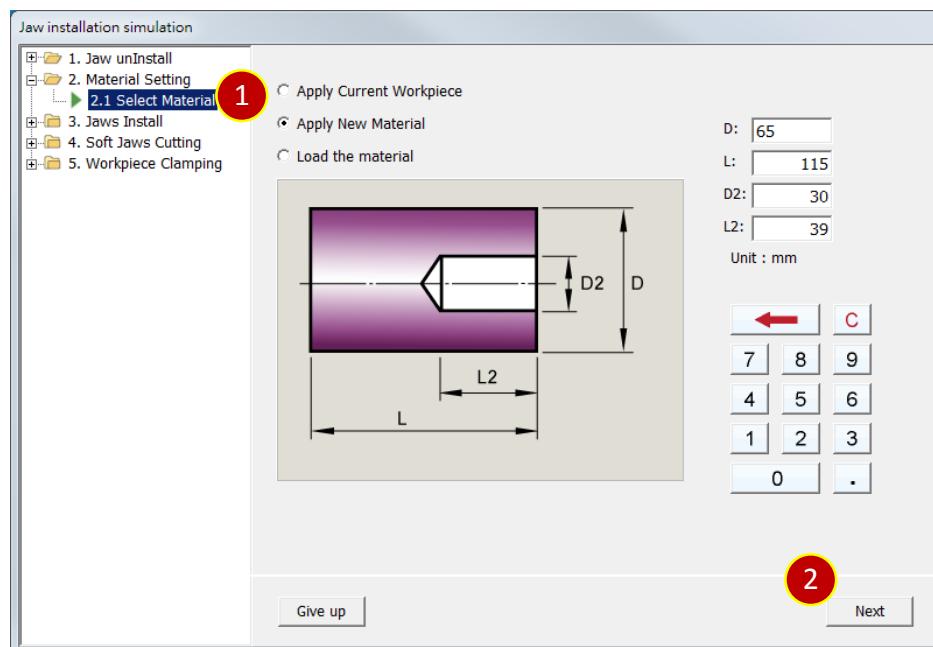
## 3.2.1 Select Material

(1) Choose the material to be clamped

e.g. Select 【Apply New Material】

Diameter(D)=65.0, Length(L)= 115.0, Hole (D2)=30.0, (L2)=39.0

(2) Press 【Next】 to enter jaws installation operation



## Manual

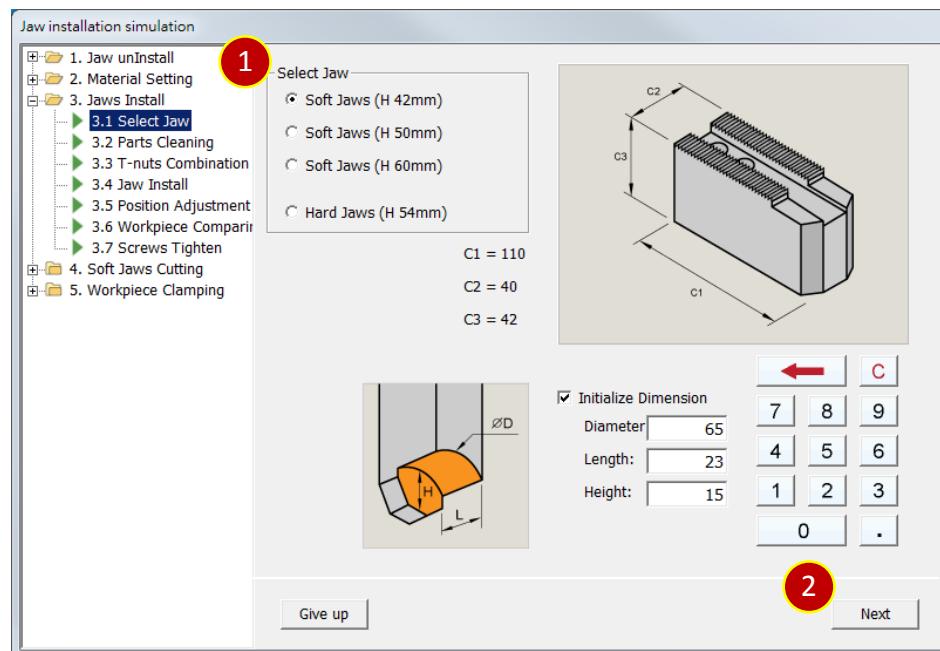
## 3.3 Jaw Install

## 3.3.1 Select Jaws

## (1) Select Jaw

e.g. Select 【Soft Jaws (Height:42mm)】

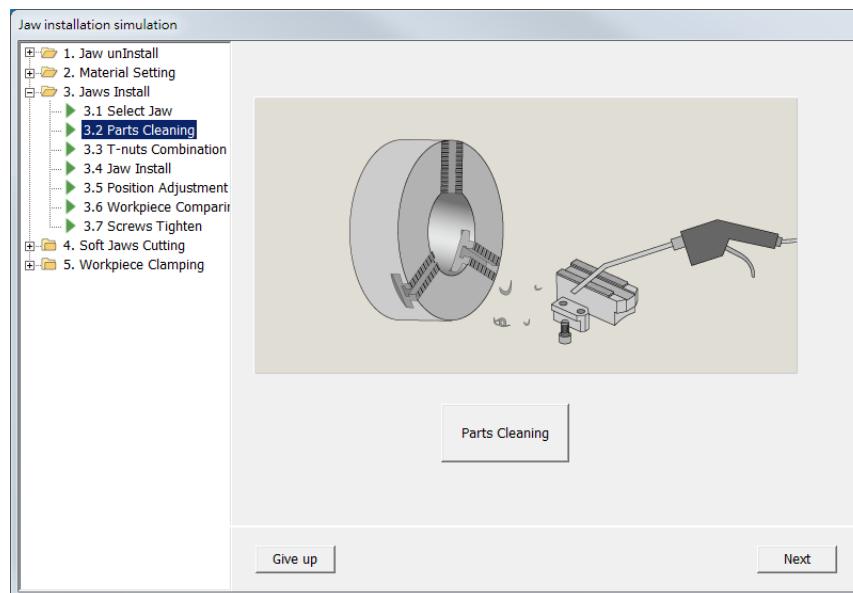
## (2) Press 【Next】



## Manual

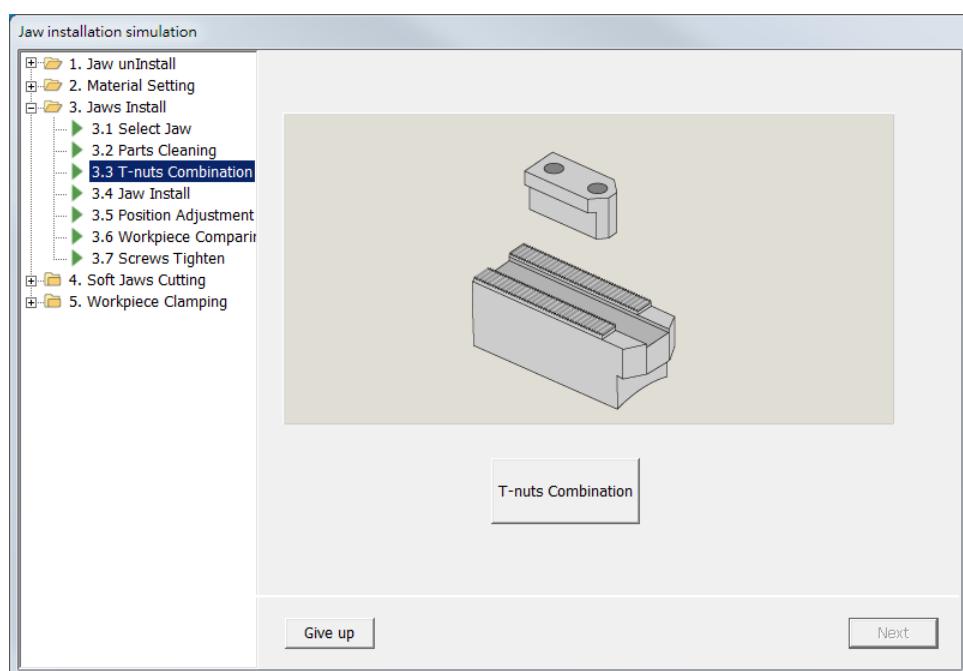
### 3.3.2 Parts Cleaning

- (1) Press 【Parts Cleaning】 to clean the chips of parts
- (2) Press 【Next】



### 3.3.3 T-nuts Combination

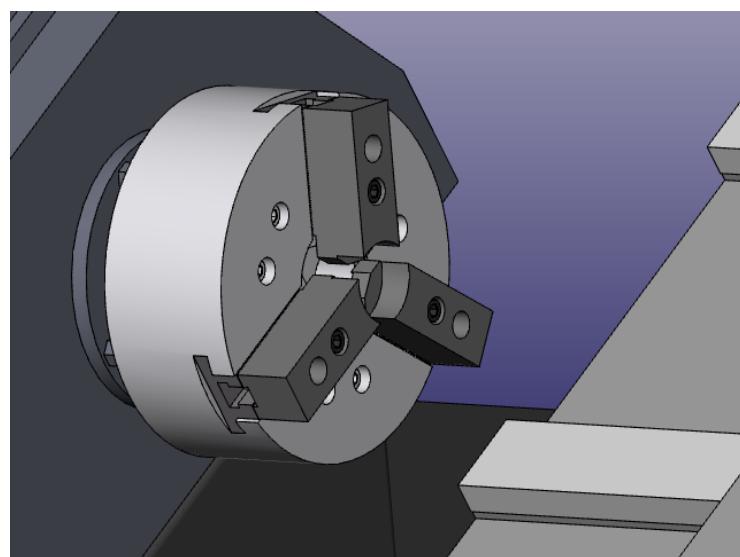
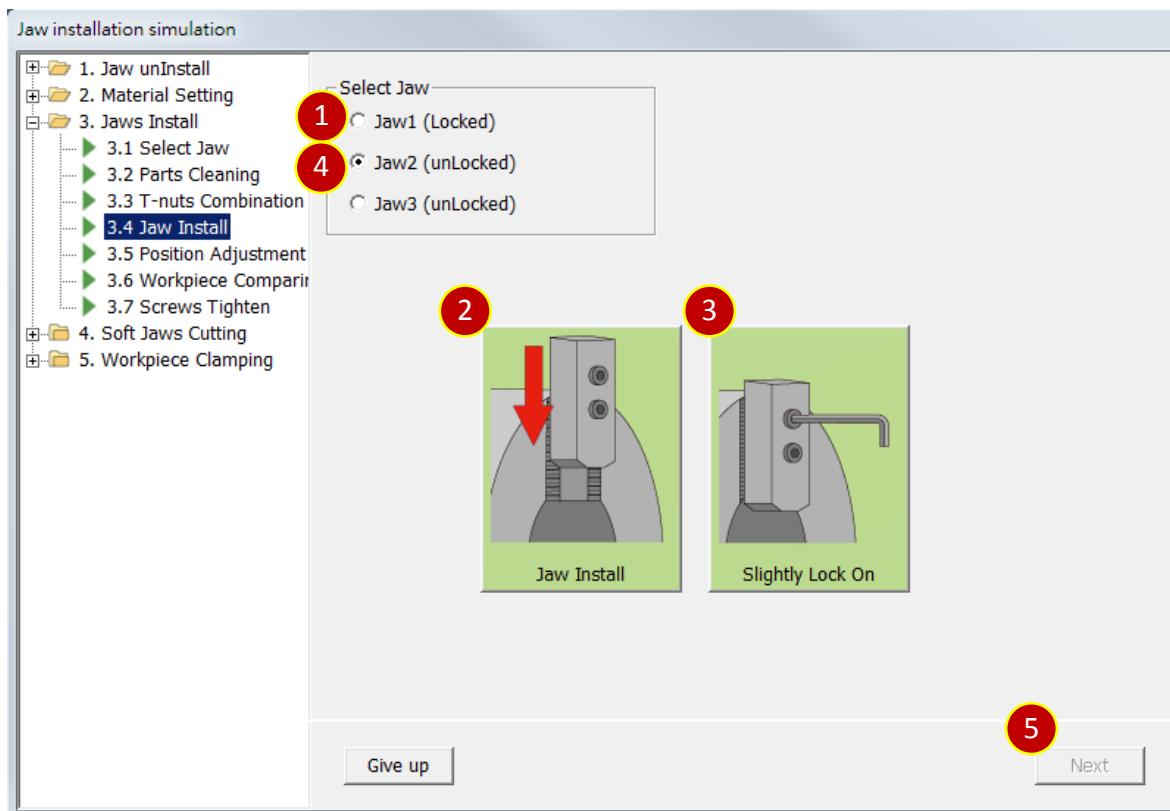
- (1) Press 【T-nuts Combination】 to combine the jaw parts
- (2) Press 【Next】



## Manual

### 3.3.4 Jaw Install

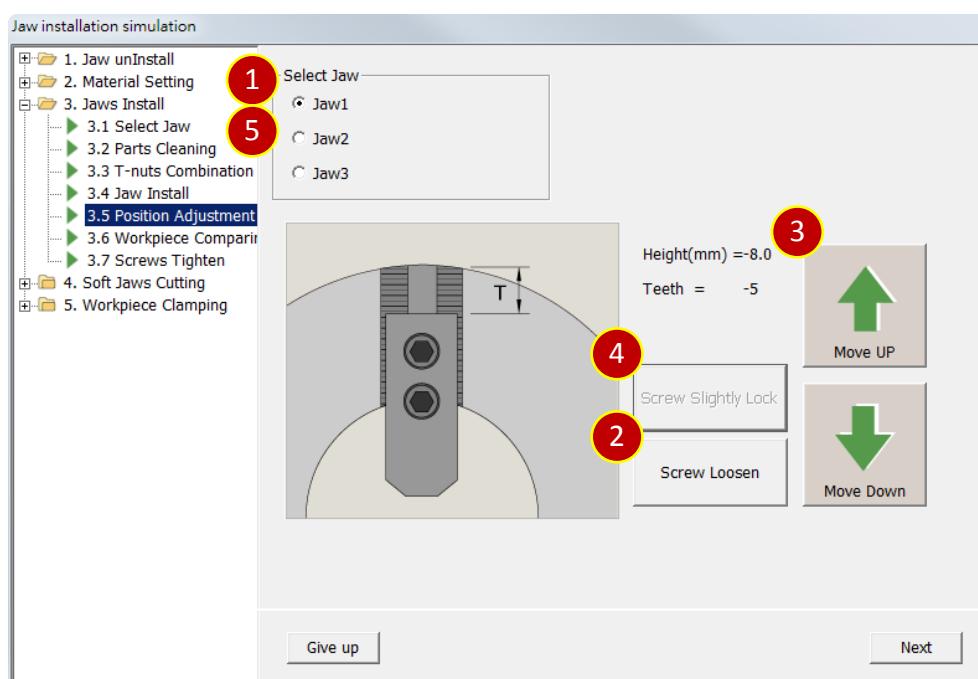
- (1) Select 【Jaw 1】
- (2) Press 【Jaw Install】 to install jaw 1
- (3) Press 【Slightly Lock On】 to lock on the jaw 1 slightly
- (4) Select 【Jaw2】 and 【Jaw 3】 to lock on them slightly
- (5) Press 【Next】 after installing all jaws



## Manual

## 3.3.5 Position Adjustment

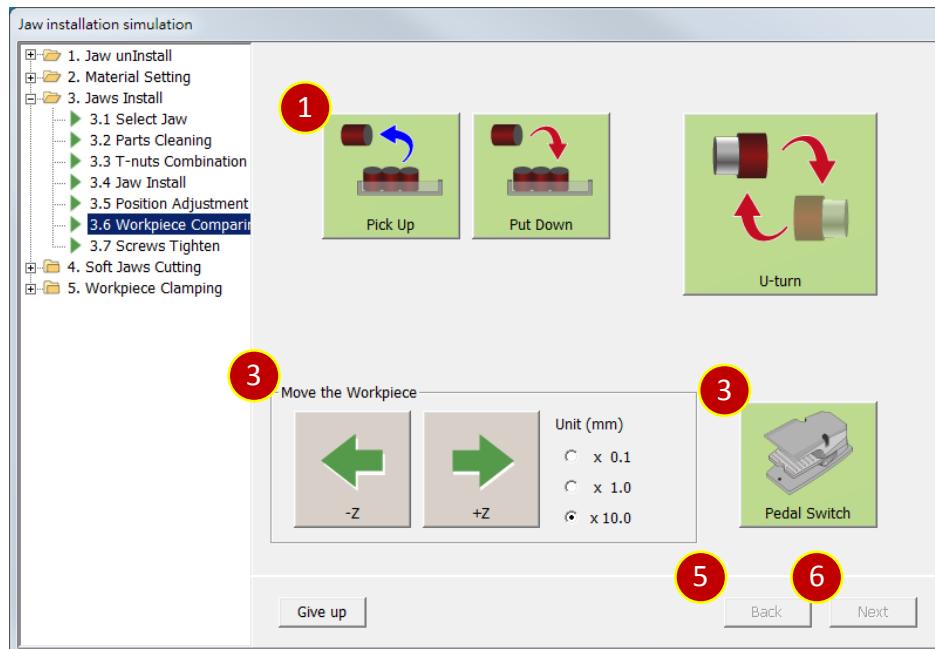
- (1) Select 【Jaw 1】
- (2) Press 【Screw Loosen】 to loosen the screw of jaw 1
- (3) Press 【Move Up】【Move Down】 to adjust the height of jaw  
e.g. Adjust the height to “-8.0”
- (4) Press 【Screw slightly Lock】 to lock on the screw slightly
- (5) Select 【Jaw 2】【Jaw 3】 to adjust the position of jaw2 and jaw3,  
to make them identical to the height of jaw 1
- (6) Press 【Next】

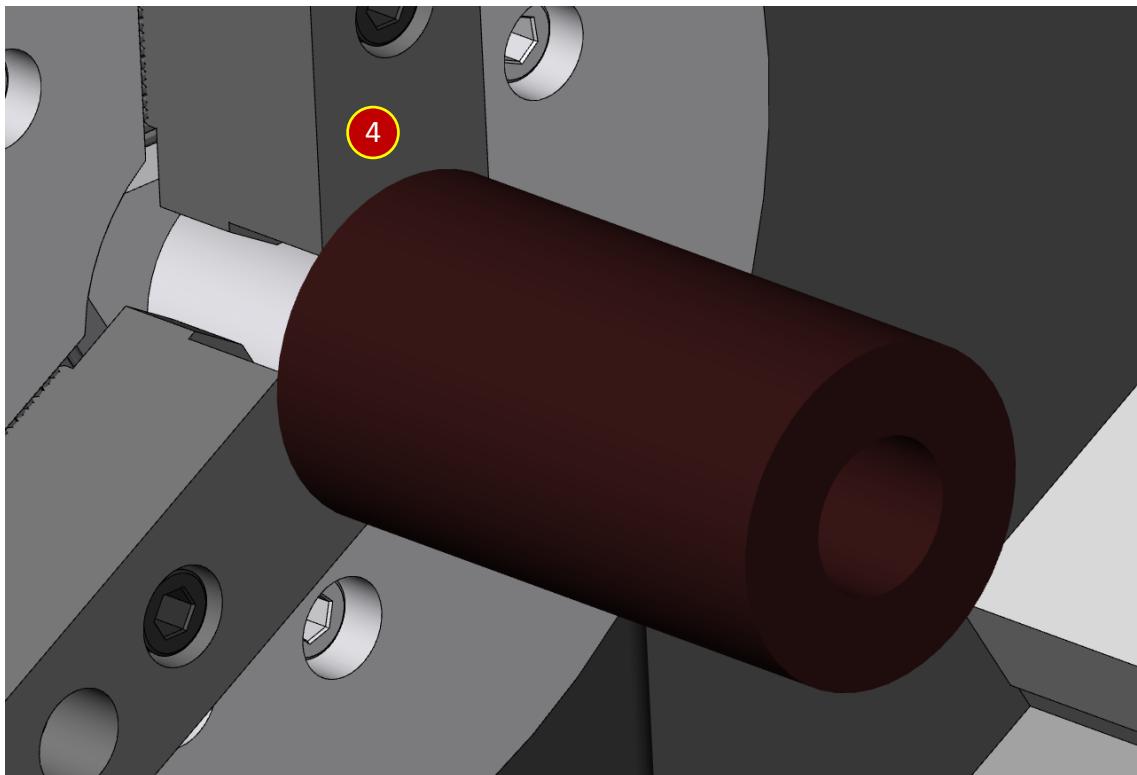
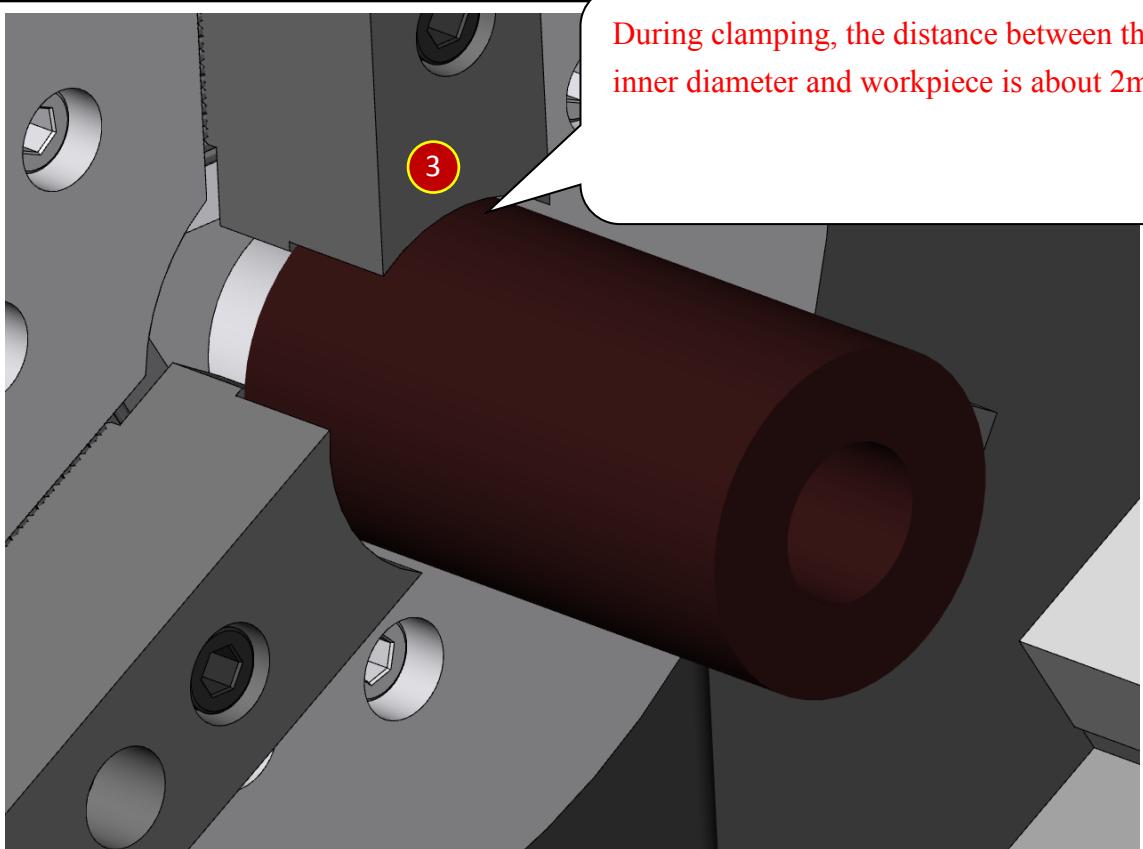


## Manual

## 3.3.6 Soft Jaw Position and Size Comparison

- (1) Make sure the soft jaws can clamp the material during clamping movement  
(Travel Path=8mm)
- (2) Press 【Pick up】
- (3) When the soft jaw is open, make sure the workpiece can fit in it
  - (a) Press 【Pedal Switch】 to open the soft jaw
  - (b) Press 【-Z】 to install the workpiece on the Soft Jaws  
and make sure the workpiece can fit in
  - (c) Press 【+Z】 to move the workpiece away from the soft jaws
- (4) When the soft jaws is clamping, make sure the workpiece doesn't fit in the soft jaws
  - (a) When the soft jaws is clamping, the workpiece can't be inside of the soft jaw.  
Make sure the workpiece is away from it
  - (b) Press 【Pedal Switch】 to clamp the soft jaw
  - (c) Press 【-Z】 to install the workpiece on the Soft Jaws  
Make sure the workpiece doesn't fit in
- (5) The recommend distance between jaws inner diameter and workpiece during the activity is about 2mm, it is easier for setting and remove the material
- (6) Press 【Put Down】 to put down the workpiece
- (7) Press 【Next】 to execute the soft jaws cutting
  - (a) When the soft jaws is not match with the workpiece dimension,  
Press 【Back】 to readjust the soft jaws position





## Manual

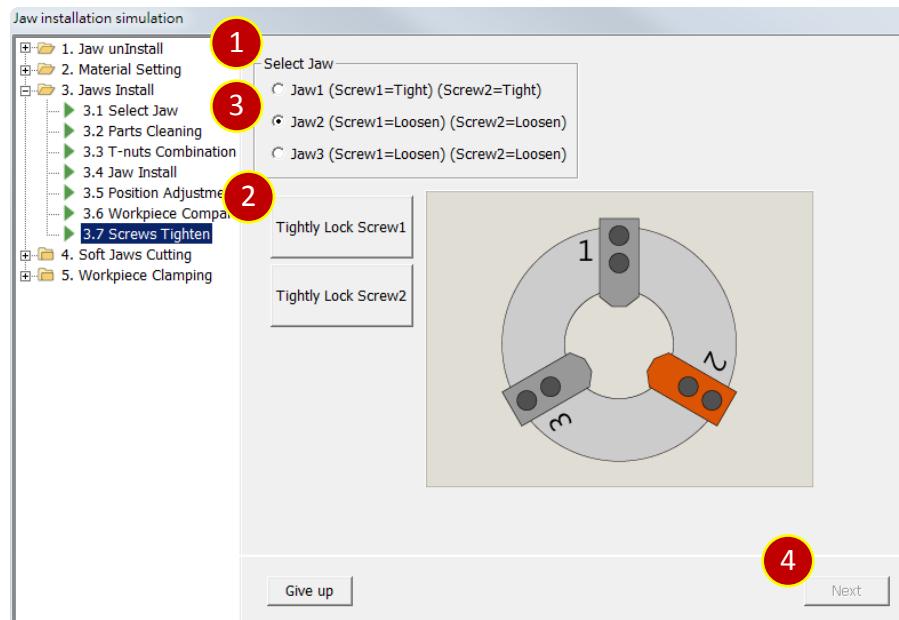
## 3.3.7 Screws Tighten

(1) Select 【Jaw 1】

(2) Press 【Tightly Lock Screw1】【Tightly Lock Screw2】 to finish jaw 1 installation

(3) Select 【Jaw 2】【Jaw 3】 to lock the screw1 and screw2 tightly

(4) Press 【Next】 to enter next step after tightly lock all of jaws



## Manual

## 3.4 Soft Jaw Modify

## 3.4.1 Pressure Adjustment

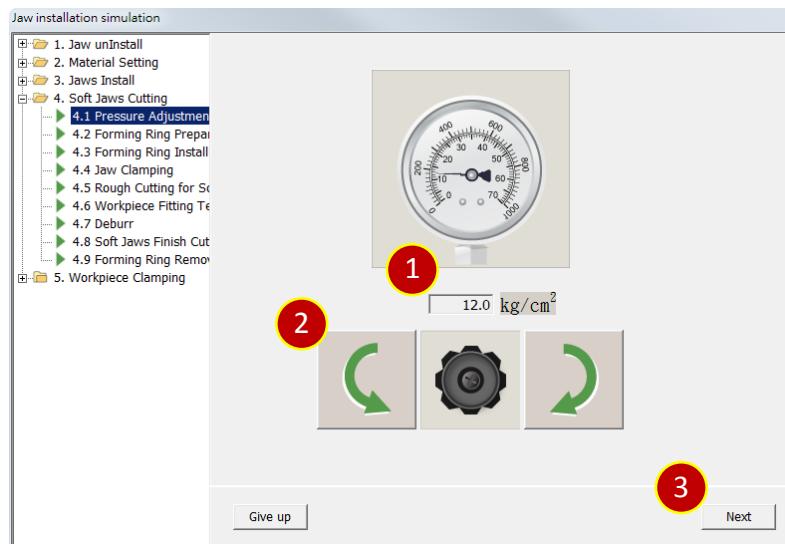
(1) When modifying soft jaws, the pressure adjustment value is between 8-13 kg/cm<sup>2</sup>

(2) Press 【Pressure Adjustment】 button to adjust the pressure e.g. 12.0kg/ cm<sup>2</sup>

Press left rotation button to decrease pressure,

Press right rotation to increase pressure

(3) Press 【Next】

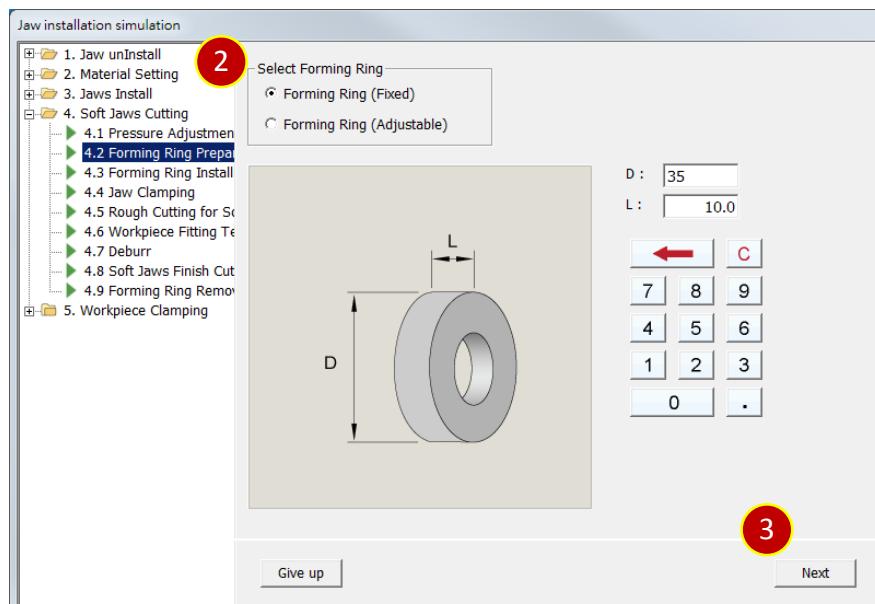


## 3.4.2 Forming Ring Prepare

(1) Select forming ring

e.g. Select 【Forming Ring (Fixed)】 D:35.0 L:10.0

(2) Press 【Next】



## Manual

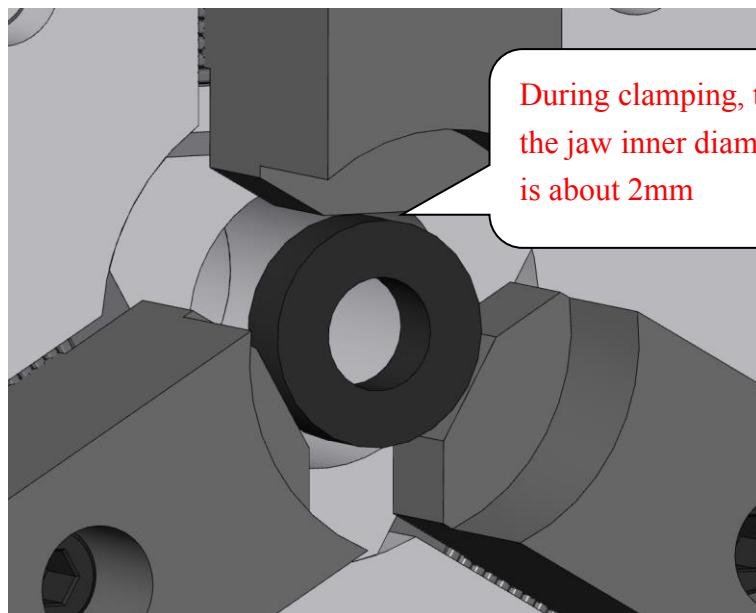
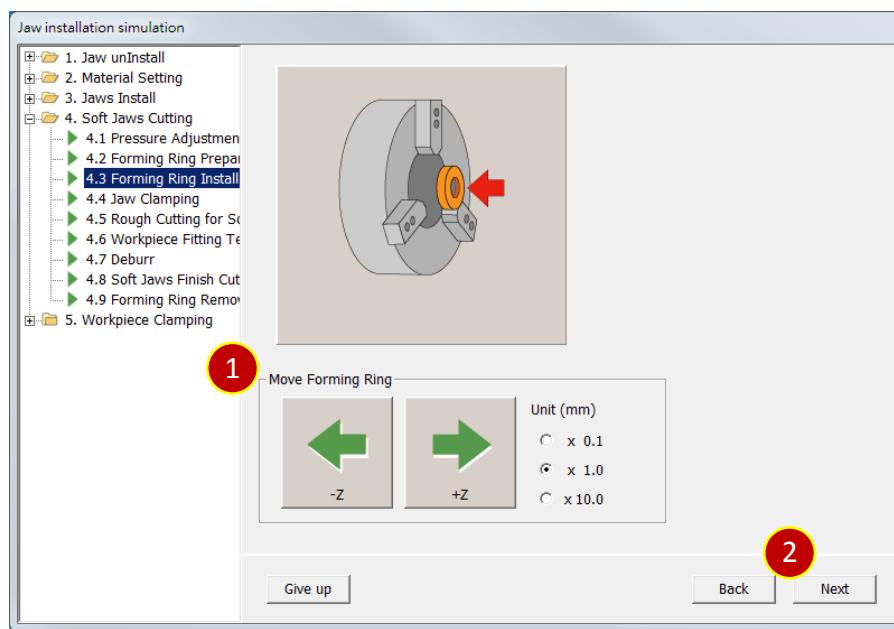
### 3.4.3 Forming Ring Install

Take forming ring as the soft ring clamping position.

Forming Ring can be classified into fixed and adjustable type

The recommend distance between jaws and forming ring during the activity is about 2mm, it is easier for setting and removal of the material

- (1) Press **【-Z】** **【+Z】** to move forming ring to the position of Soft Jawss
- (2) Press **【Next】**



## Manual

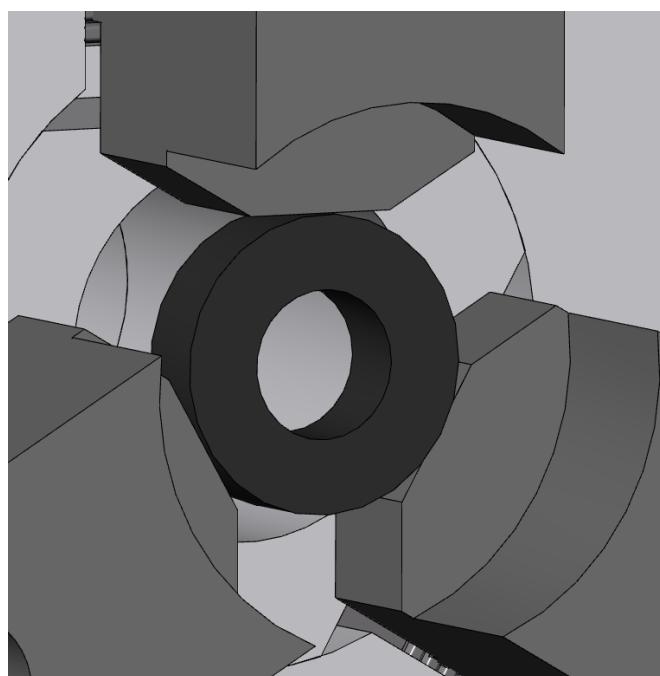
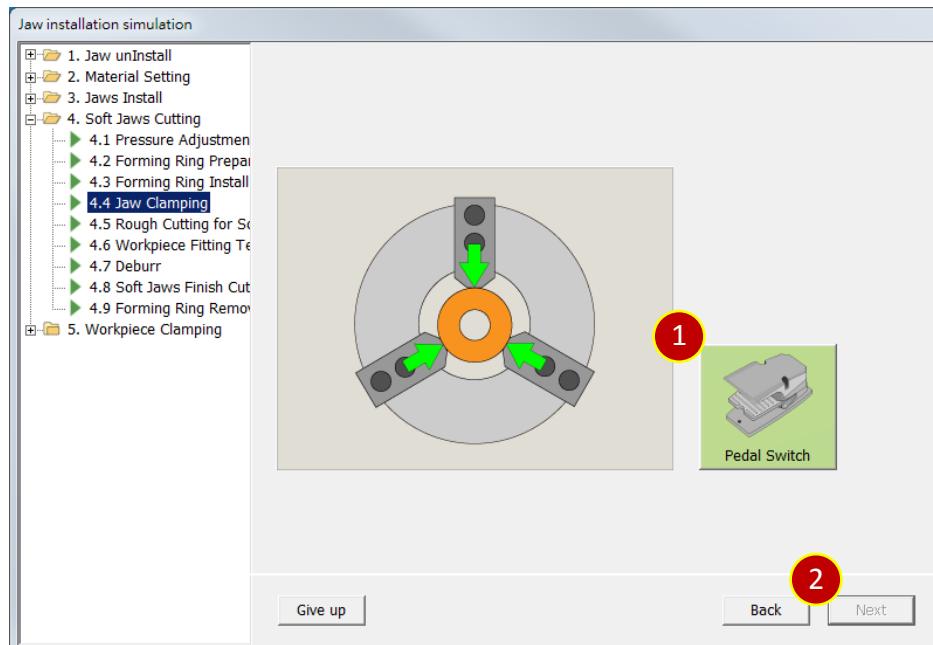
## 3.4.4 Hydraulic Clamping

(1) Press 【Pedal Switch】 to clamp the forming ring

(2) Press 【Next】 after finish clamping

(a) When the soft jaw dimension is not match with the forming ring,

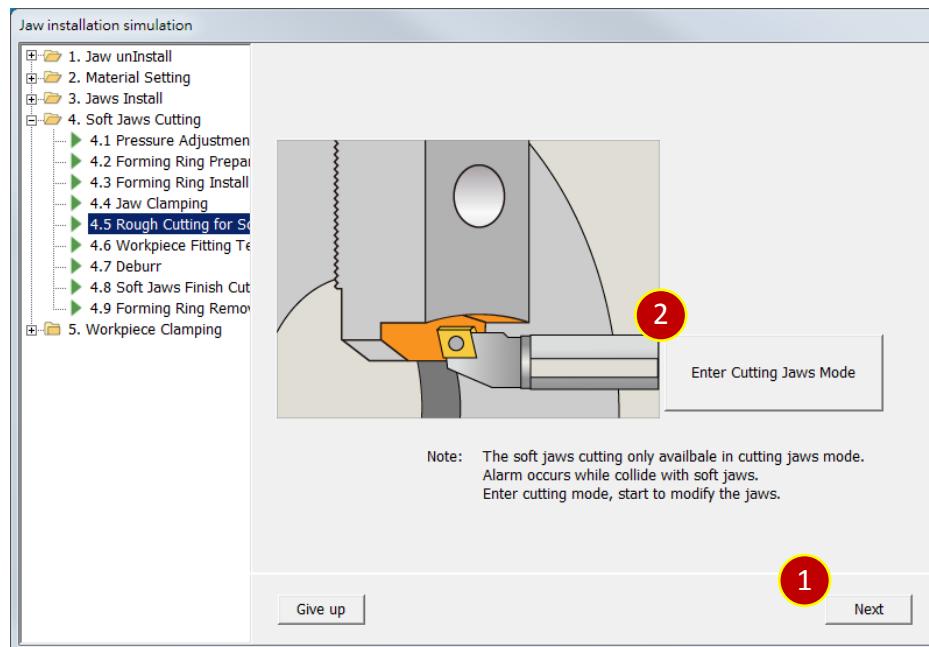
Press 【Back】 to change forming ring dimension



## Manual

## 3.4.5 Rough Cutting for Soft Jaw

- (1) The forming soft jaw doesn't need to be cut, Press 【Next】 to execute next steps
- (2) If need to rough cut the jaws, Press 【Enter Cutting Jaws Mode】



## Manual

## 3.4.6 Workpiece Fitting Test

(1) Press 【Pick Up】

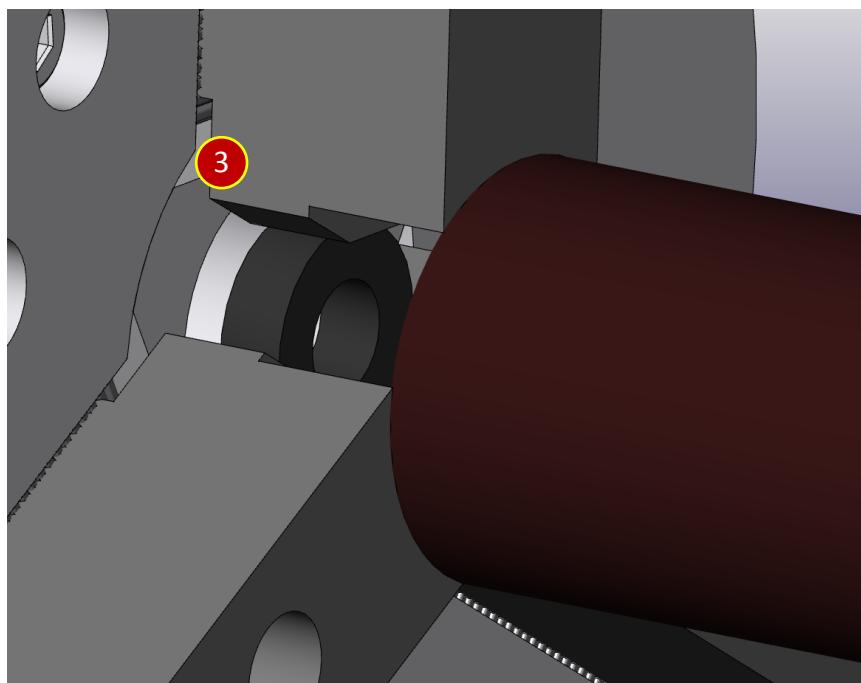
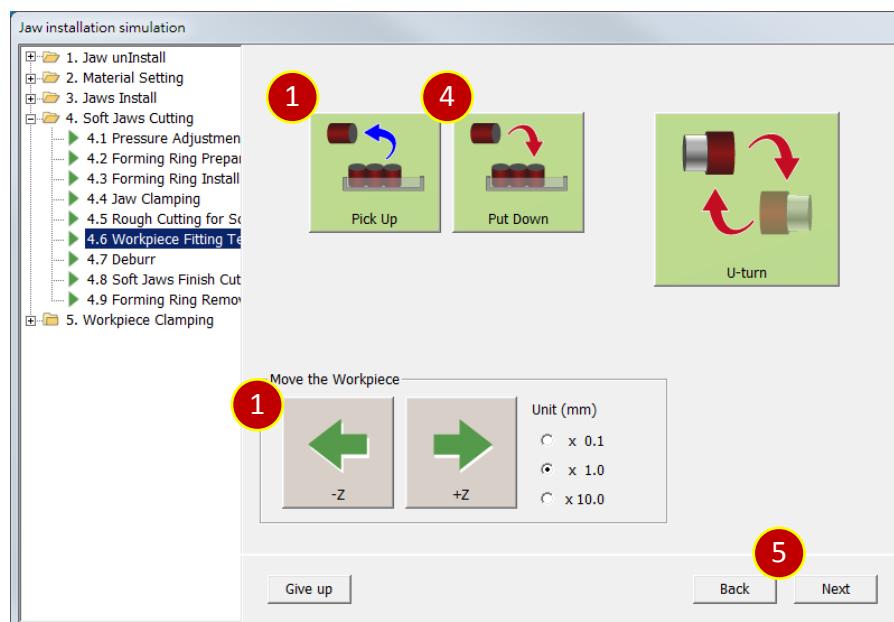
(2) Press 【-Z】 to put the workpiece on the soft jaw to compare

(3) Make sure the soft jaw clamping position is identical to workpiece dimension

e.g. The workpiece dimension is about 1mm below larger than soft jaw

(4) Press 【Put Down】 to put down the workpiece

(5) Press 【Press】

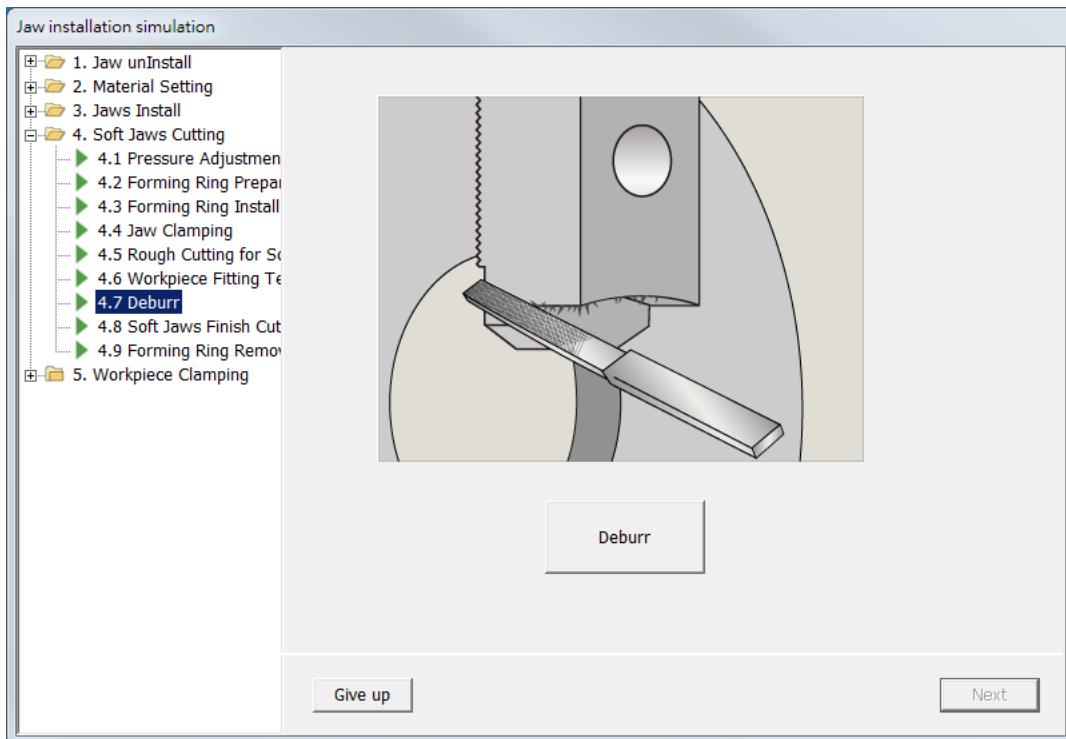


## Manual

## 3.4.7 Deburr

(1) Press 【Deburr】

(2) Press 【Next】



## Manual

### 3.4.8 Soft Jaw Finish Cut Preparation

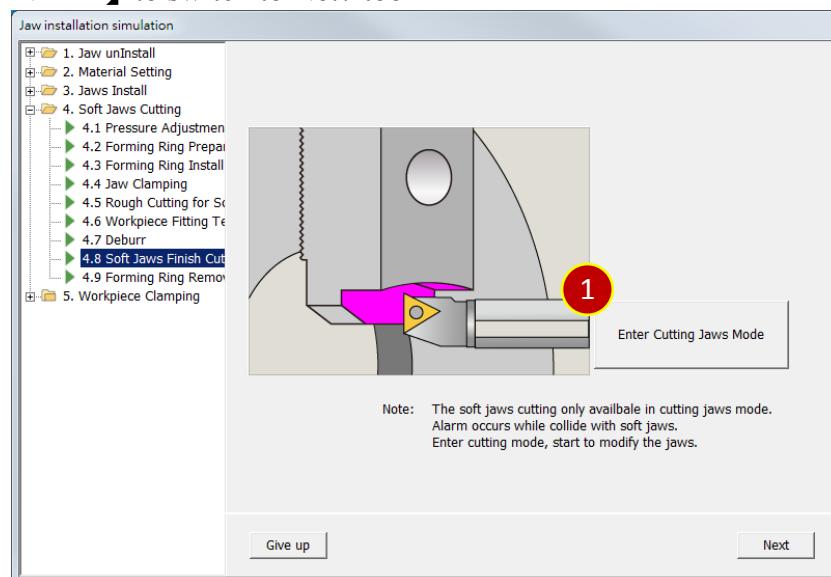
The required tool for cutting soft jaw is inner diameter finishing tool e.g. No.7  
(Tool Nose Radius R=0.4)

(1) Press 【Enter Cutting Jaws Mode】

(2) Switch to RAPID mode

(3) Before manual operation, switch 【Rapid Override】 to 50% (or 25%),  
switching to 100% is forbidden

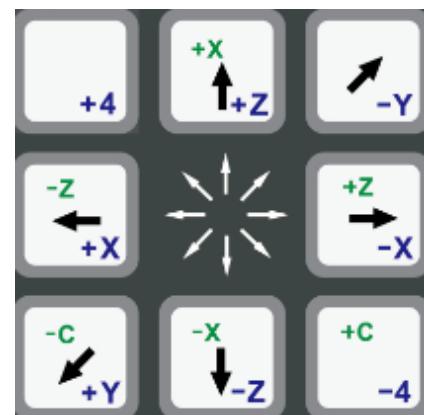
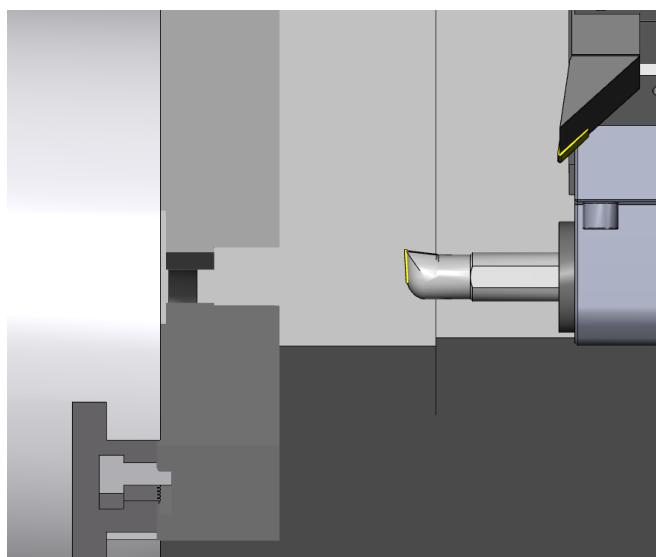
(4) Press 【INDEX】 to switch to No.7 tool





(5) Press Axial Control Button to move the tool rapidly

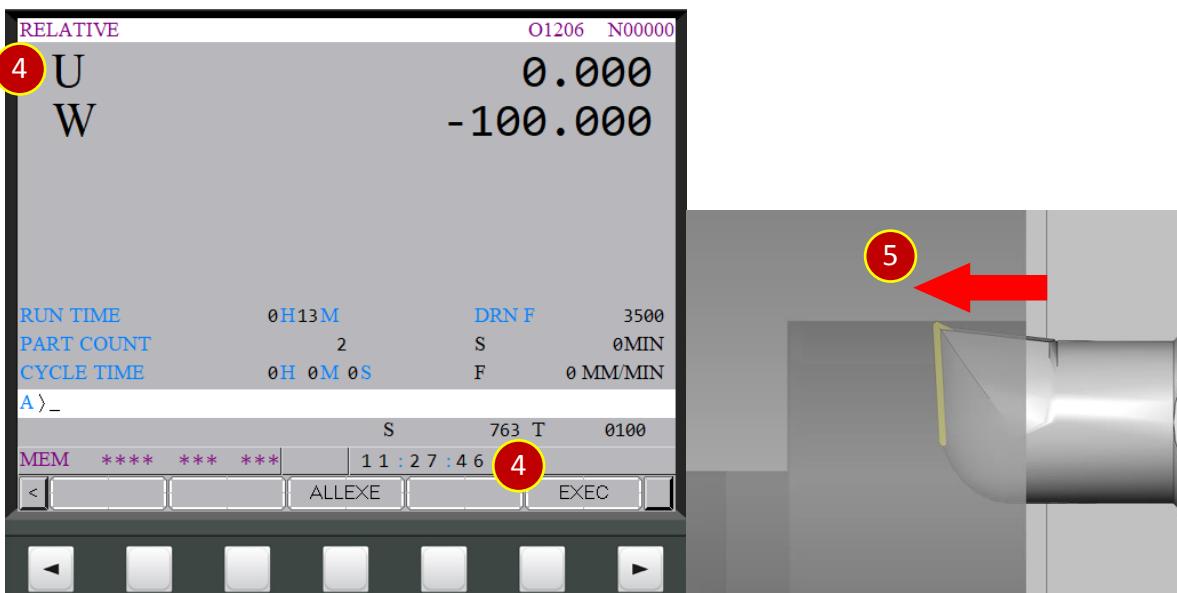
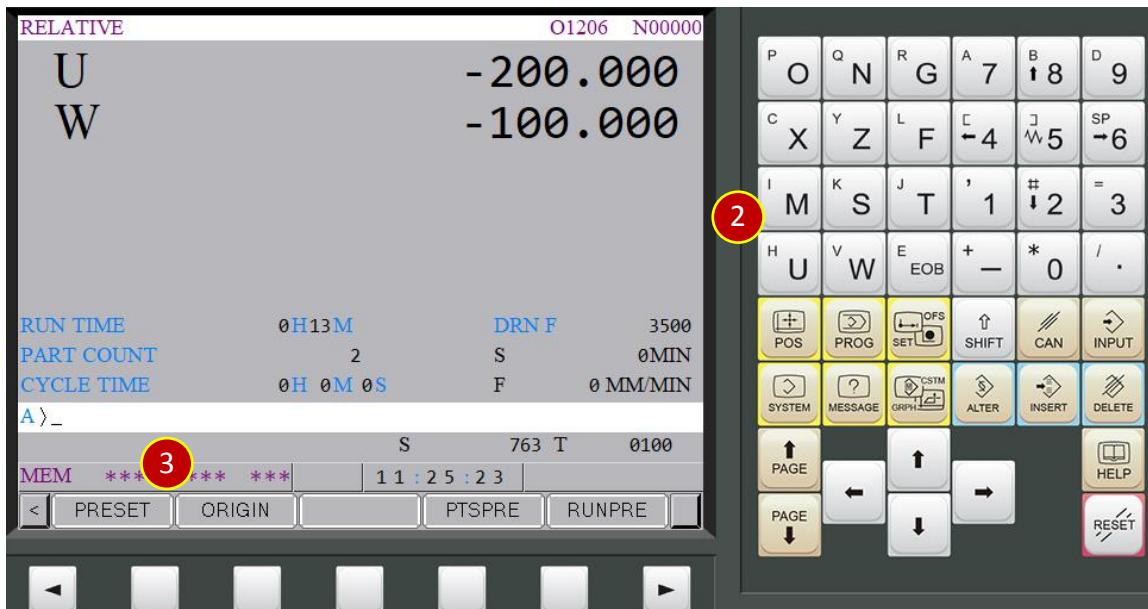
to the position about 50mm left to the soft jaw



## Manual

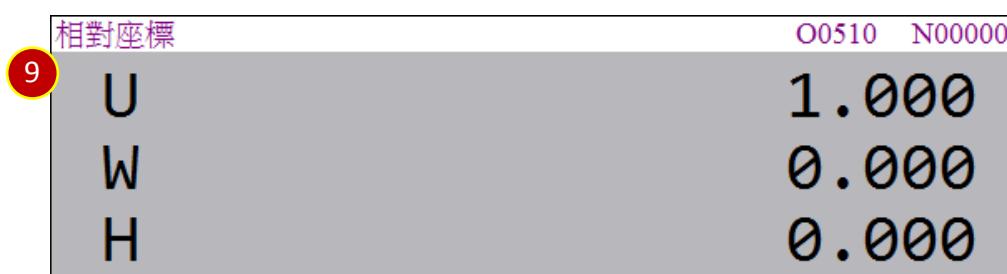
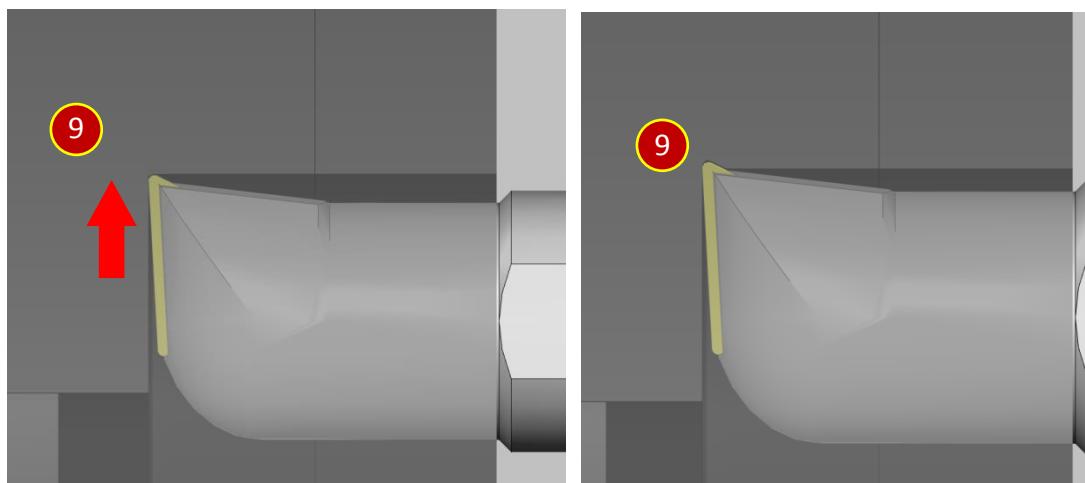
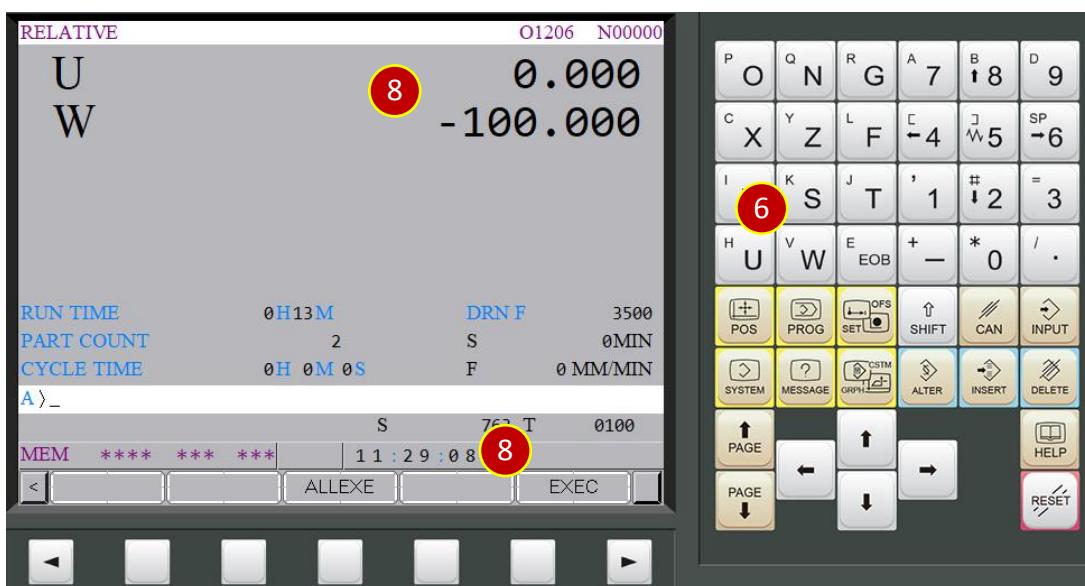
### 3.4.9 Soft Jaw Finish Cutting

- (1) Manually cuts the Soft Jaw inner diameter for about 0.1mm via Handwheel
- (2) Press 【U】
- (3) Press 【Origin】
- (4) Press 【Execute】 to zero the relative coordinate U
- (5) Manually cuts it till reached the bottom of the Soft Jaw via Handwheel



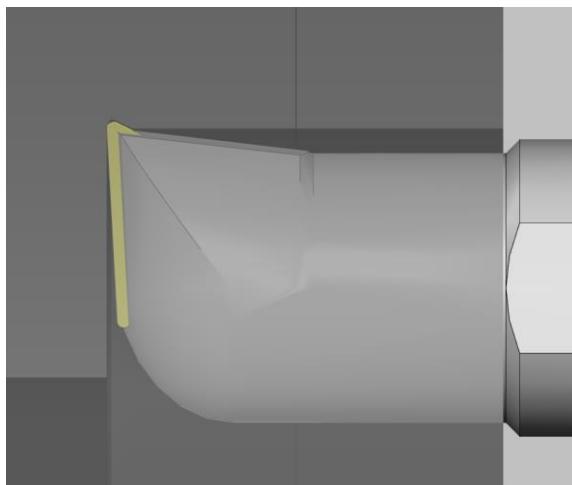
## Manual

- (6) Press 【W】
- (7) Press 【Origin】
- (8) Press 【Execute】 to zero the relative coordinate W
- (9) Lift the tool upward for 1.0 mm to avoid Corner Rounding (Tool Radius R0.4)



## Manual

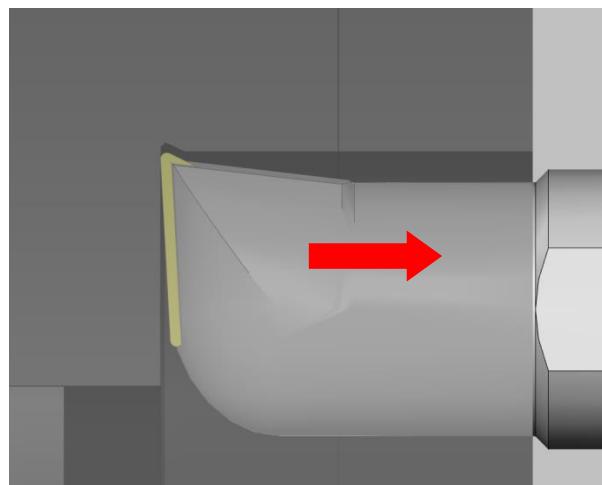
- (10) Keep cutting downward in (-X) direction till leaving the soft jaw
- (11) Move the tool in -Z for 0.02mm
- (12) Cutting upward in (+X) till slotting position



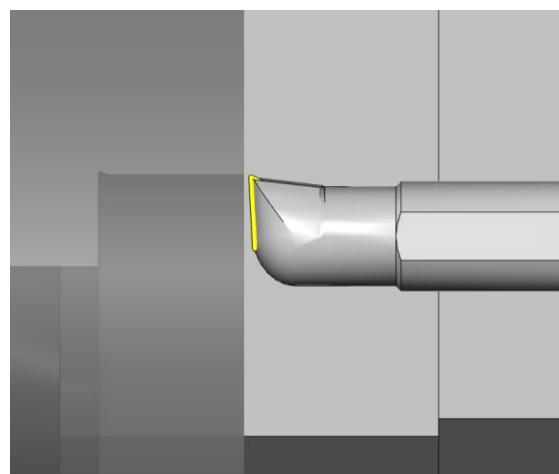
相對座標		O0510 N00000
U	1.000	
W	-0.020	
H	0.000	

- (13) After retracting to the position

where the soft jaw inner diameter was cuts for 0.02 mm (Retracts 0.98mm),  
cutting in +Z till reaching soft jaw external diameter Endface

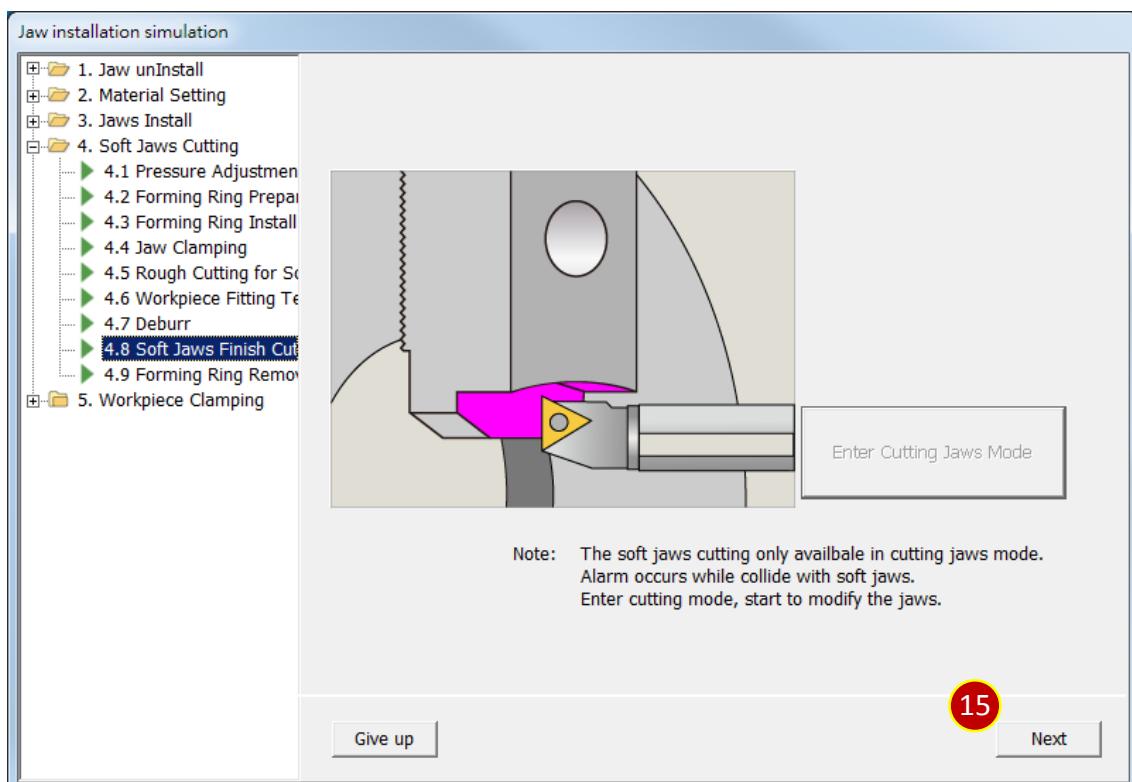


相對座標		O0510 N00000
U	0.020	
W	-0.020	
H	0.000	



## Manual

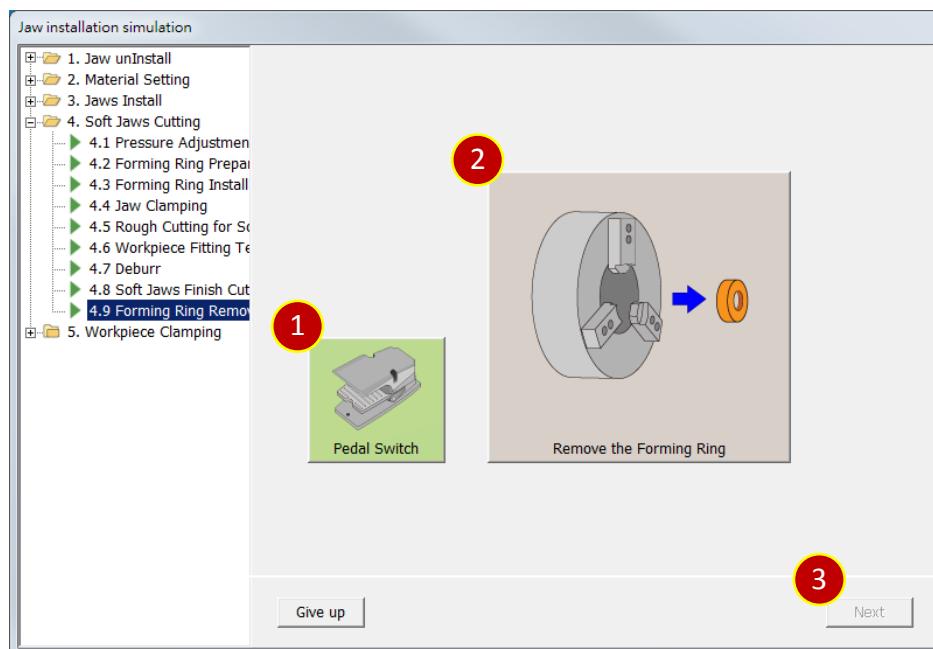
- (14) Press 【Stop】 to stop spindle rotation
- (15) Press 【Next】 after finish cutting soft jaw



## Manual

## 3.4.10 Remove the Forming Ring

- (1) Press 【Pedal Switch】 to open jaw
- (2) Press 【Remove the Forming Ring】
- (3) Press 【Next】 to enter workpiece clamping procedure



## Manual

### 3.5 Workpiece Clamping

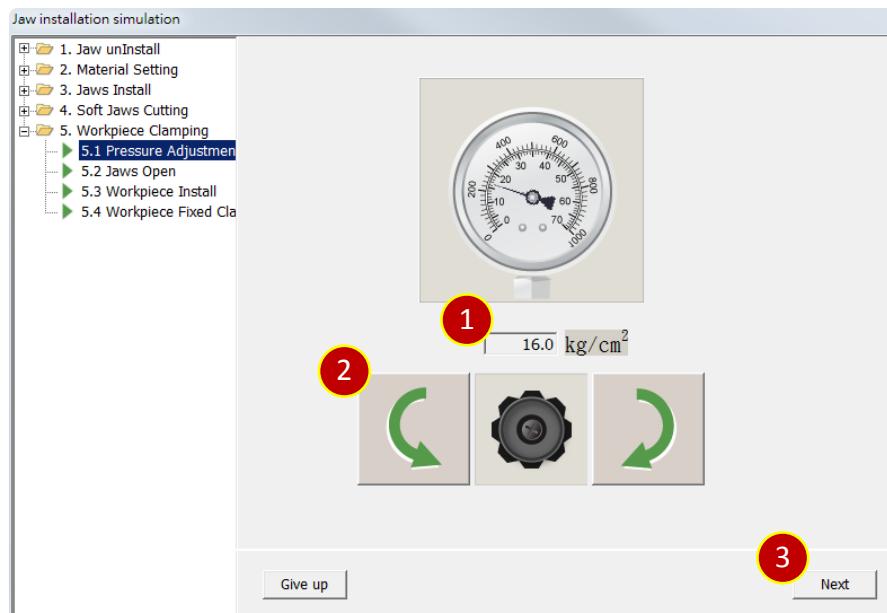
#### 3.5.1 Pressure Adjustment

(1) When clamping workpiece, the pressure value is between 12-20 kg/ cm<sup>2</sup>

(2) Press 【Pressure Adjustment】 button to adjust the pressure e.g. 16.0 kg/cm<sup>2</sup>

Press left rotation button to decrease pressure, Press right rotation to increase pressure

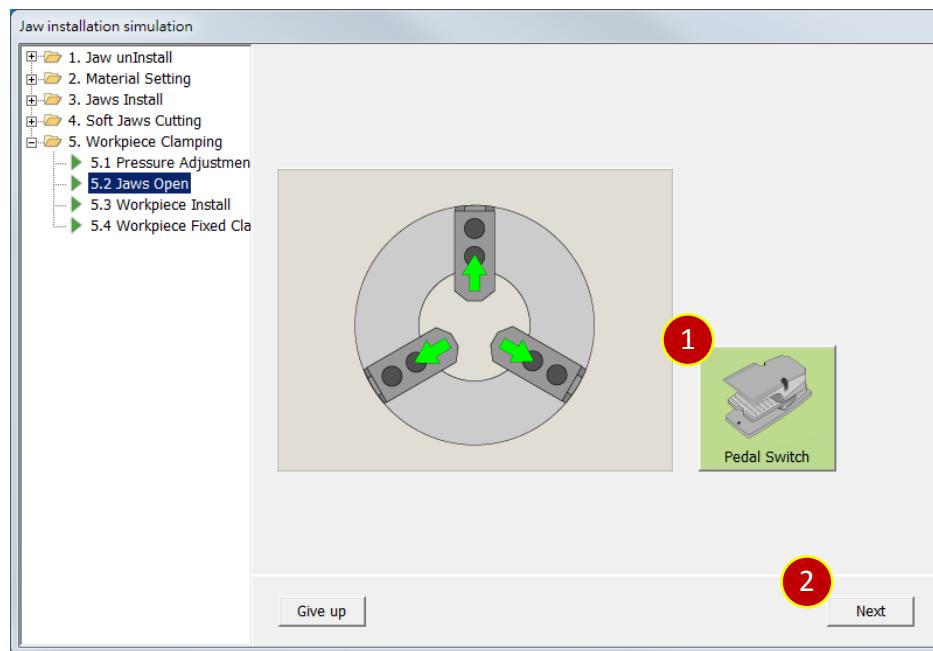
(3) Press 【Next】



## Manual

## 3.5.2 Jaws Open

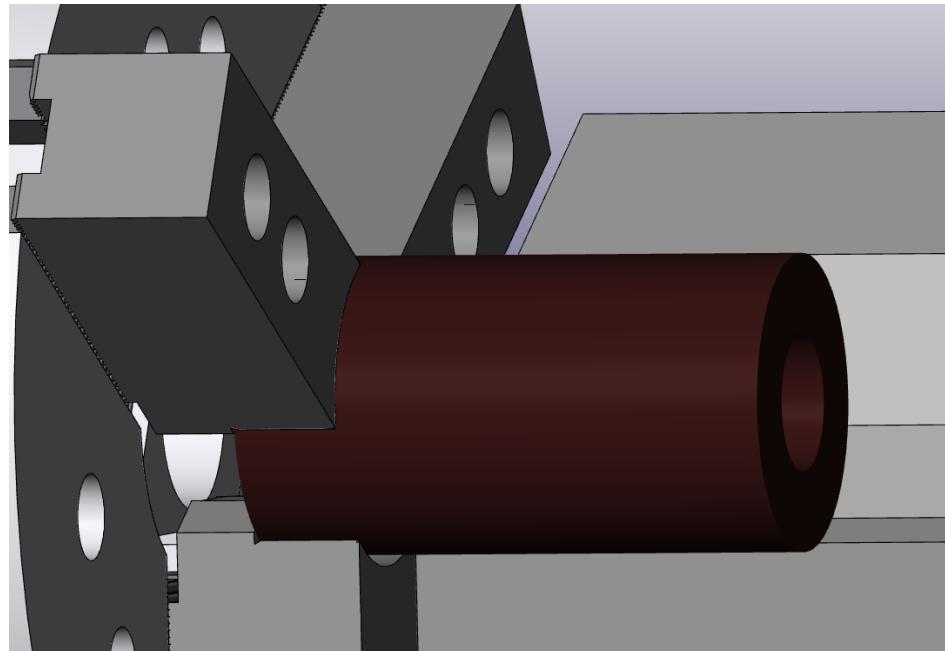
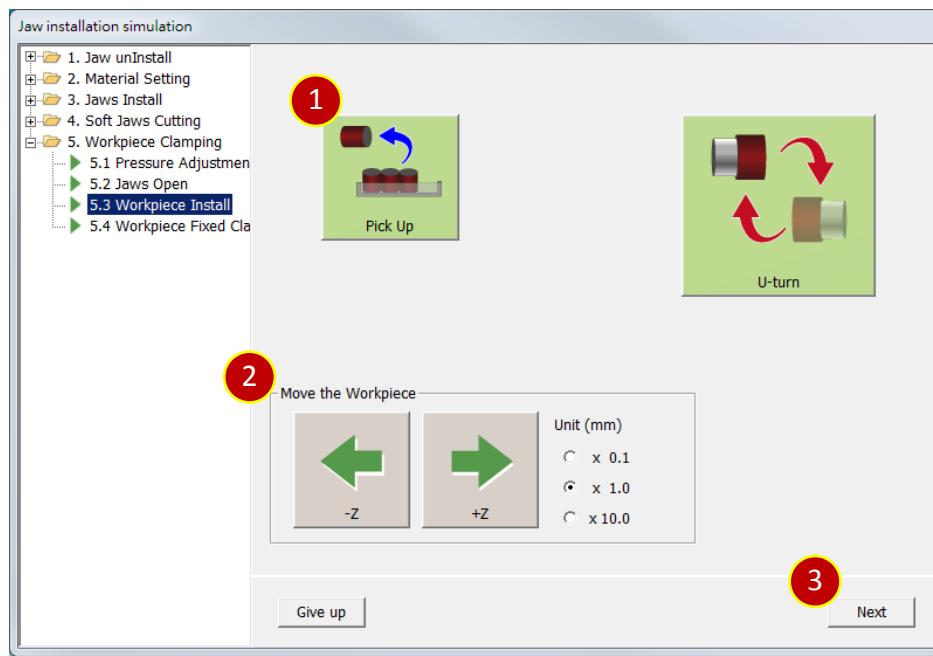
- (1) Check the jaw, if it is close Press 【Pedal Switch】 to open it.
- (2) Press 【Next】



## Manual

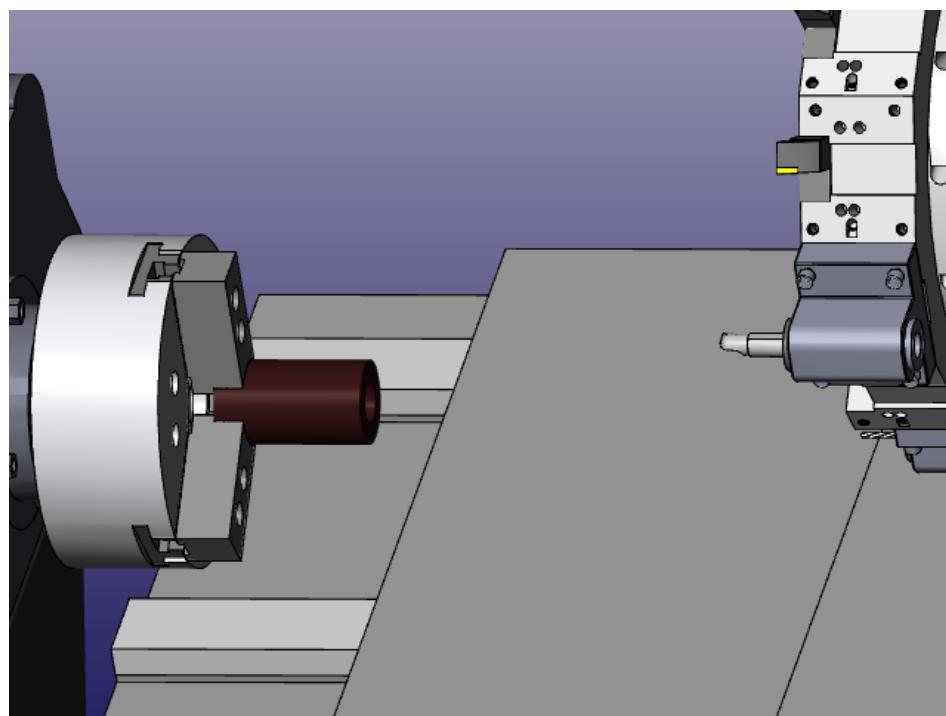
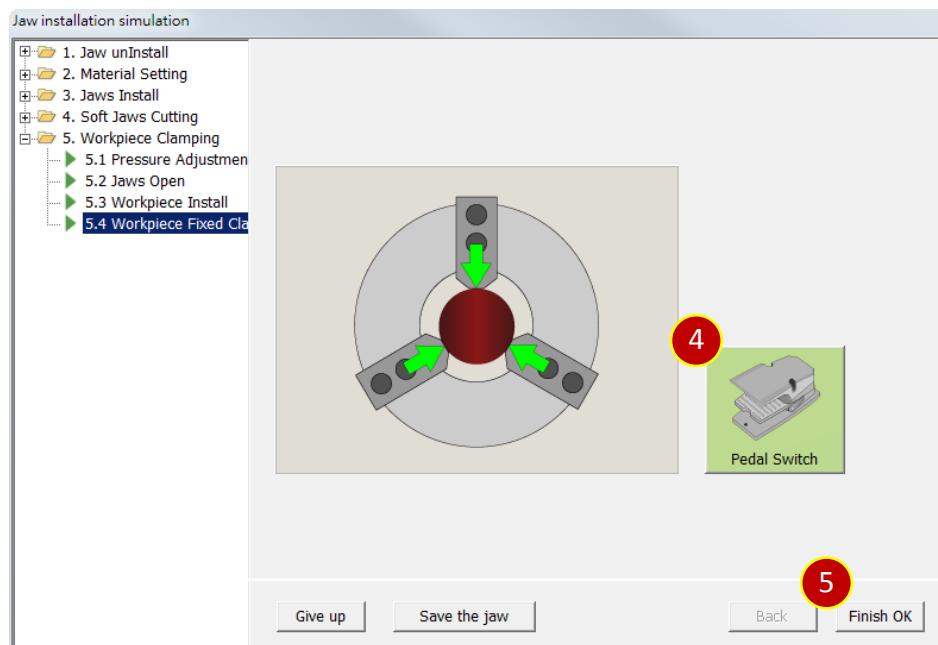
## 3.5.3 Workpiece Install

- (1) Press 【Pick Up】
- (2) Press 【-Z】 to install the workpiece on the Soft Jaws
- (3) Press 【Next】



## Manual

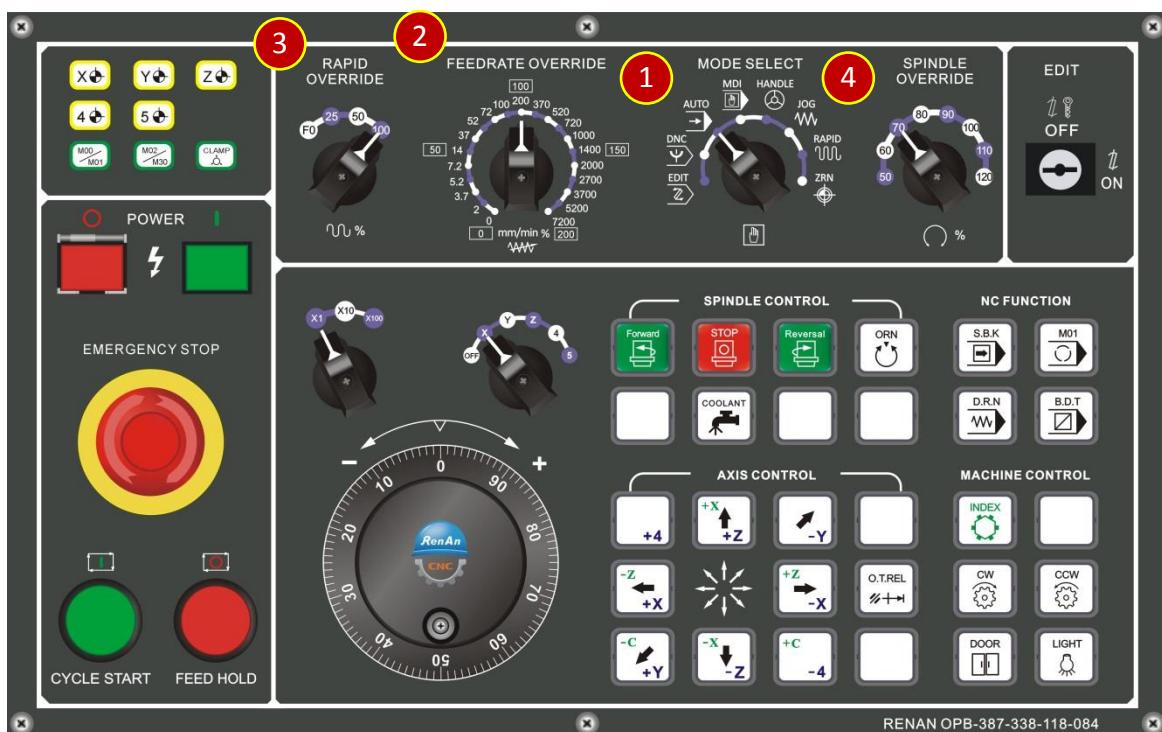
- (4) Press 【Pedal Switch】 to clamp the material tightly
- (5) Press 【Finish OK】 to finish the soft jaw installation

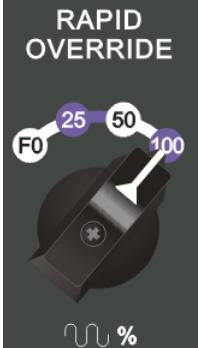


## 4. Controller Panel

### 4.1 Controller Panel Introduction

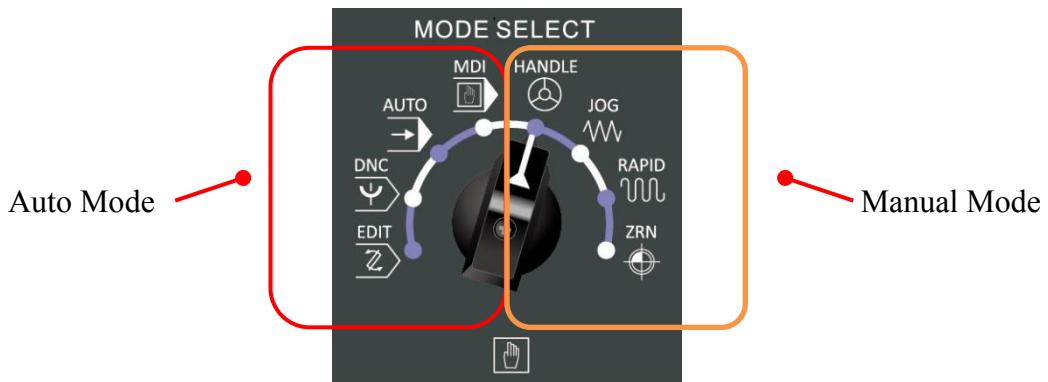
- (1) Mode Select Knob
- (2) Feedrate Override Knob
- (3) Rapid Override Knob
- (4) Spindle Override Knob



Knob	Name	Description
	MODE Select	Only the function that is included in the selected mode will be active
	FEEDRATE Override	To adjust the feedrate 0 - 200%
	RAPID Override	To adjust the feedrate in Rapid mode F0 (=150mm/Min) 25%, 50%, 100%

## 4.2 Machine Operation Mode Select Buttons

[Mode Select]: (1)Auto Mode (2)Manual Mode



(1) Auto Mode Description:

Mode	Name	Description
	EDIT	Available for edit and search the CNC program in the controller
	DNC	Load in CNC program from the external device
	AUTO	Execute CNC program in the controller
	MDI	Execute immediately after the CNC program was key-in to MDI window

(2) Manual Mode Description:

Mode	Name	Description
	HANDLE	Manually control the axial movement of the tool with Handwheel
	JOG	Manually control Continuous Feed with axis keys
	RAPID	Manually control Rapid Traverse of the bed with axis keys
	ZRN	Tool returns to the home position

## 4.3 Select Tool Manually

Manually select tool number, E.g.: No.7

(1) Select the [JOG] operating mode (or other mode in manual)

(2) Press 【INDEX】 , rotate clockwise to position No.7



RENAN OPB-387-338-118-084

## 4.4 Handwheel Feed Operation

When in HANDLE mode, move the tools manually via Handwheel

### 4.4.1 HANDLE Introduction

#### (1) Continuous Feed:

Continuously rotate the handwheel with the Handwheel holder

#### (2) Adjustment for accuracy:

Grabbing the side of handwheel scale ring and slowly rotate it



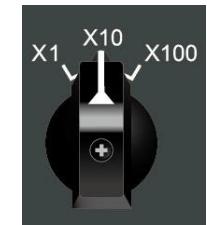
## Manual

## 4.4.2 [Axial Select] &amp; [Ratio Select] description

## (1) [Axial Select] description:

เลือกแกน	คำอธิบาย
Axial Select	Description
	Handwheel Movement using [X] axis .
	Handwheel Movement using [Z] axis

## (2) [Ratio Select] description:

เลือกอัตราส่วน	คำอธิบาย
Ratio Select	Description
	Set Handwheel Movement Ratio by [x1] Move 0.001mm per Scale (The basic unit of the controller length: 0.001mm)
	Set Handwheel Movement Ratio by [x10] Move 0.01mm per Scale
	Set Handwheel Movement Ratio by [x100] Move 0.1mm per block

## Manual

## 4.5 Common Buttons Description

## 4.5.1 Classification

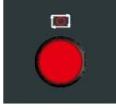
- (1) Auto execution
- (2) Operation in Manual (Axial movement, spindle control)
- (3) Other auxiliary buttons



## Manual

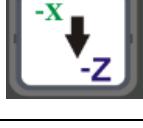
## 4.5.2 Buttons Description

## (1) Auto execute buttons description:

ปุ่ม Button	ชื่อ Name	คำอธิบาย Description
	Single Block	When the light on, execute block by block, then pause the program
	Optional Block Skip	When the button's light is illuminated, the information from the slash to immediately before the EOB code is ignored
	Optional Stop (M01)	When the button's light is illuminated, memory operation is stopped after a block containing M01 is executed.
	CYCLE START	Program starts to work automatically
	FEED HOLD	When Feed Hold button is pressed during memory operation, the tool decelerates to a stop
	Emergency Stop	Emergency stop button

## Manual

(2) Manual Control Buttons' description: (2) คำอธิบายปุ่มควบคุมแบบมือ

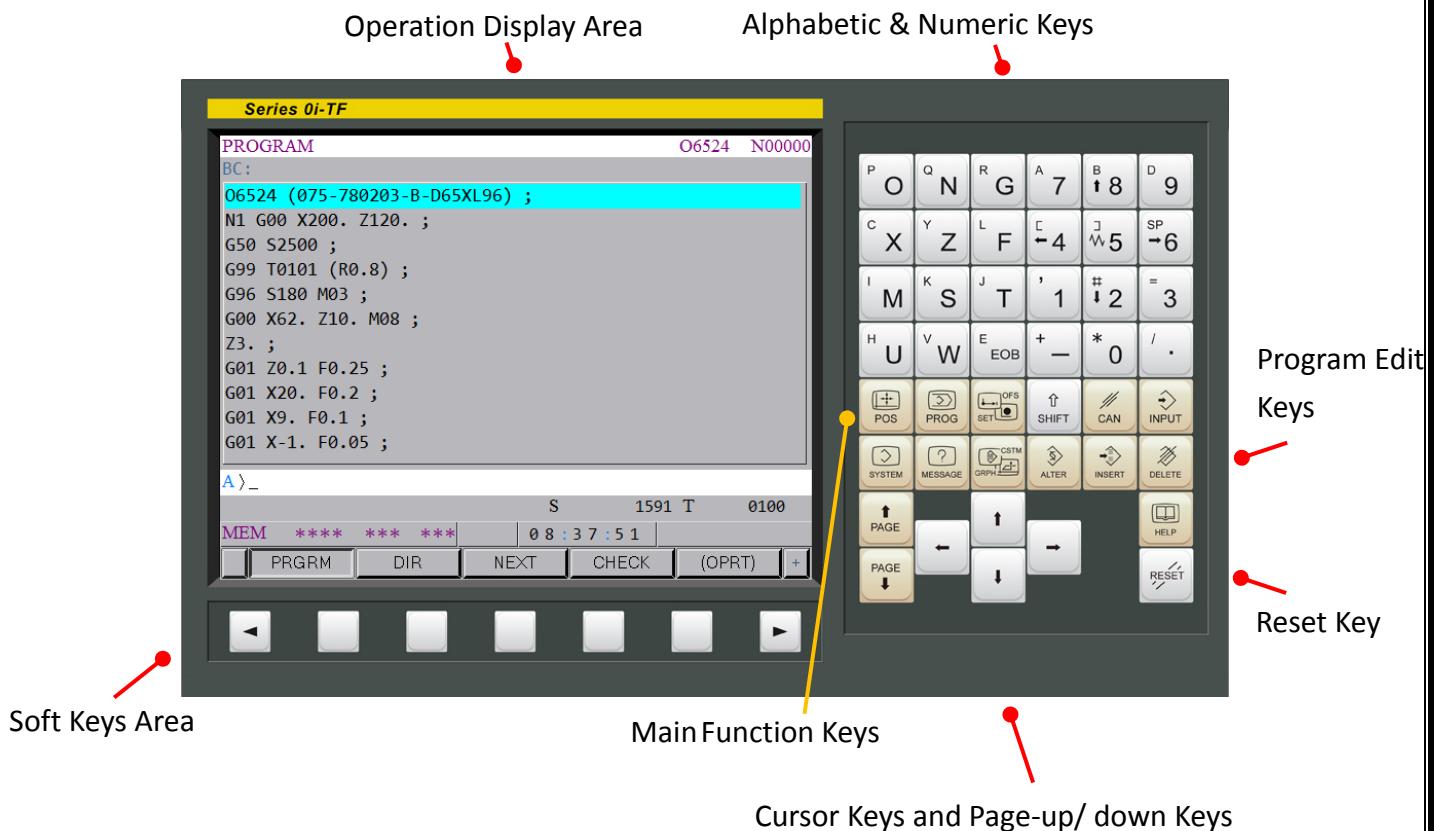
Button	Name	Description
	Spindle Forward	Spindle rotates forward (counter clockwise)(CCW)
	Spindle Stop	Spindle stops
	Spindle Reverse	Spindle rotates reversely (clockwise)(CW)
	+X	Turret & Tool Move in [+X] direction
	-X	Turret & Tool Move in [-X] direction
	+Z	Turret & Tool Move in [+Z] direction
	-Z	Turret & Tool Move in [-Z] direction

## (3) Other auxiliary buttons' description:

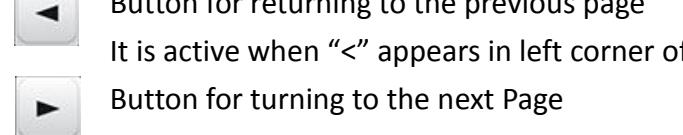
Button	Name	Description
	DOOR	Safety Doors switch
	O.T.REL	If collision happened between tools and the retraction of the tools was not available, press this button & use ZRN mode to retract the tool
	COOLANT	Coolant switch On/Off

## 5. Controller Panel Operation

### 5.1 Controller Panel Introduction



#### 5.1.1 Controller Panel Description

รายการ Item	คำอธิบาย Description
Operation Display Area	To display all kind of operation data enables users to monitor all kind of data
Soft Keys Area	 Button for returning to the previous page It is active when “<” appears in left corner of screen  Button for turning to the next Page It is active when “+” appears in right corner of screen (OPRT) Operation Key, to display operation options 

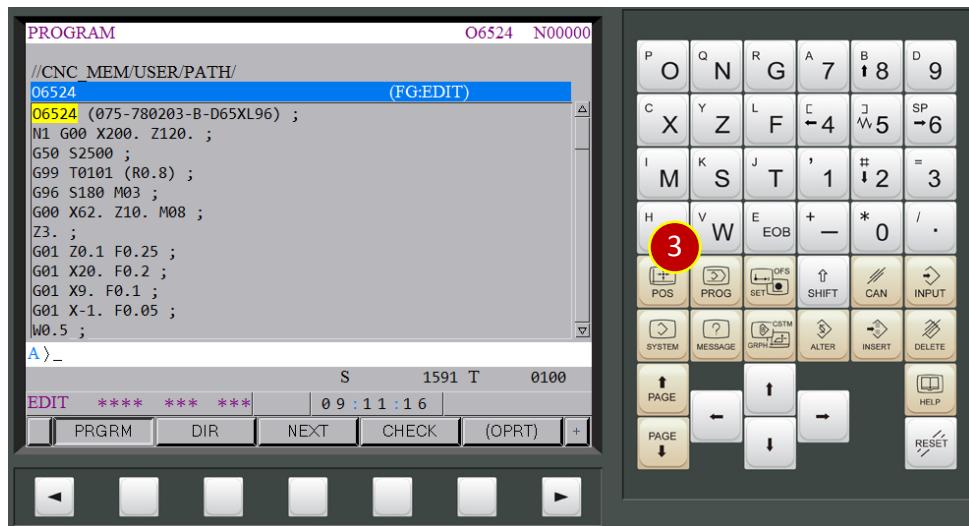
รายการ Item	คำอธิบาย Description
【RESET】 Key	<p>Use when reset data or NC status</p> <p>(1) Clear Alarm</p> <p>(2) Mandatorily pause the auto-run program</p> <p>(3) Move cursor to the beginning of the program in EDIT mode</p>
Main Function Keys	 Display Position Screen  Display Program Screen  Display Offset and Setting Screen  Display System Screen  Display Message/Alarm Screen
Program Edit Keys	 Alter, Change the Program Content  Insert, Add NC Program Content  Delete, Delete Program Content  Cancel (clear the key input buffer)
Alphabetic & Numeric Keys	<p>Key-in letters, numbers</p>  Press 【EOB】 to show “;” means the end of the single block
Cursor Keys and Page-up/ down Keys	Control the cursor

## Manual

## 5.2 Program Edit

## 5.2.1 Enter Program Edit Mode

- (1) Change the mode to 【EDIT】 mode
- (2) Change the program lock ③
- (3) Press 【PROG】 of Main Function Keys area,  
and it will change to program inspection window



## Manual

## 5.2.2 New NC Code and Comment

(1) New NC Code O0224, press 【O】 first and key in number “0224”

(2) Press 【INSERT】 Key, to add new program

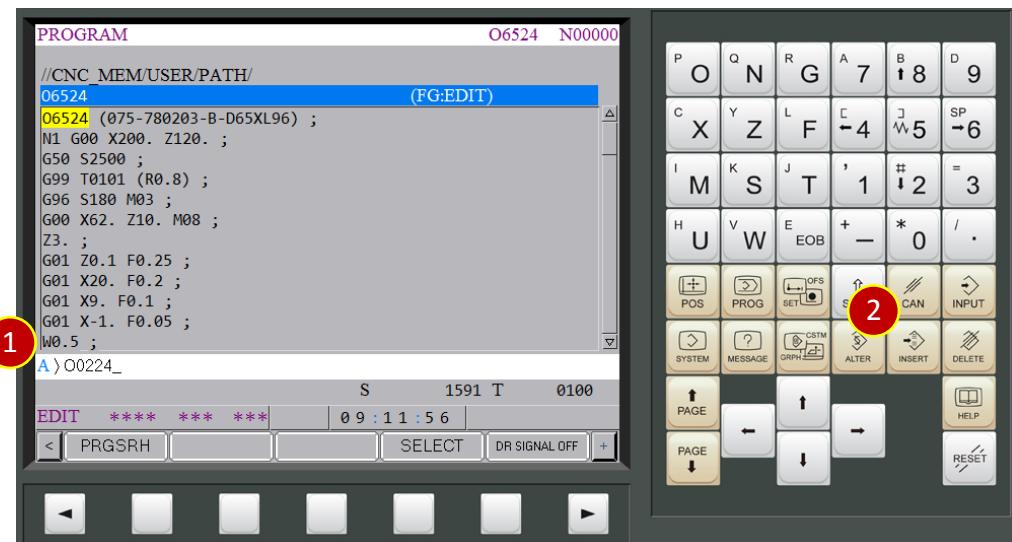


(3) Key in the comment, e.g. (075-780203-A) ;

Note: CNC controller would regard the words in parenthesis as comment and will not be executed.

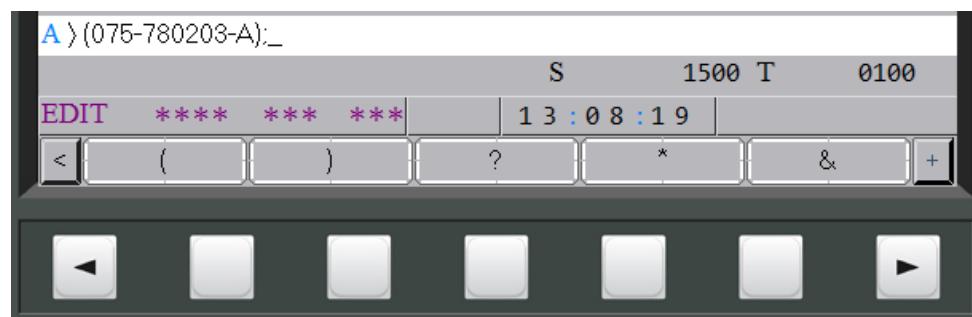
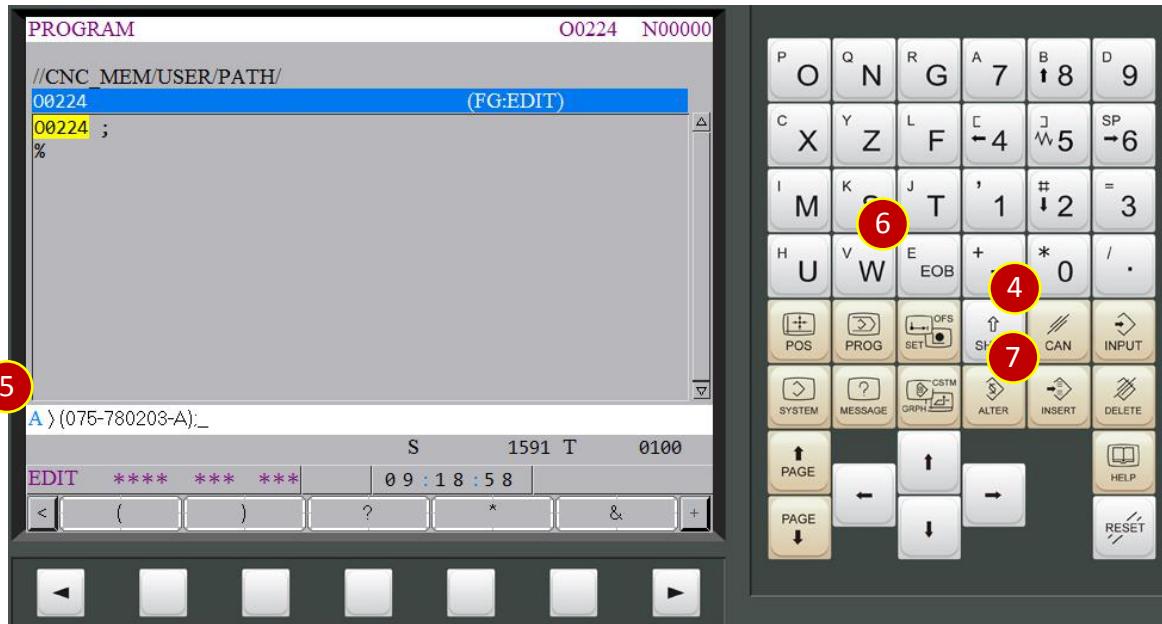
The parenthesis can be found in Operation Button Area

Press 【OPRT】 > Press 【➡】 (twice) > Press 【KEYINP】 to find the parenthesis



## Manual

- (4) Press 【CAN】 Key, to delete last character when the key input buffer displays
- (5) Key in comment e.g.(075-780203-A)
- (6) Press 【EOB】 and the symbol “;” will appear
- (7) Press 【INSERT】 to insert the comment



## Manual

## 5.2.4 Add New Program Content

- (1) Move cursor to key in the NC code in the program content

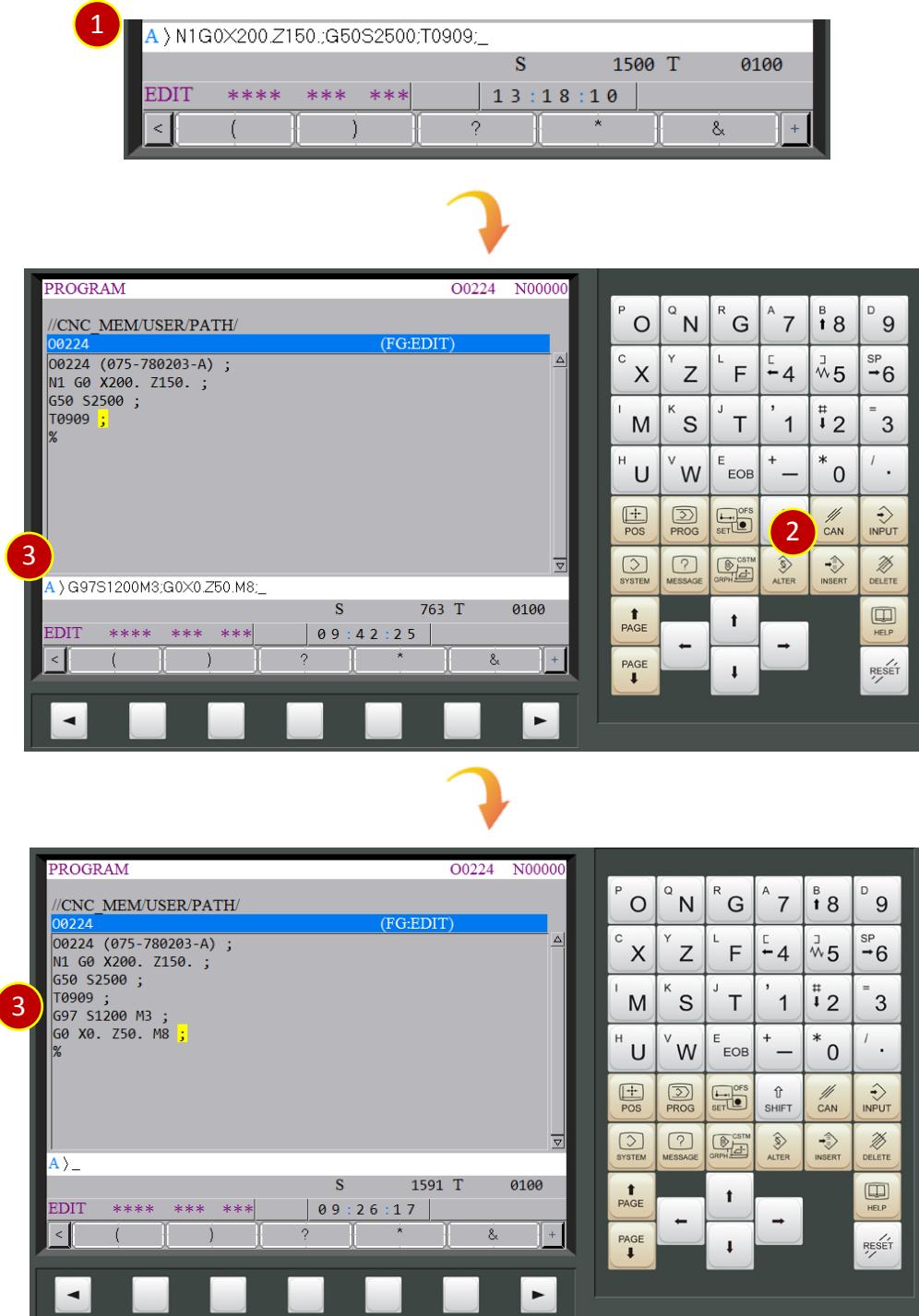
e.g. N1 G0 X200. Z150. ; G50 S2500 ;T0909 ;

- (2) Afte key in the NC code above, press 【INSERT】 to insert the NC code



- (3) Continue to key in the NC code

e.g. G97 S1200 M3 ; G0 X0. Z50. M8;



## Manual

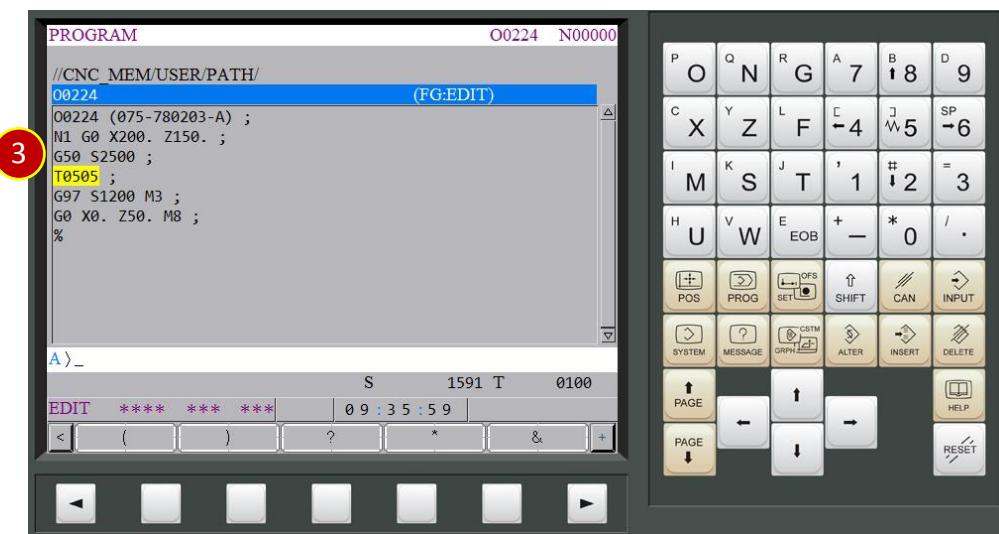
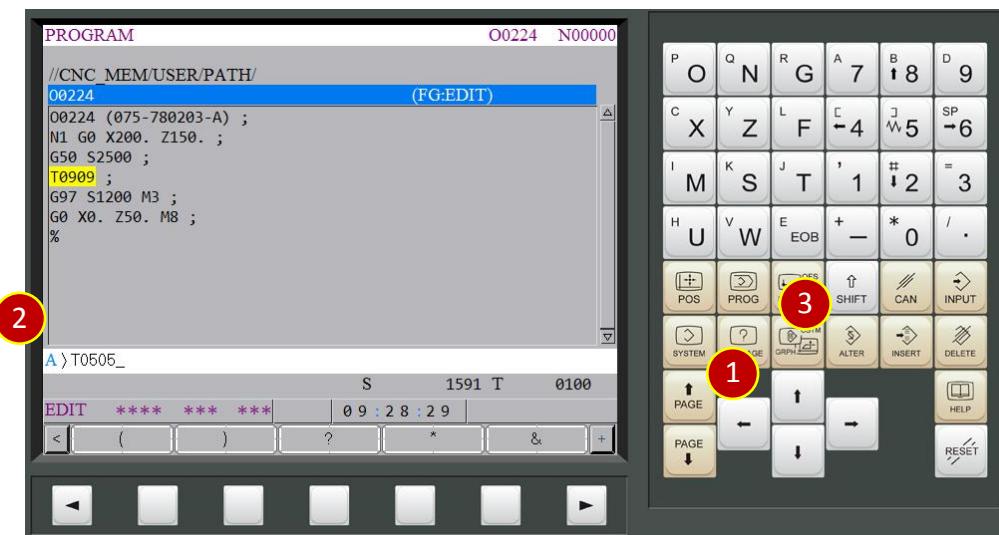
## 5.2.5 Change Program Content 【ALTER】

(1) When the program content need to be changed,  
move corsur to the change NC code position



(2) Key in new NC code “T0505”

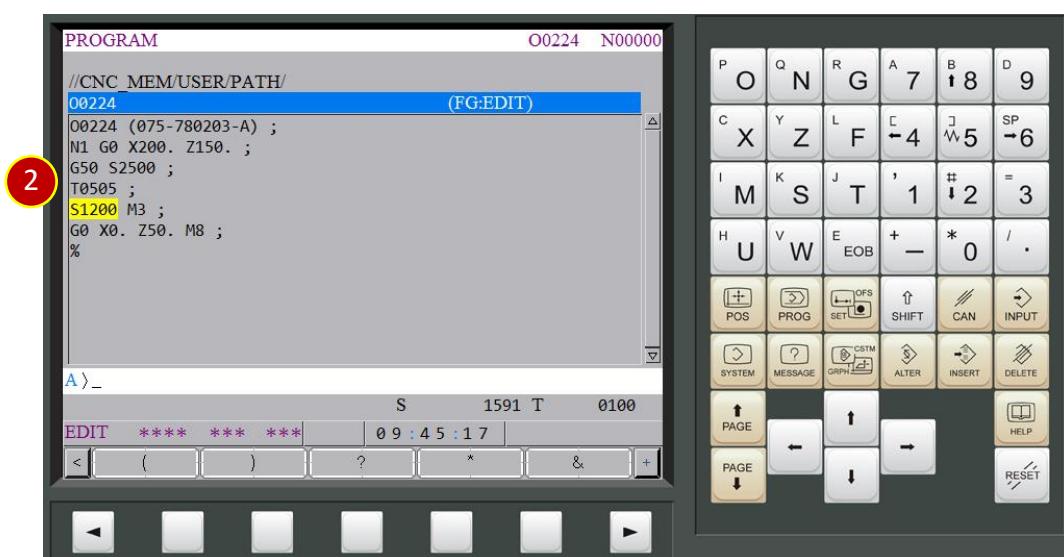
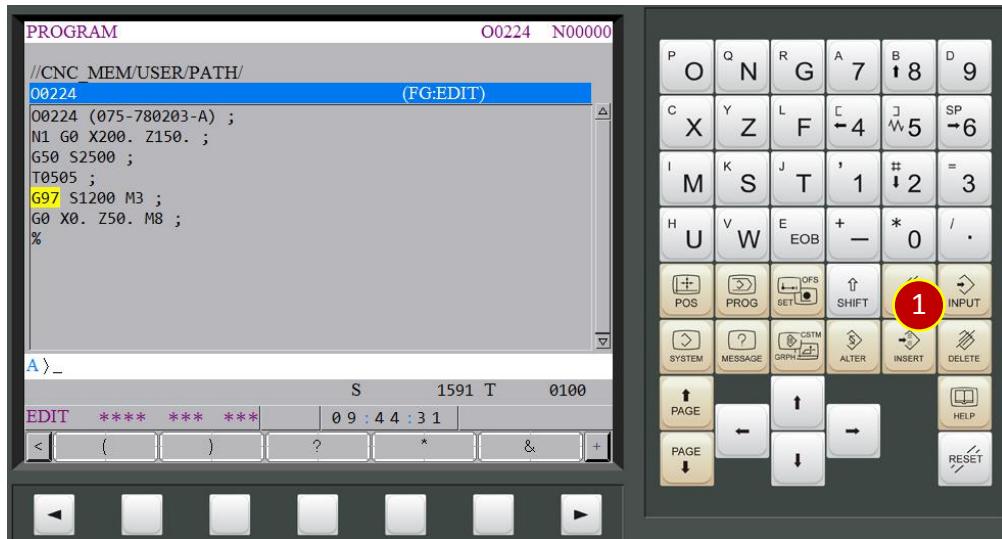
(3) Press 【ALTER】 program change key,  
the NC code will be replaced



## Manual

## 5.2.6 Delete Program Content 【DELETE】

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



## 6. Transmission Parameter Setting and Operation

We offer RS232 in the Training Machine that can connect with external device (such as PC). Transmit data through RS232



### 6.1 RS232 COM Setting

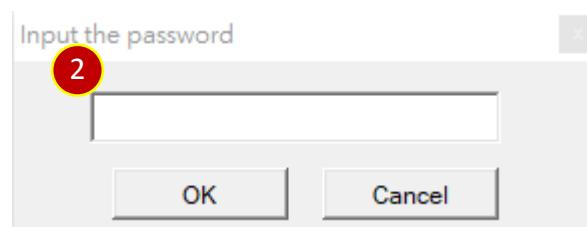
Assign the COM Port in Training Machine to the controller

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

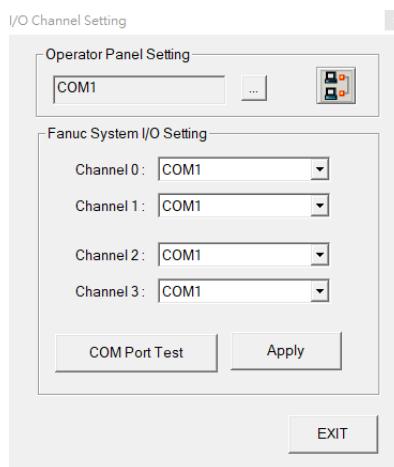
Please follow the steps below when the connection port need to be changed

#### 6.1.1 Open the setting interface of RS232

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



## 6.1.2 RS232 Connection Interface Setting



## 6.1.3 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

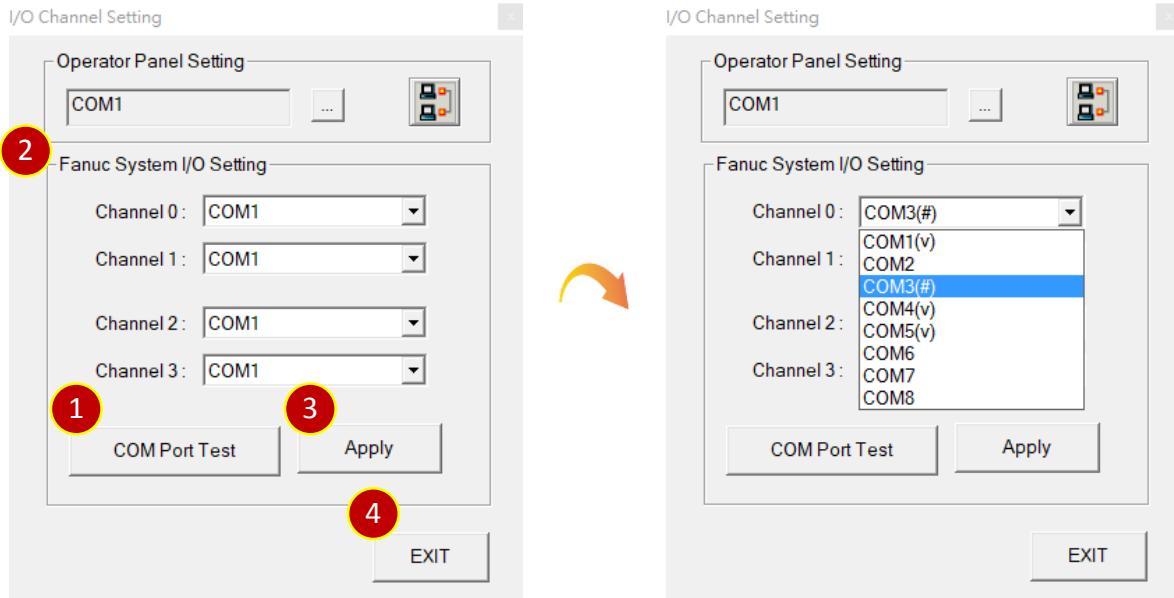
(2) Select the available and connected COM Port

e.g. CHANNEL 0, select 【COM3】

It means COM3 is used and is available for CHANNEL 0 to transmit

(3) Press 【Apply】

(4) Press 【Exit】



## 6.2 Transmission Parameter Setting

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

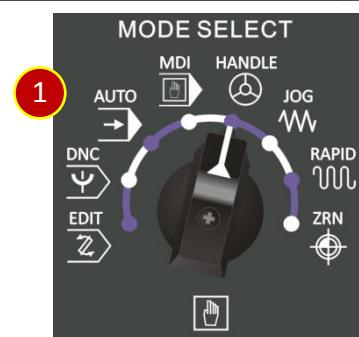
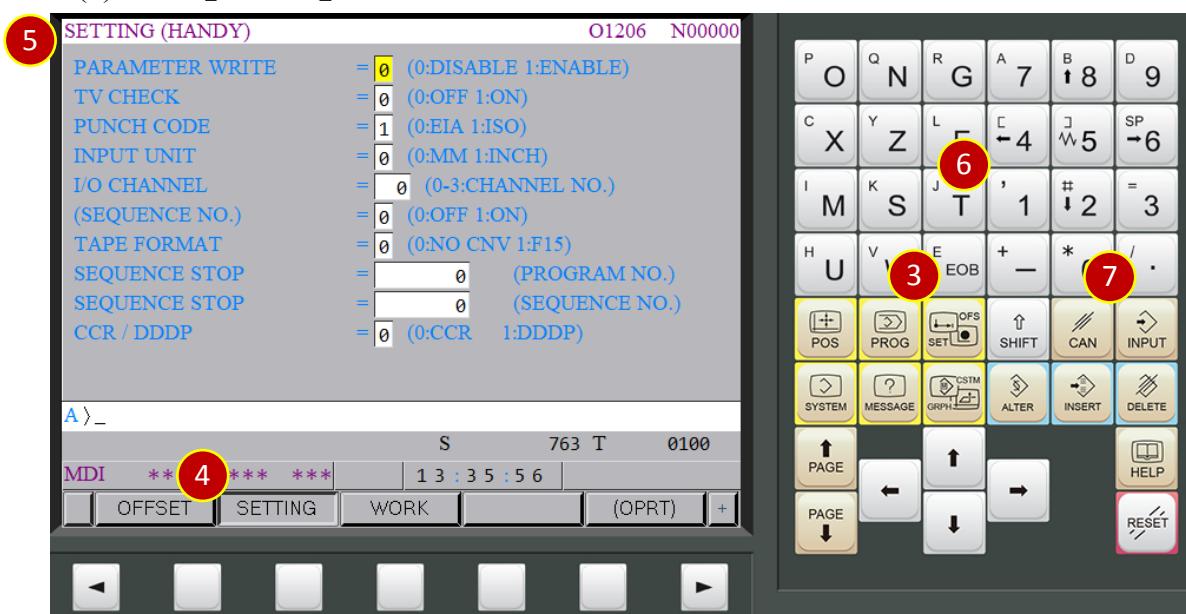
Code Type = ISO	Parity Check = None
Baud Rate = 4800	Stop Bit = 1
Data bits = 8	

### 6.2.1 Cancellation of Parameter Input Protection

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

Steps:

- (1) Change the Mode Switch to 【MDI】 mode
- (2) Switch the 【Program Edit Lock】 to the edition status
- (3) Press 【OFS/SET】 to display parameter setting interface
- (4) Press 【SETTING】
- (5) Select 【PARAMETER WRITE】
- (6) Press 【1】
- (7) Press 【INPUT】



## Manual

## 6.2.2 Parameter Modification

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

(1) Press 【OFS/SET】 , show the interface of parameter setting

(2) Assign PUNCH CODE as a transmission code type

e.g. PUNCH CODE= 1 (0: EIA, 1: ISO)

(3) “I/O CHANNEL”, assign the channel when transmitting

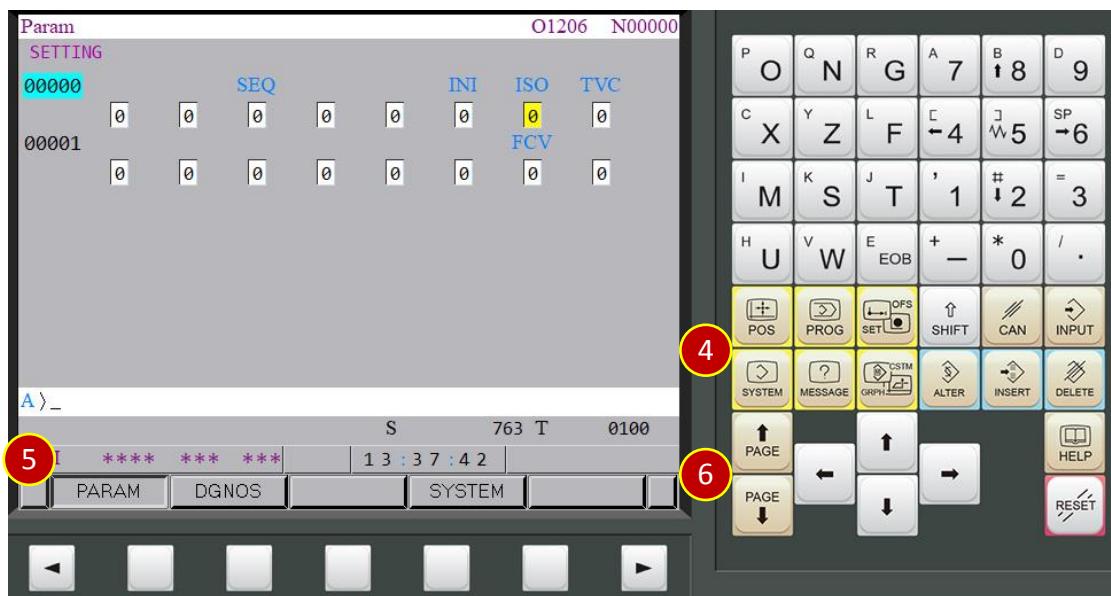
e.g. I/O CHANNEL= 0 (0: CHANNEL 0)



(4) Press 【SYSTEM】 , show the system parameter

(5) Press 【PARAM】

(6) Press 【 PAGE ↓ 】 , enter the setting page of CHANNEL 0

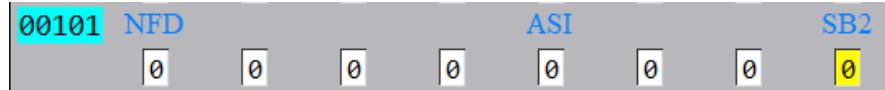


# TM-Fanuc-0i-TF Manual

(7) Set the transmission parameter of CHANNEL 0

00101 NFD = 0, ASI = 0, SB2 = 0

(The description of transmission parameters, please see appendix 1)



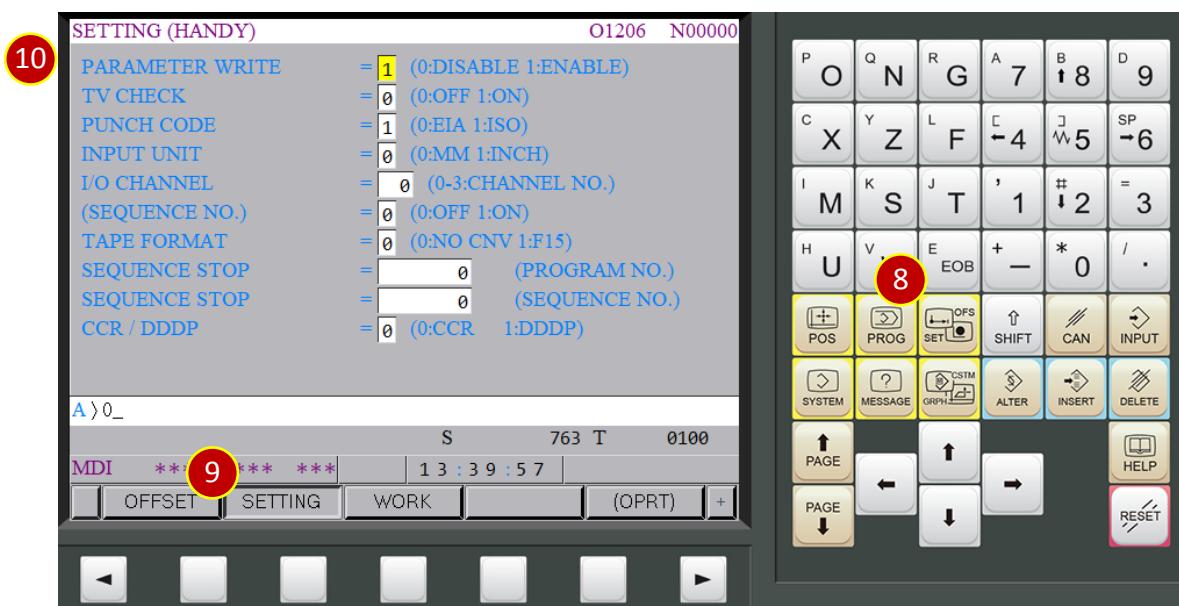
00103 BAUD RATE = 10, Transmission of CHANNEL 0 baud rate



(8) Press **[OFS/SET]** after finish the parameter setting

(9) Press [SETTING]

(10) Modify PARAMETER WRITE to **[0]**



## 6.3 Receive Program from the Controller

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

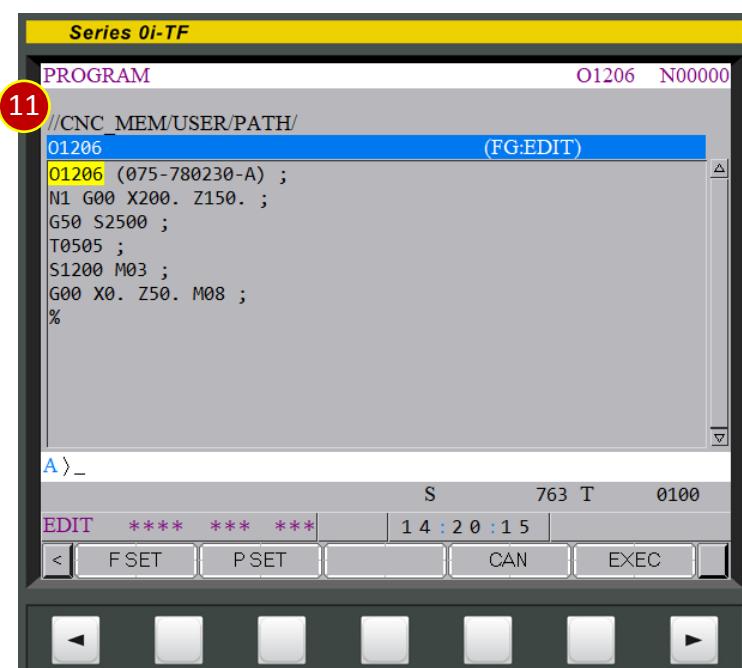
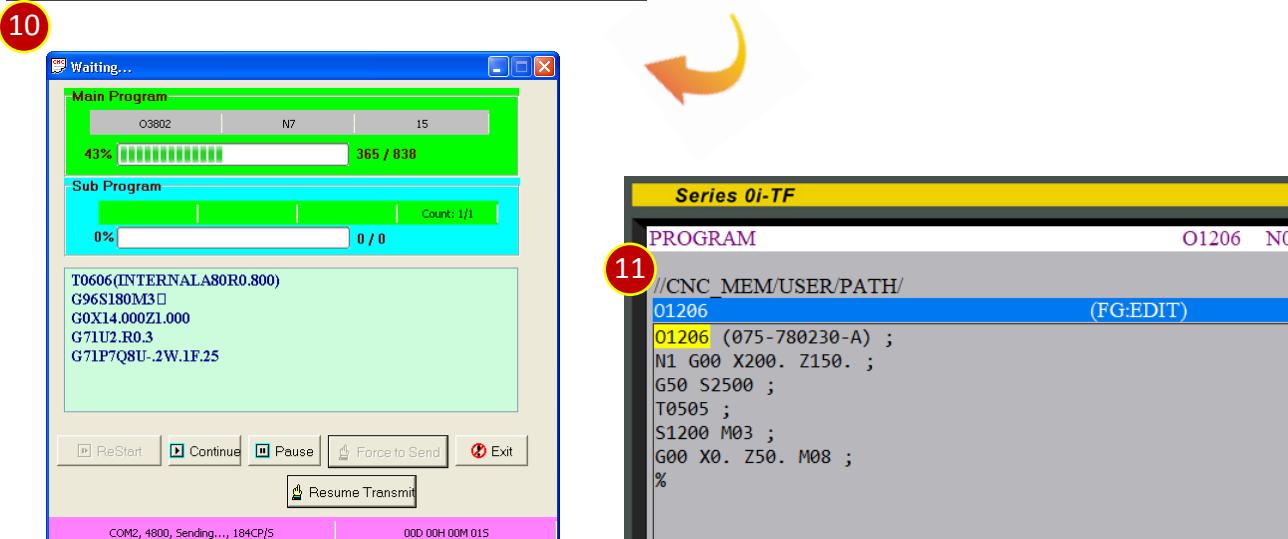
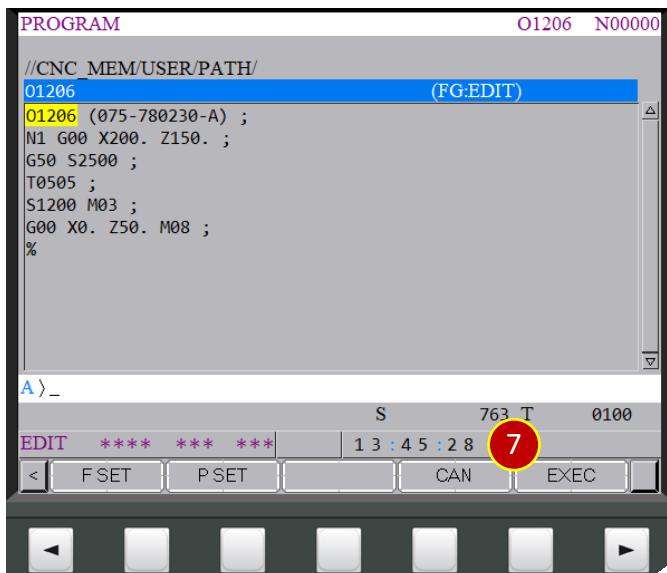
### 6.3.1 Operation Steps

- (1) Open up NcEditor to wait for the transmission
- (2) Change the Mode Switch to 【EDIT】 mode
- (3) Switch the 【Program Edit Lock】 to the edition status Press 【PROG】
- (4) Press 【(OPRT)】
- (5) Press 【▶】 button 4 times, the menu displays 【F Input】 and 【F Output】
- (6) Press 【F Input】 , receive the program



## Manual

- (7) Press 【EXEC】 , receive the program and NcEditor will send the program by auto notification
- (8) “LSK” means to wait for receiving the program
- (9) “INPUT” means receiving the program
- (10) NcEditor will show the transmission process
- (11) Auto-show the program while finish receiving

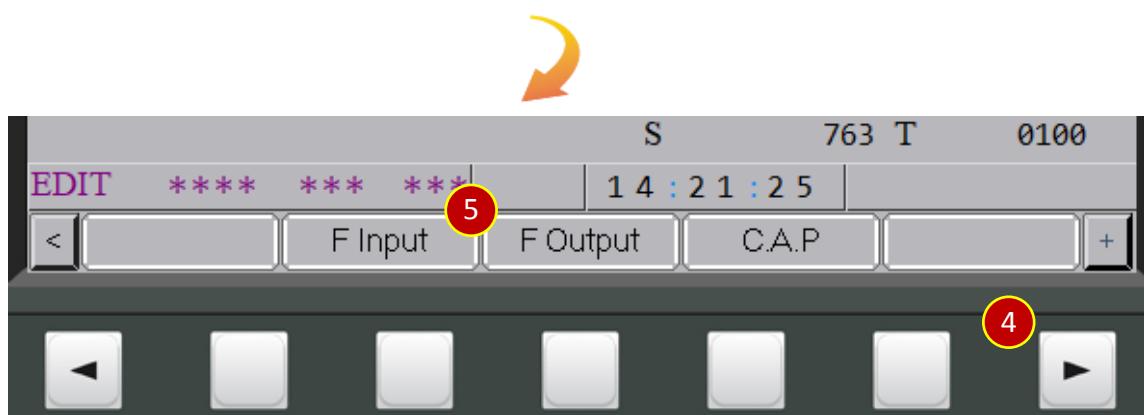
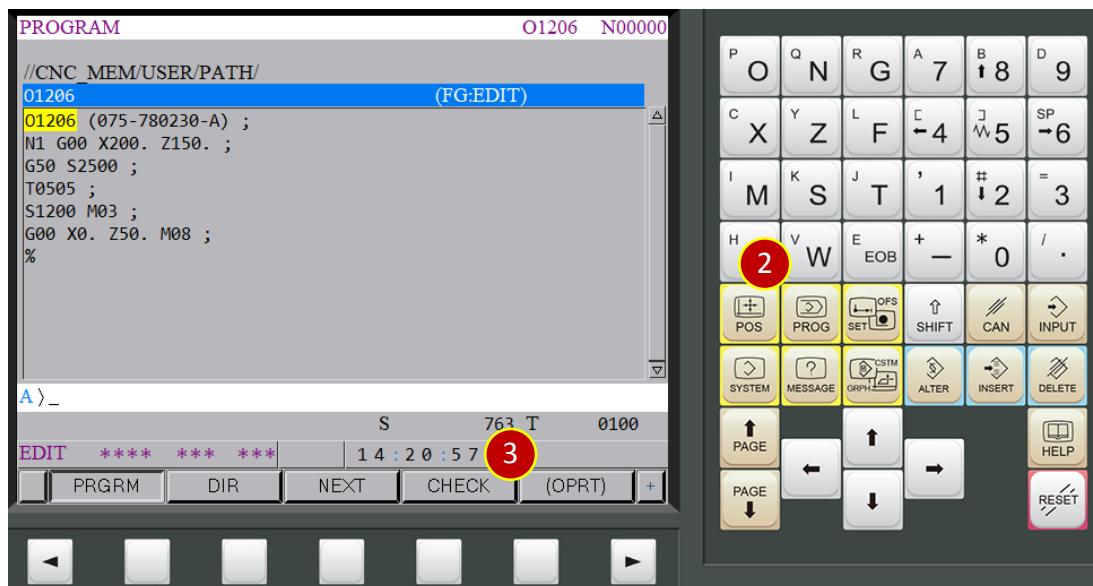


## 6.4 Send Program from the Controller

Send the program in the controller to PC

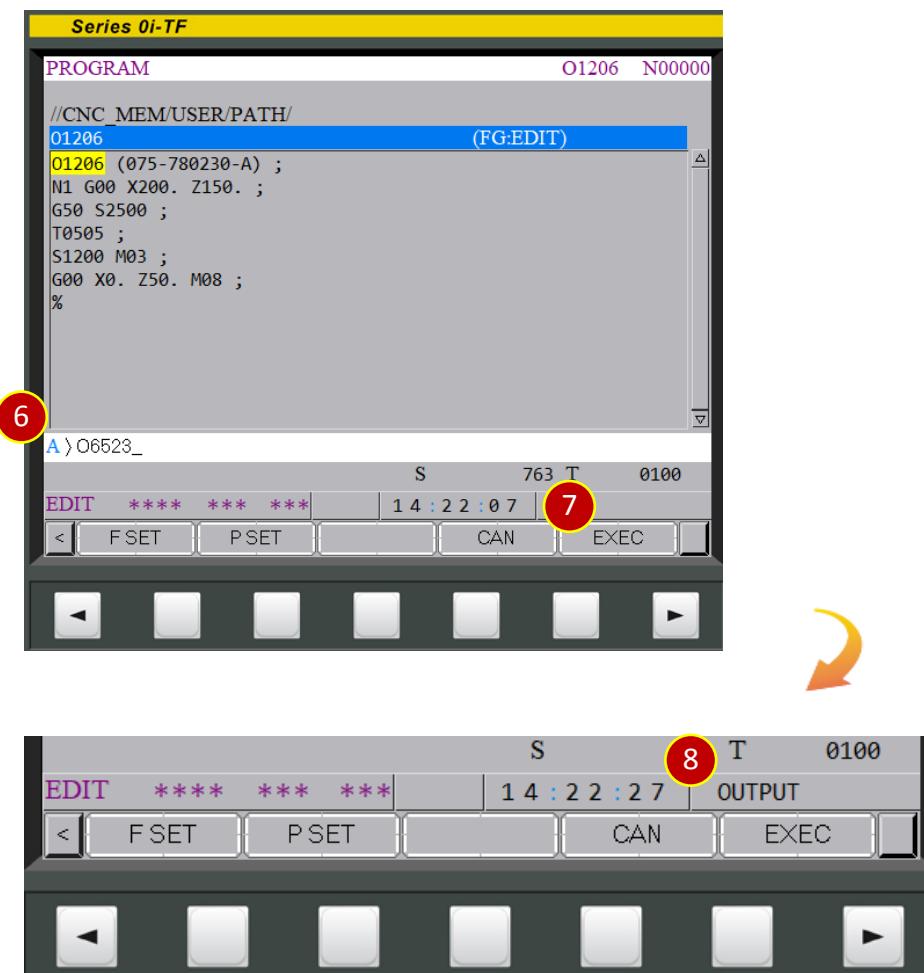
### 6.4.1 Operation Steps

- (1) Change the Mode Switch to 【EDIT】 mode
- (2) Press 【PROG】
- (3) Press 【(OPRT)】
- (4) Press 【▶】 button 4 times, the menu displays 【F Input】 and 【F Output】
- (5) Press 【F Output】



## Manual

- (6) Key in the program O code to be uploaded, e.g.: O6523
- (7) Press 【EXEC】
- (8) “OUTPUT” shows up,  
it means that the program is sending and NcEditor is receiving
- (9) NcEditor shows the program content

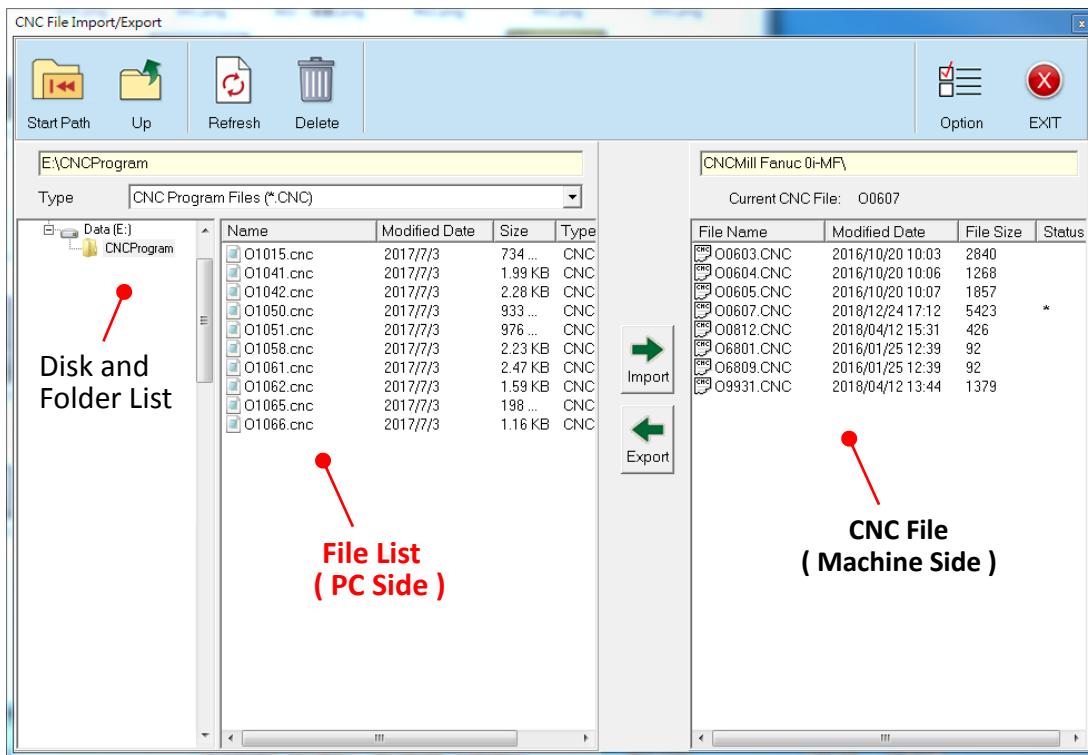


## 6.5 CNC File Import/Export Management

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



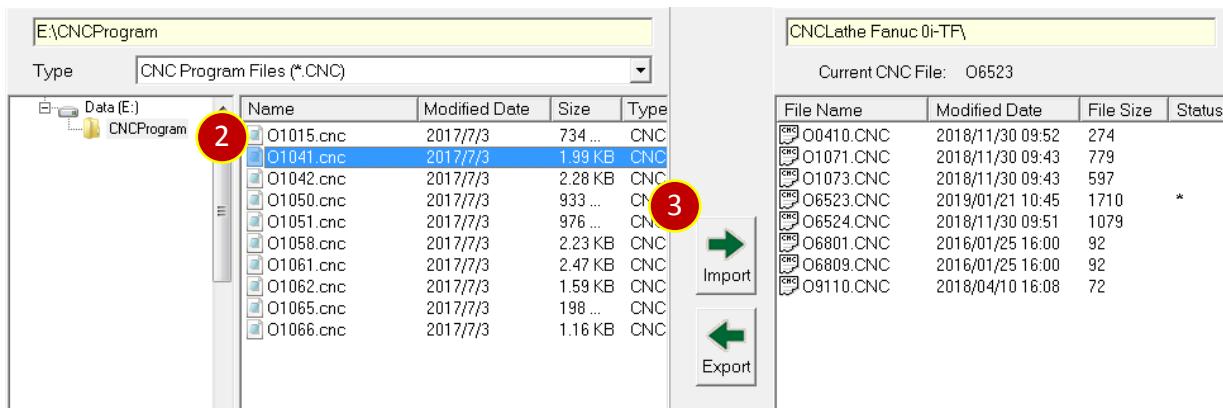
## 6.5.1 Open CNC File Import/ Export Function



Button	Name	Description
	Start Path	Back to Specify Folder
	Up	Return to previous folder
	Refresh	Refresh the file list
	Delete	Delete the selected file
	Option	Set options
	Exit	Exit the file import/export window
	Import	Import the files to Machine Side from PC side
	Export	Export the files to the PC side from the Machine side

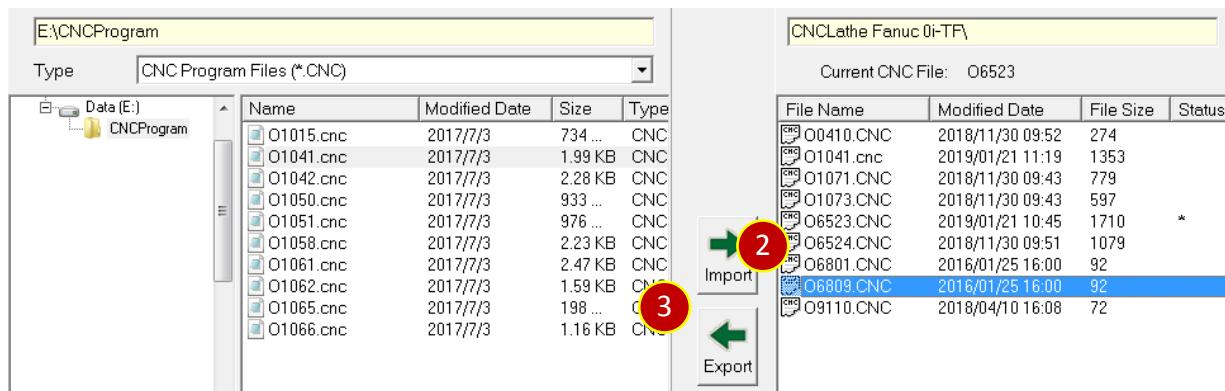
## 6.5.2 Import CNC File

- (1) Press 【CNC File Import/ Export】
- (2) Choose the CNC file from PC side, e.g. O1041.CNC
- (3) Press 【 ➔ 】 to import CNC file to machine side



### 6.5.3 Export CNC File

- (1) Press 【CNC File Import/Export】
- (2) Select the exporting CNC file from the menu of left side  
e.g. O6809.CNC
- (3) Press 【←】 to export the CNC file



## 7. Machine Basic Operation

### 7.1 ZRN Operation

Please do return each axis to the home position in order to establish a reference position

#### 7.1.1 Procedure

- (1) Press 【POS】 to display the coordinate function
- (2) Press 【ALL】 to display all of the coordinate value
- (3) Interpretation of different Coordinates Systems:

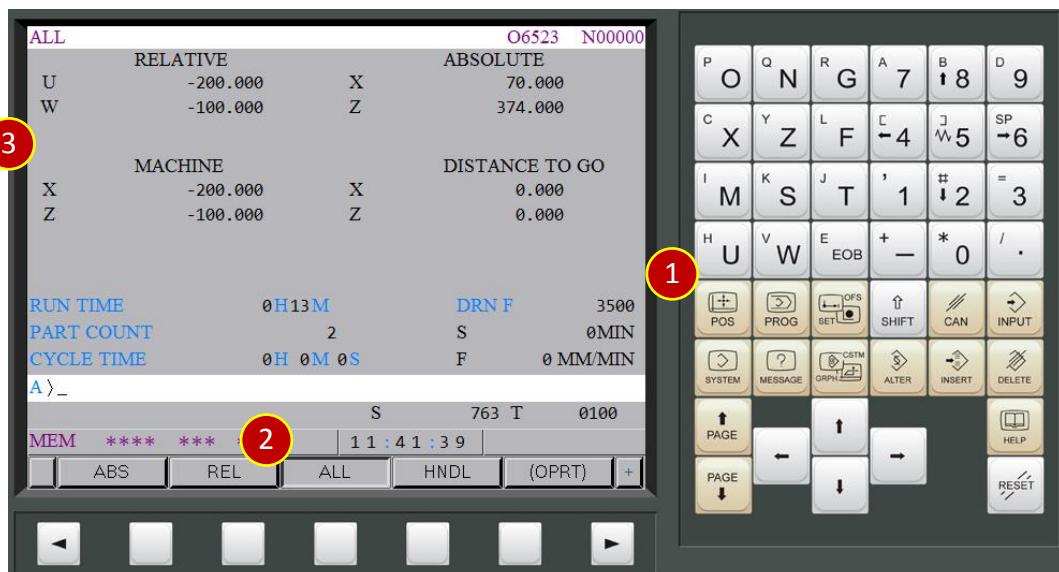


**RELATIVE:** To support the presetting of the coordinate to specified values

**ABSOLUTE:** Current position in the workpiece coordinate system

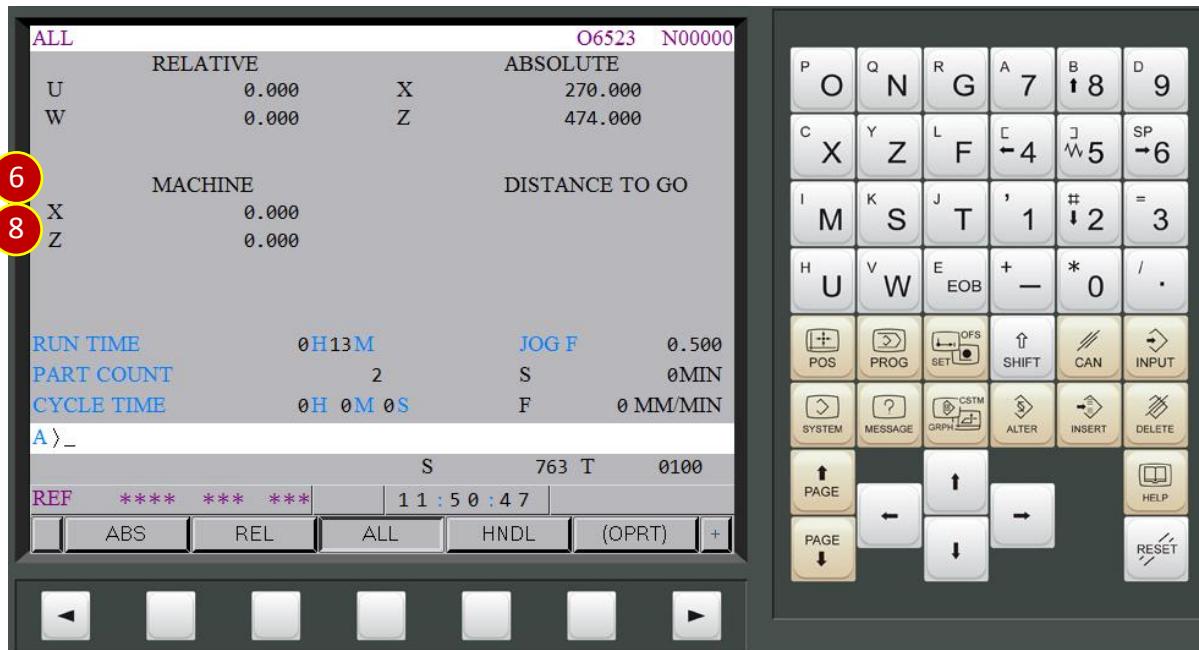
**MACHINE:** Current position in the machine coordinate system

**DISTANCE TO GO:** The distance the tool is yet to be moved in the current block



## Manual

- (4) Switch to 【ZRN】 mode (Zero Return)
- (5) Press [+X], X axis starts to move toward reference point rapidly
- (6) When the Machine coordinate of X axis shows 0, the light of the X axis zero point is illuminated, it means X axis ZRN is completed
- (7) Press [+Z], Z axis starts to move toward reference point,
- (8) When the Machine coordinate of Z axis shows 0, the light of the Z axis zero point is illuminated, it means Z axis ZRN is completed

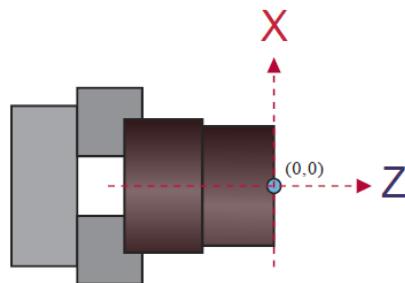


## 7.2 Work Coordinates Setting (OFFSET)

When replacing the workpiece, preset the work coordinate system (X & Z axis) according to the workpiece's dimension.

After the installation, tool geometry offset is necessary so that the cutting process will be executed correctly.

The program zero point in workpiece coordinate system usually sets in the right side of the workpiece center

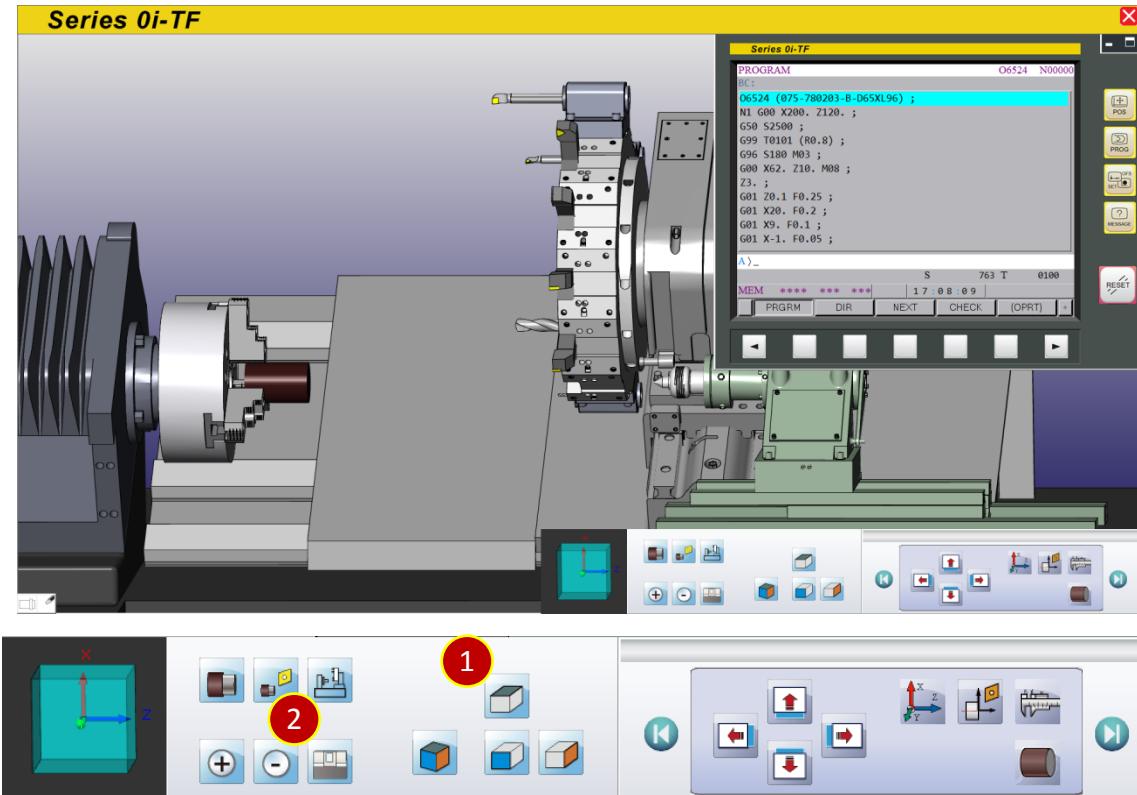


### 7.2.1 View Adjustment

Adjust the view to the proper angle and ratio for offset

(1) Press 【Top View】 via toolbar

(2) Press 【Hide Case】 to hide the shell of machine



### 7.2.2 Work Shift Coordinate Setting (Master Tool)

The definition of Work Shift:

- (1) The shifted distance between zero point and reference position is called Work Shift Value
- (2) By aligning the Master Tool, the nose will be shown as a shift reference point for machine origin, hence become the reference point for the Turret tool
- (3) When the turret returned to reference position, Work shift value will be specified as an absolute coordinate of the program. Hence, machine will run the program and do all kind of cutting and positioning based on this position

Reference Point Definition Example:

- (1) X Axis: Make “center line of the Inner Tool Holder” as a reference position
- (2) Z Axis: Make the nose of the external-rough tool as a reference position

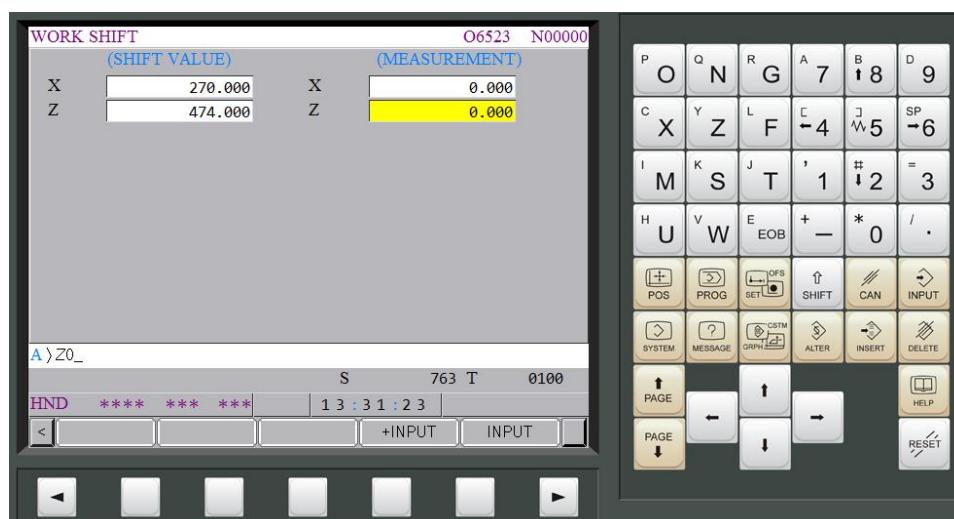
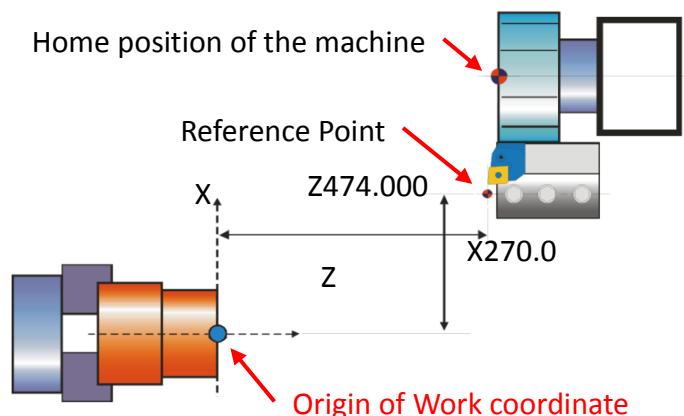
Advantages:

- (1) Set the central hole line of internal holder as the reference point of X axis
  - (a) The value of the distance between the center position of internal holder and the center position of spindle is fixed. It won't be changed by workpiece or tool. Therefore, input the value once and the X value will be permanently fixed
  - (b) The fixed X axis geometry Offset value of drilling, tapping, milling is 0. Offset here is not necessary
  - (c) The X axis geometry Offset value of internal tool is equal to the minimum diameter for cutting of internal cutting which can be much easier to identify the precision of the X axis offset value and to assure its readability

## Manual

(2) Set the tool nose of Roughing insert as the reference position of Z axis

- (a) Roughing insert often exists on the turret  
and can easily cut the casting face of the material
- (b) After offsetting,  
the geometry value Z of each tool is approximate to its elongation value

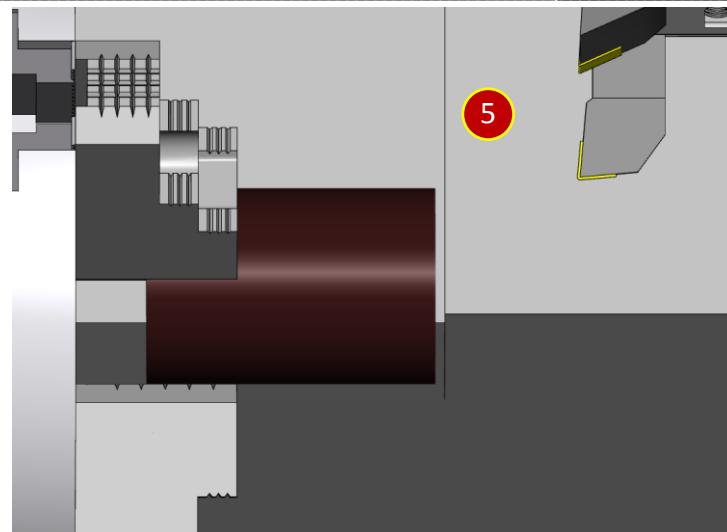


# Manual

## 7.2.3 Work Shift Coordinate Setting (Base Plane Offset)

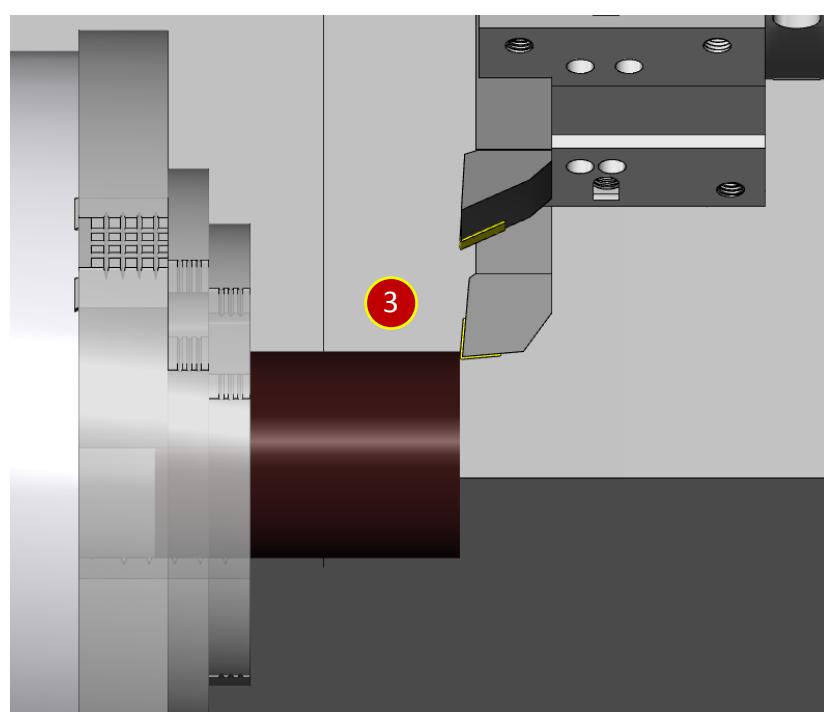
It is necessary to cut a shared base plane before setting the Work shift and offsetting

- (1) Switch to 【RAPID】 mode
- (2) Before manual operating, switch 【RAPID OVERRIDE】 to 50% (or 25%),  
100% rapid movement is not allowed under machining execution
- (3) Move the holder to the Safe Tool Change position  
(about 150mm left to the Workpiece)
- (4) Select External Rough Tool, take No.1 tool (default) as example, change to No.1 tool
- (5) Press【-X】,【-Z】to rapid traverse the tool to the position about 50mm left to the Workpiece



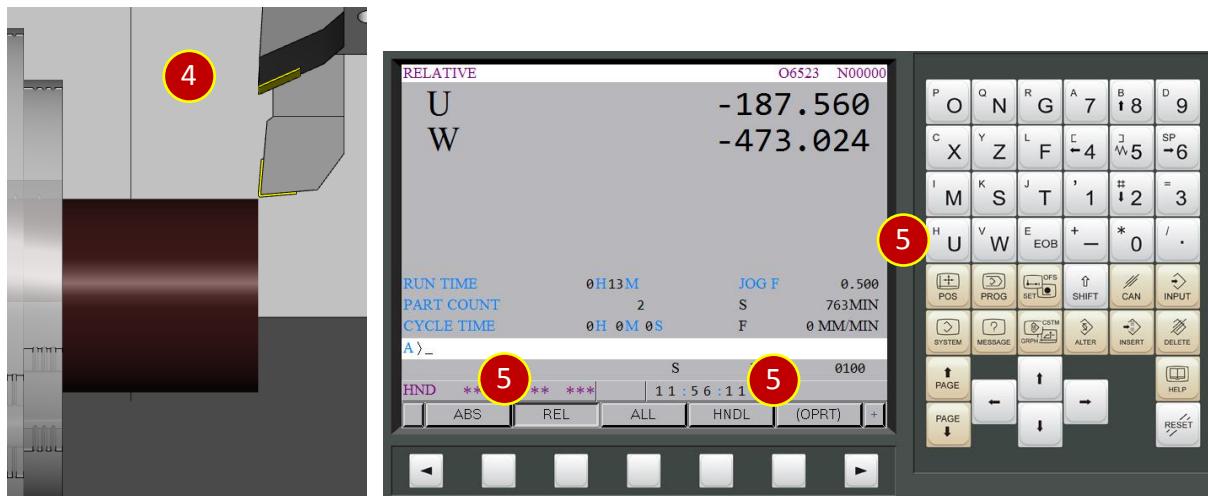
## 7.2.4 Cutting the Reference Face

- (1) Switch to 【MPG】 mode
- (2) Press 【Forward】 to rotate the spindle forward
- (3) Move the tool to the position about 3mm left to external diameter of Workpiece and slightly attach to it  
(till chips appear)

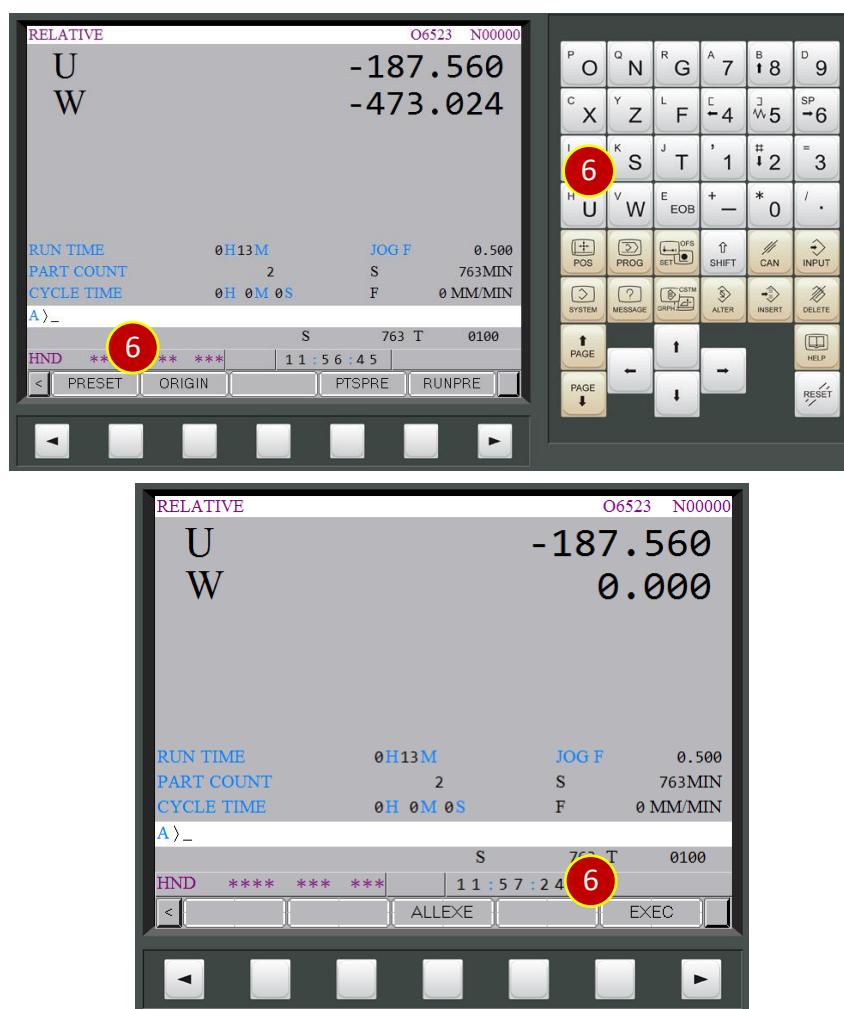


## Manual

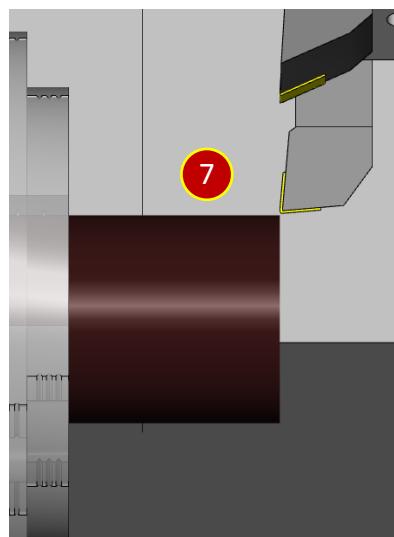
- (4) Move the tool upward to the External diameter of the Workpiece (+X direction)  
 (5) Press 【POS】 to display the coordinate position,  
 press 【REL】 to open the relative coordinate page, then press 【(OPRT)】



- (6) Zero the W coordinate, press 【W】 > 【ORIGIN】 > 【EXEC】 ,  
 W coordinate will be zeroed

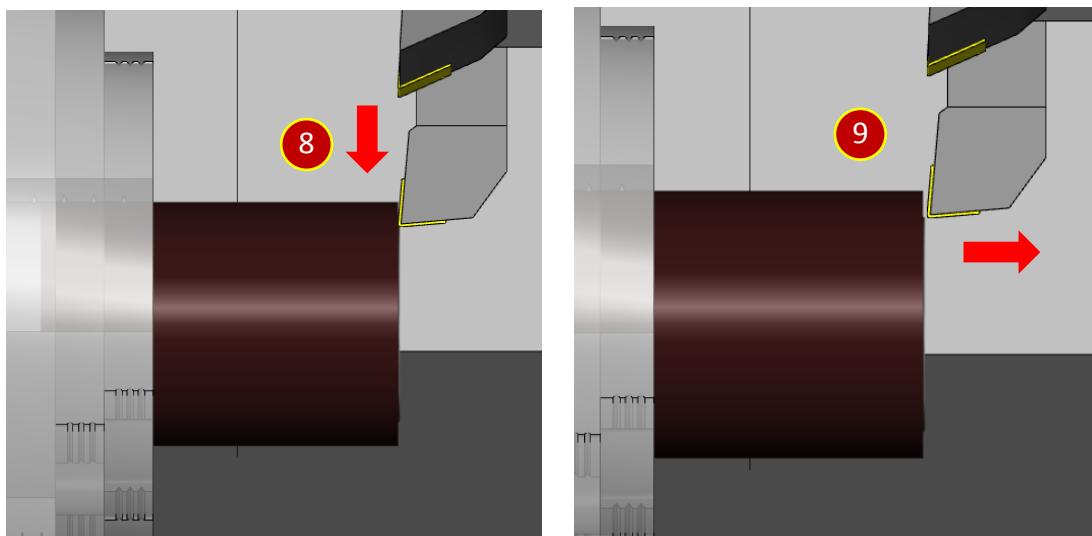


(7) Move the tool leftward W-0.5( Cut End Face 0.5mm deeper)



(8) Cut downward (in the -X direction) for about 15mm

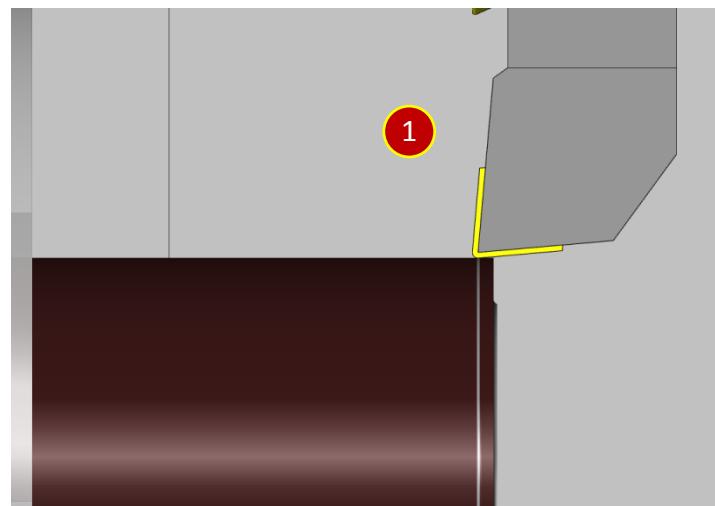
(9) Move right (in the +Z direction) to leave the workpiece



### 7.2.5 Cut the Reference External Diameter

(1) Move the tool over of the Workpiece, where is about 3mm left to the end face.

Slightly attached to the external diameter (till chips and cuts appear)



(2) Move the tool rightward (in the +Z direction) to the Workpiece end face.

(3) Press 【POS】 to display the coordinate position,

press 【REL】 to open the relative coordinate page.

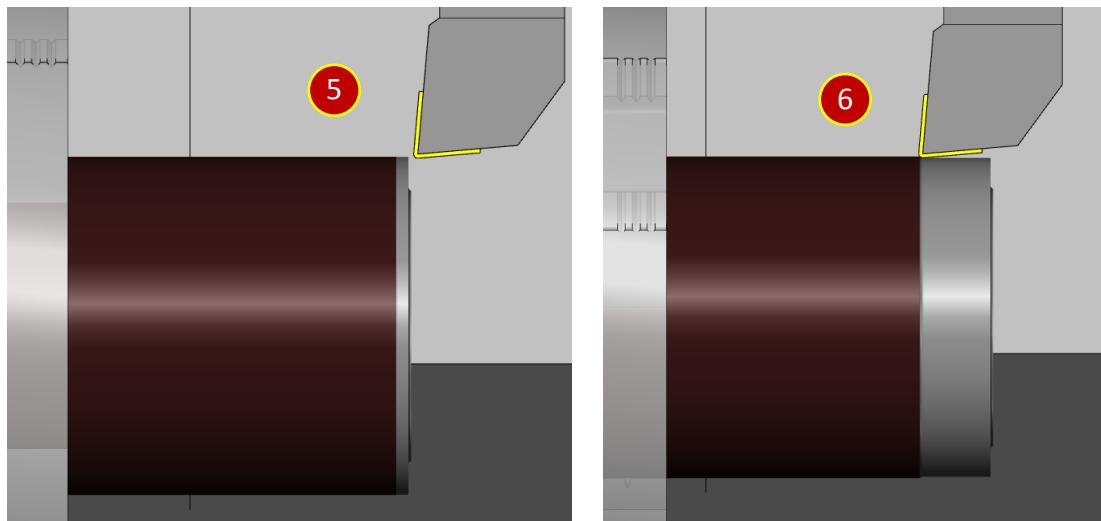
(4) Zero the U coordinate, press 【(OPRT)】 > 【ORIGIN】 < 【EXEC】 ,  
the U coordinate will be zeroed



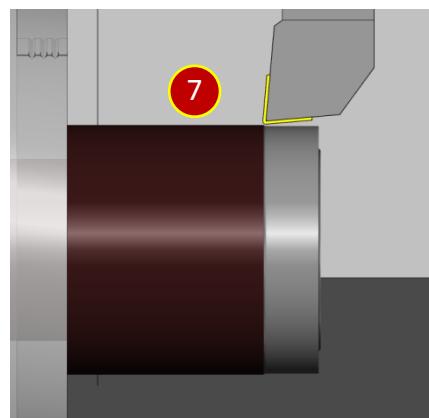
## Manual

(5) Move the tool downward to the X axis U-0.5 (Cut external diameter 0.5mm deeper)

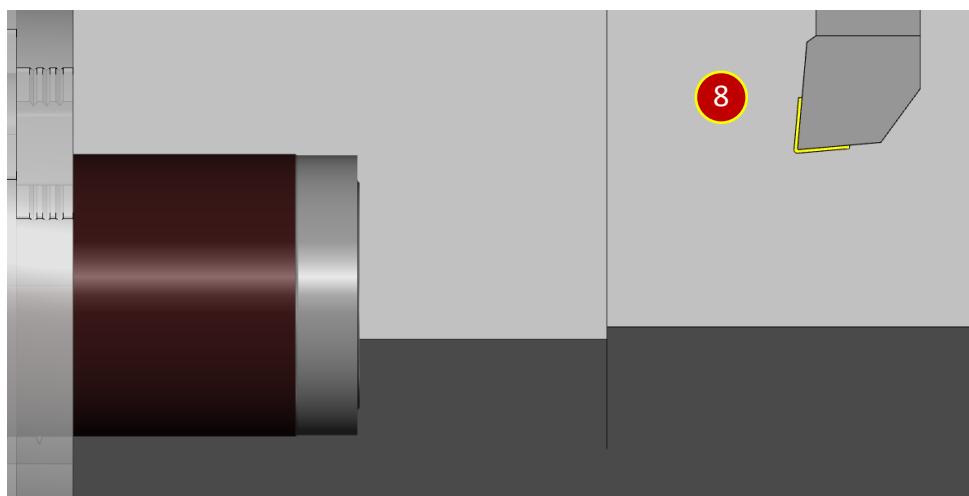
(6) Move leftward in (-Z) direction to cut for about 15mm



(7) Retract the tool in (+X) direction for about 1 mm



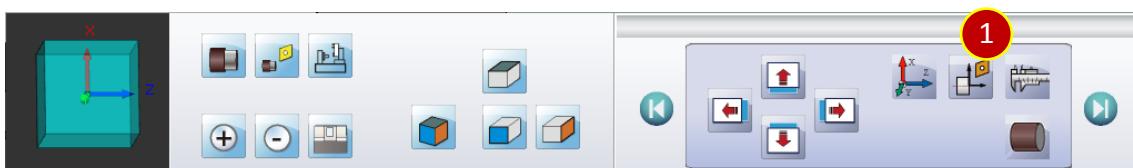
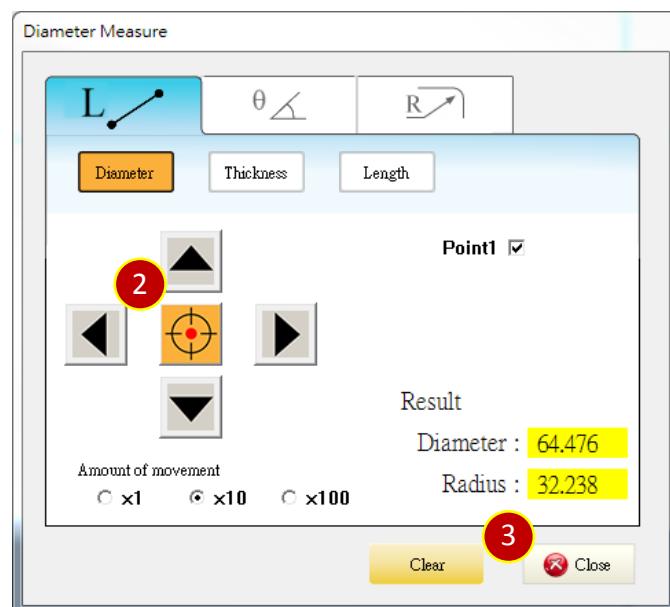
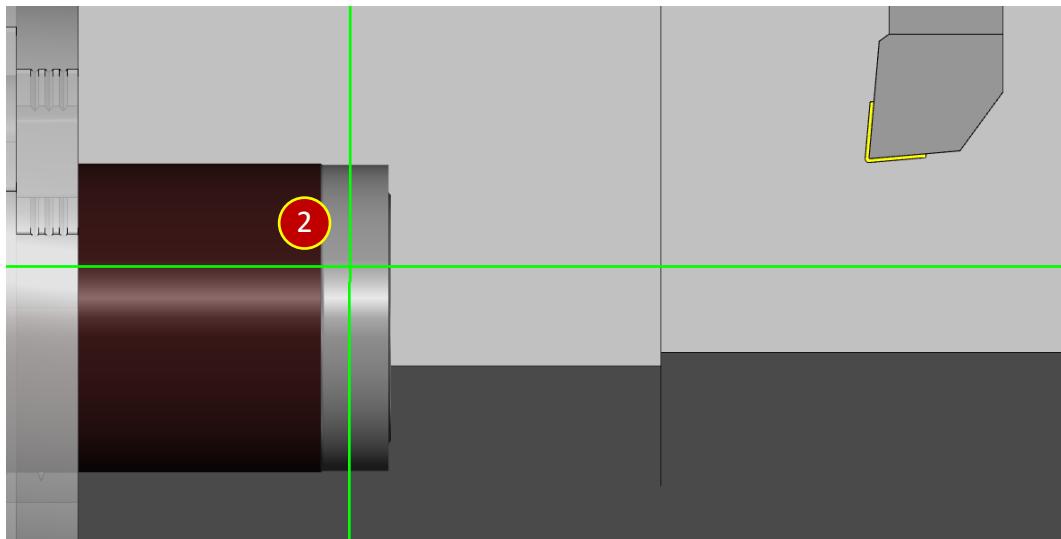
(8) Move the tool leftward in (+Z) direction away from the Workpiece for about 100mm,  
press 【Spindle Stop】



## Manual

## 7.2.6 External Diameter Measurement

- (1) Press 【Meas. Tool】 to open the dimension measurement auxiliary function
- (2) Use Arrow keys and Select button of measurement panel to press the cut external diameter, the result diameter is 64.476
- (3) Press 【Close】 to close the window



### 7.2.7 Work Shift Coordinate Setting (Master Tool)- Z axis operation

Select External Rough insert,

Take No.1 tool (default) as example to introduce the offset method

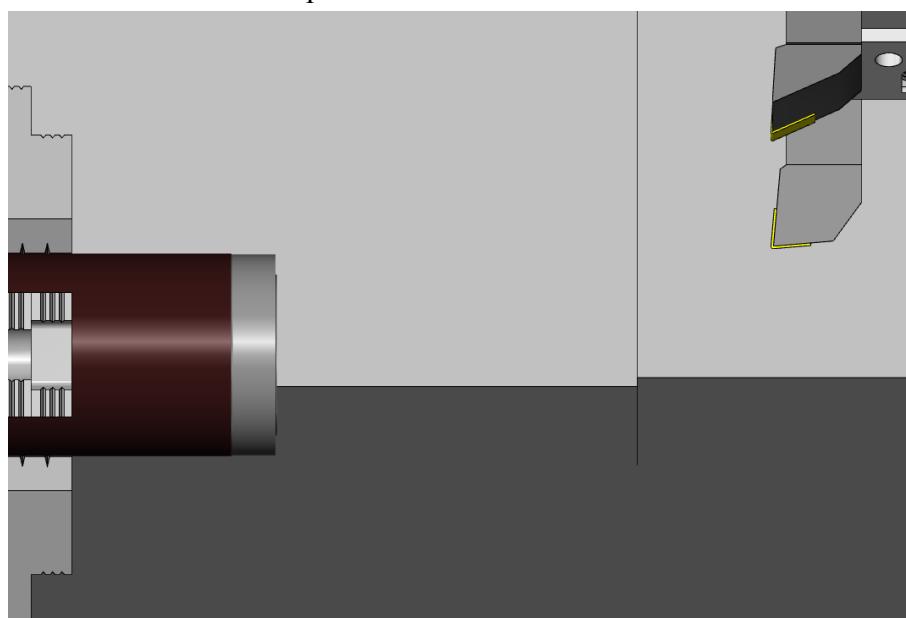
(1) Switch to 【RAPID】 mode and move the turret to Safe Tool Change position

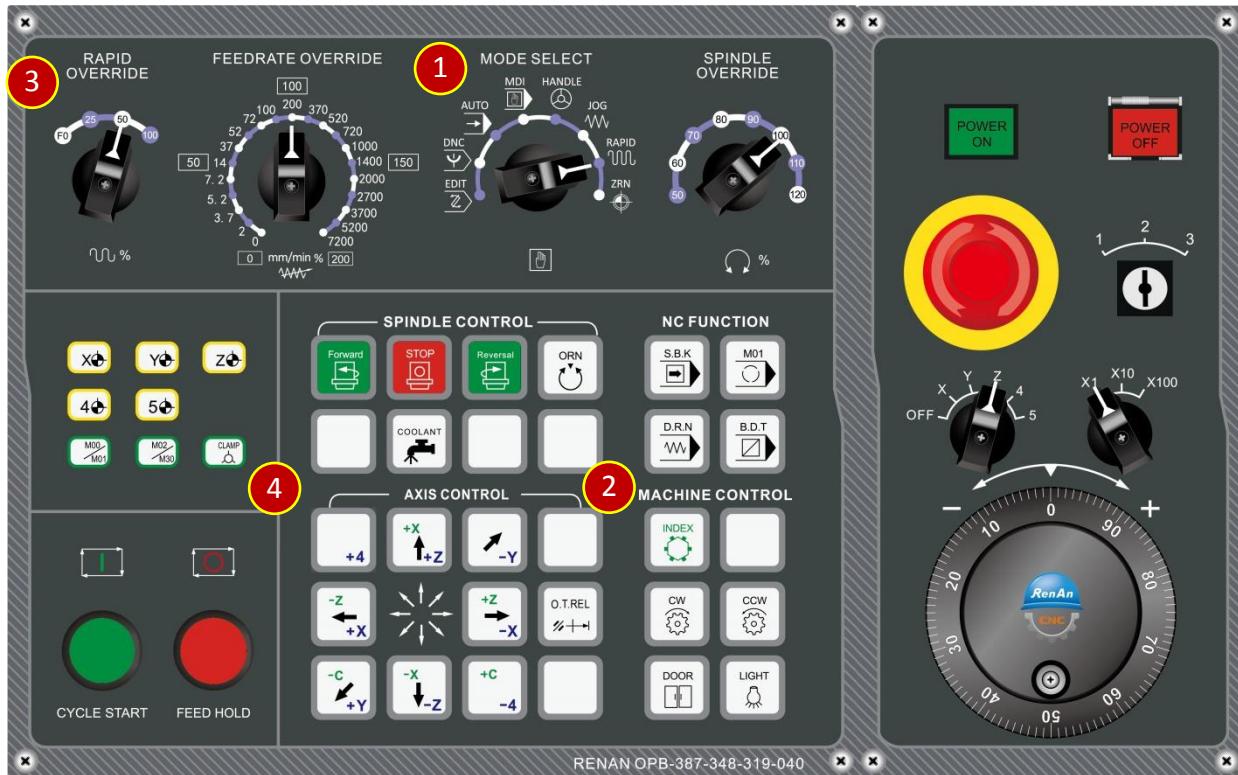
(2) Press 【INDEX】 to rotate to tool No.1

(3) Before manual operation, switch 【RAPID OVERRIDE】 to 50% (or 25%)

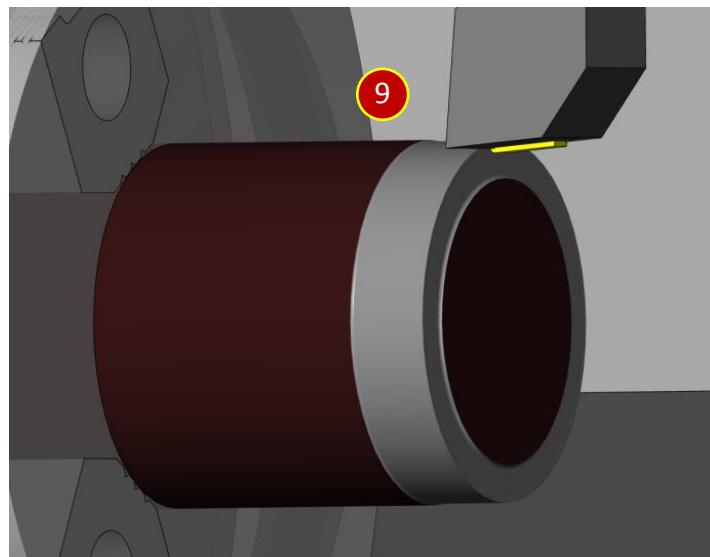
(4) Use Axis keys to move the tool to the position

about 50mm left to the workpiece



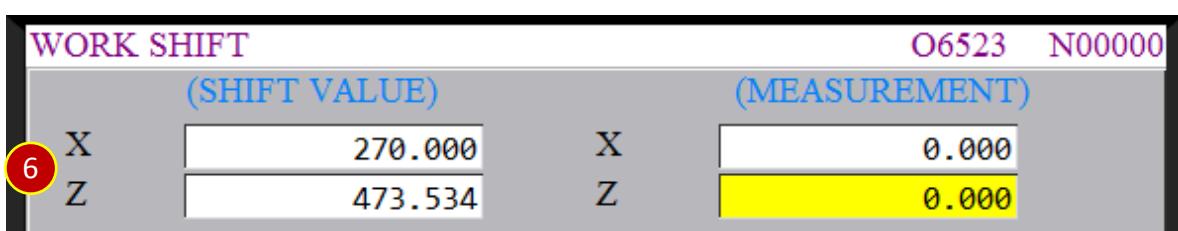
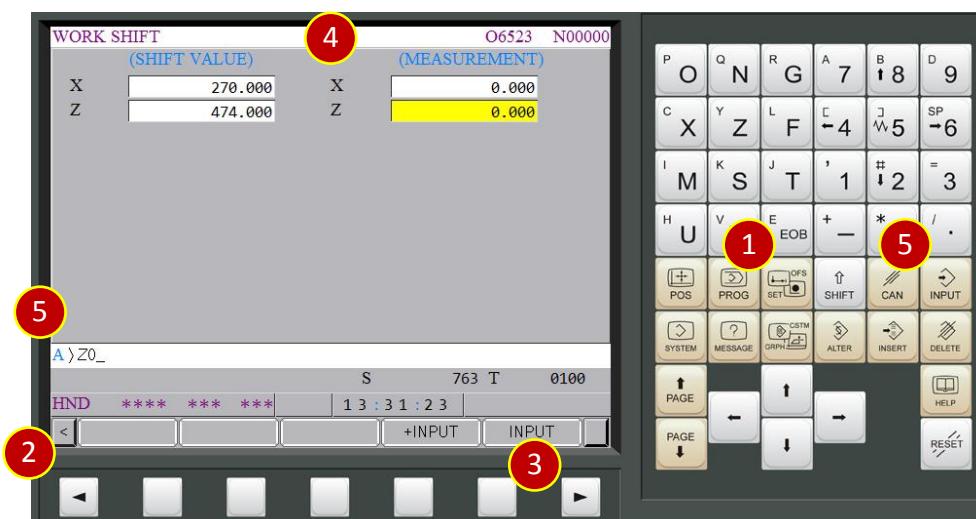


- (5) Switch to 【MPG】 mode
- (6) Press 【Forward】 to rotate the spindle forward
- (7) When the distance between the tool and the workpiece is about 15mm or above, set the handwheel feed rate to x100  
When the distance between the tool and the workpiece is about 15mm or below, set the handwheel feed rate to x10  
Select X axis or Z axis and rotate the handwheel in negative direction (-)
- (8) Rotate 【Handwheel】 to move the tool to workpiece end face, about 5mm below the max external diameter
- (9) Move the tool in -Z direction and stop when it cut the workpiece end face slightly.  
Retract two scales (0.02mm)  
Note: When chips and cuts appear, it means the cutting has reached the workpiece)



### 7.2.8 Work Shift Coordinate Setting (Master Tool)- Z Axis Operation

- (1) Press 【OFS/SET】 offset button
- (2) Press 【◀】 on the bottom left corner  
to return to the initial page of the function options
- (3) Press 【▶】 to flip two pages and enter 【WORK SHIFT】 setting window
- (4) Move the cursor to the measurement Z column
- (5) Key in “Z0” and press 【INPUT】
- (6) When the shift value shows Z= 473.534, Z axis work shift setting is done



## 7.2.9 Work Shift Coordinate Setting- X Axis Operation

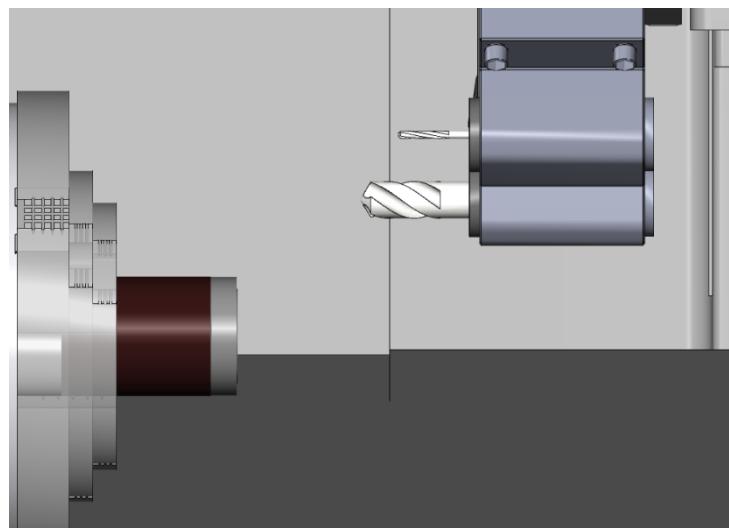
Take Drill No.9 as the example (Drill default size=20mm) to describe the offset method

(1) Switch to 【RAPID】 mode and adjust Rapid override to 50%

(2) Move the tool holder to the Safe Tool Change position  
(about 150mm left to the workpiece)

(3) Press 【INDEX】 to rotate to tool No.9  
(4) Use the Axis keys to move the tool

to the position about 50 mm left to the workpiece Endface

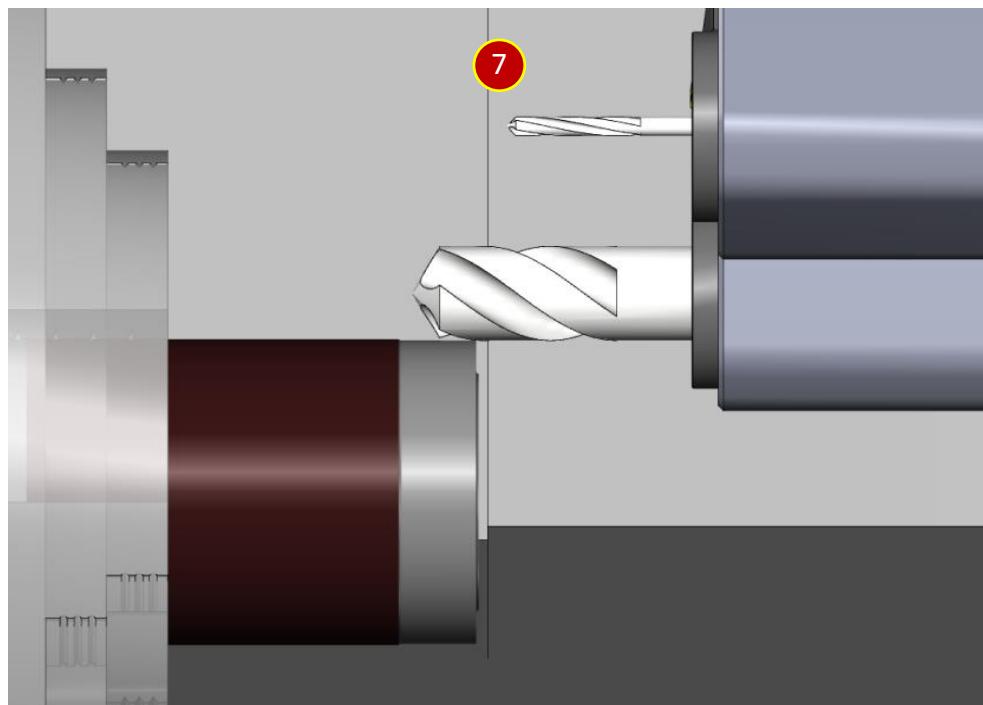


## Manual

(5) Switch to 【MPG】 mode

(6) Rotate handwheel to Move the tool Setter to the top of the external diameter, about 10mm left to the end face

(7) Move the tool in the -X direction and slightly attach the external diameter (till chips and cuts appear), and retract two Scale (0.02mm)



## 7.2.10 Work Shift Coordinate Setting (Master Tool)-X Axis Setting

(1) Move cursor to the measurement X column

(2) Key in X 84.476

(a) (64.476+20=84.476)

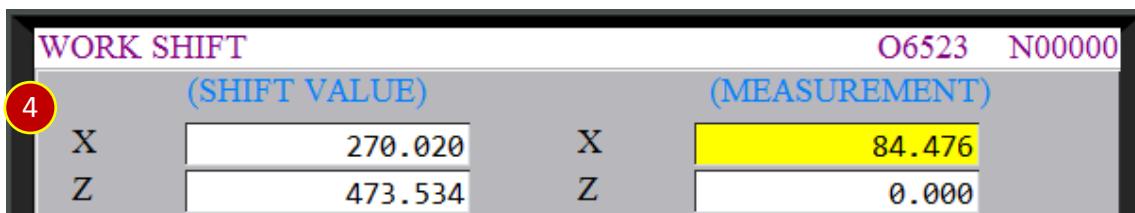
(The measured material diameter= 64.476mm, Drill diameter= 20mm)

(3) Press 【INPUT】

(4) When work shift value shows X=270.006, it means the work shift setting of X axis is done

\*Note: The distance between the holder center and spindle center line is provided,

it allows to key in the value in the X axis work shift without offsetting to gain the data



## Manual

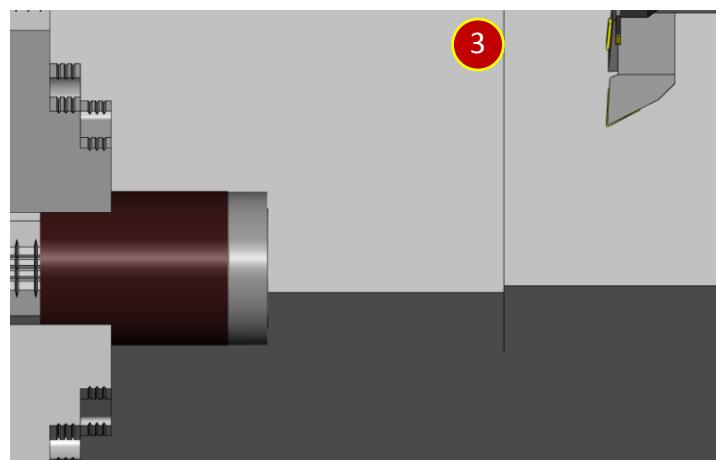
## 7.3 Z Axis Tool Offset (Offset)

### 7.3.1 Tool Geometry Offsetting (Each Tool-Turning Tool)-Z Axis Offset Operation

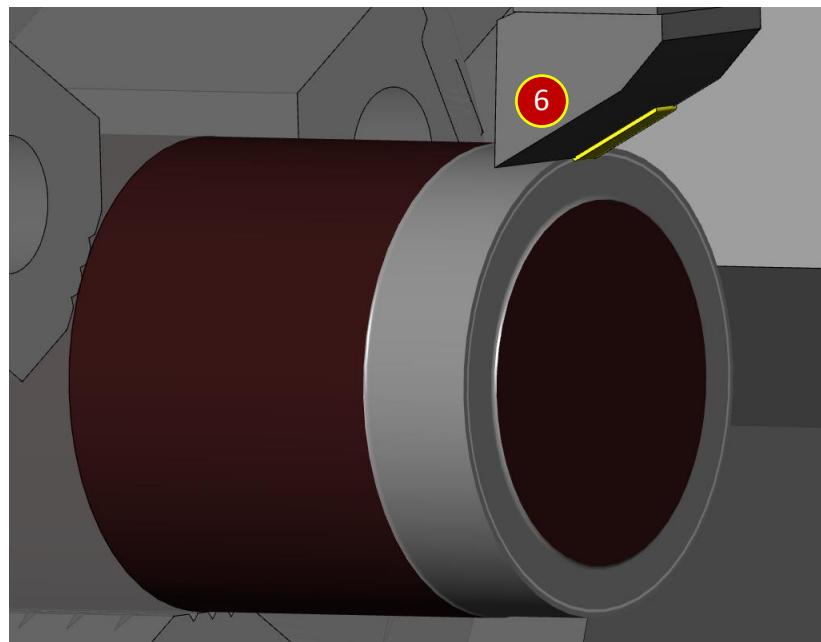
Set the geometry offset of each tool.

Take tool No.2 as the example, introduce the offset method.

- (1) Move the tool holder to Safe Tool Change position (about 150mm left to the workpiece)
- (2) Press 【INDEX】 to switch to tool No.2
- (3) Use Axis keys to rapid traverse the tool to the position about 50mm left to the workpiece
- (4) Switch to 【MPG】 mode
- (5) Press 【Forward】

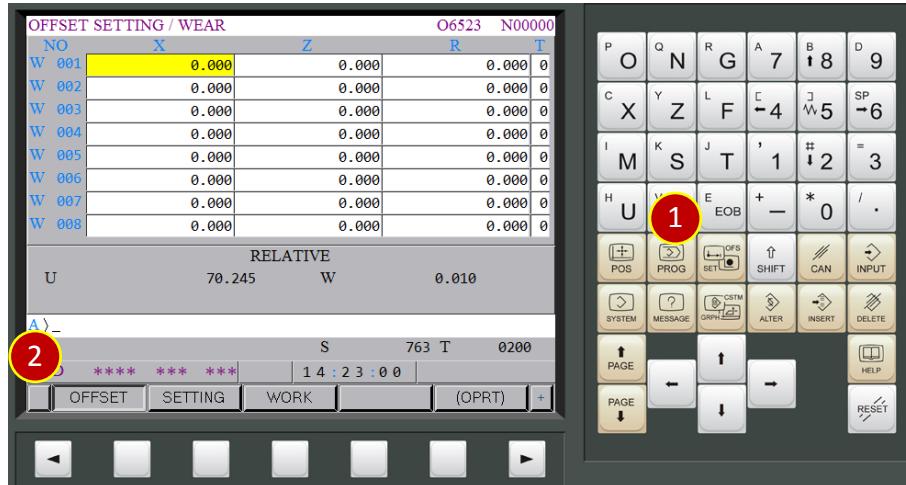


- (6) Use the hand wheel to move the tool to the workpiece Endface,  
about 5 mm below the max diameter of the workpiece  
(The end face that is already cut by the master tool )  
(When chips and cuts appear, it means the cutting has reached the workpiece,  
please stop moving forward)
- (7) Retract the tool by 0.02 mm (In the Z Axis direction)  
(Return to the base plane, because when it attached to the workpiece,  
it means the cutting has reached the workpiece)  
(Avoid the noises made by blades friction)

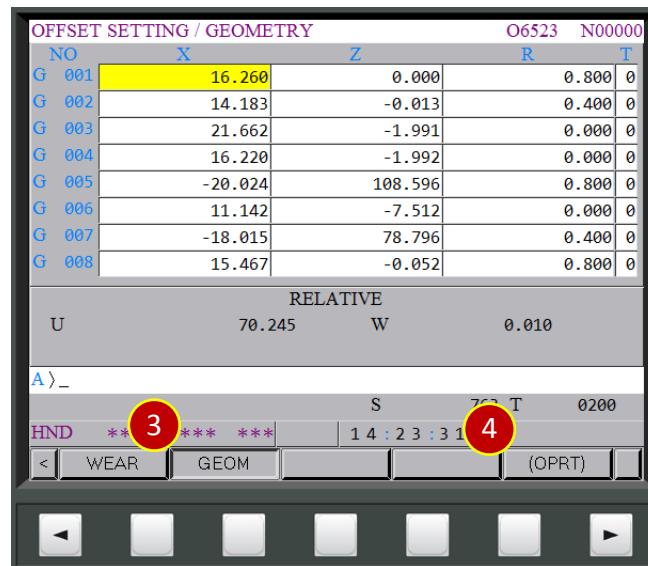


## 7.3.2 Tool Geometry Offset Setting (Each Tool-Turning Tool)-Z Axis Offset Setting

- (1) Press 【OFS/SET】 via controller panel to display Offset setting window
- (2) Press 【OFFSET】 to open the offset window

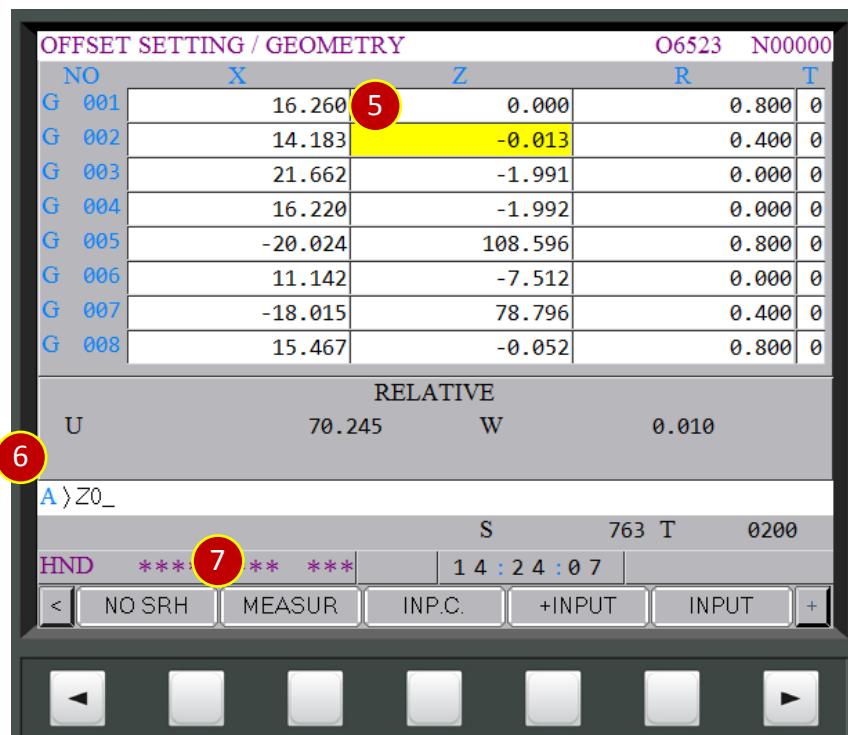


- (3) Press 【GEOM】 to display the geometry offset of each tool
- (4) Press 【(OPRT)】 to display auxiliary function



## Manual

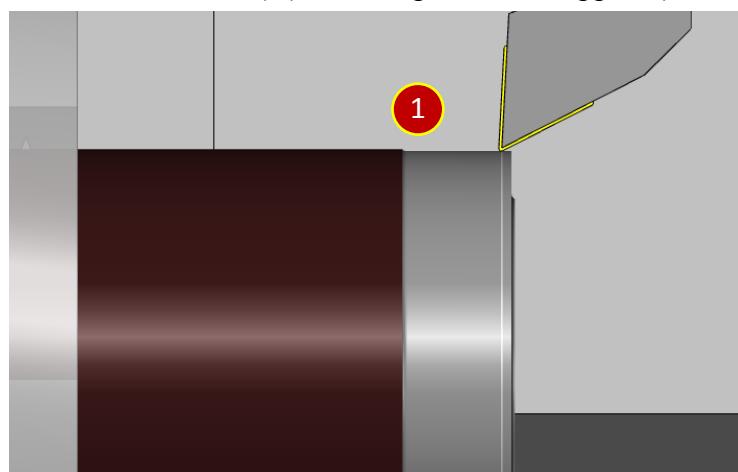
- (5) Move the cursor to the position of current offset tool G002
- (6) Key in "Z0"
- (7) Press 【MEASUR】. When the column displays 0.010,  
it means Z axis offset has completed



### 7.3.3 Tool Geometry Offset (Each Tool-Turning Tool) - X Axis Offset

(1) Rotate handwheel to slightly attach the tool to the external diameter

(About 5mm left to the Endface) (Until chips and cuts appears)



(2) Key in the measured external diameter value “X64.476”

(3) Press 【MEASUR】 and when it displays X=14.247, it means X axis offset has completed

(4) Offset the Geometry Offset of all the desired tool with the method above mentioned

(Note: Compare the elongation to other external tool as dimension reference)

OFFSET SETTING / GEOMETRY O6523 N00000				
NO	X	Z	R	T
G 001	16.260	0.000	0.800	0
G 002	14.183	0.010	0.400	0
G 003	21.662	-1.991	0.000	0
G 004	16.220	-1.992	0.000	0
G 005	-20.024	108.596	0.800	0
G 006	11.142	-7.512	0.000	0
G 007	-18.015	78.796	0.400	0
G 008	15.467	-0.052	0.800	0

OFFSET SETTING / GEOMETRY O6523 N00000				
NO	X	Z	R	T
G 001	16.260	0.000	0.800	0
G 002	14.247	0.010	0.400	0
G 003	21.662	-1.991	0.000	0
G 004	16.220	-1.992	0.000	0
G 005	-20.024	108.596	0.800	0
G 006	11.142	-7.512	0.000	0
G 007	-18.015	78.796	0.400	0
G 008	15.467	-0.052	0.800	0

RELATIVE				
U	78.723	W	-4.456	
A >	X64.476	_	S	763 T 0200
HND	***	*** ***	1 4 : 3 0 : 1 7	
< NO SRH	MEASUR	INP.C.	+INPUT	INPUT +
<input type="button" value="◀"/> <input type="button" value=""/> <input type="button" value="▶"/>				

RELATIVE				
U	78.723	W	-4.456	
A >	_	S	763 T 0200	
HND	***	*** ***	1 4 : 3 0 : 4 6	
< NO SRH	MEASUR	INP.C.	+INPUT	INPUT +
<input type="button" value="◀"/> <input type="button" value=""/> <input type="button" value="▶"/>				

# Manual

## 7.4 Auto-Run

### 7.4.1 Start the Program

Use Auto-Run to execute program

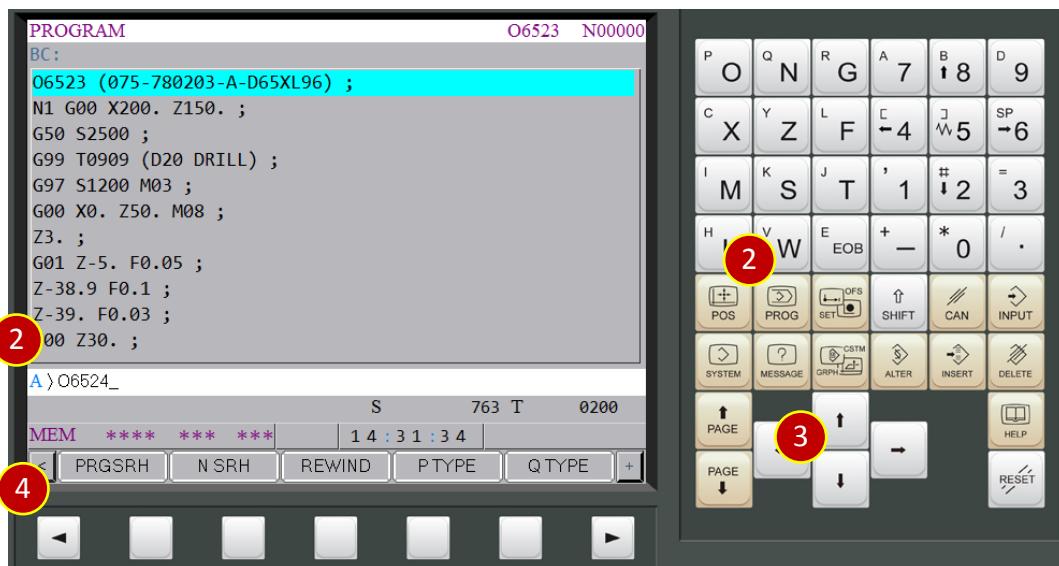
(1) Switch the mode to 【AUTO】

(2) Press 【PROG】

Key in the NC code that is desired to simulate e.g. O6524

(3) Press 【↓】 search button to open and display the program

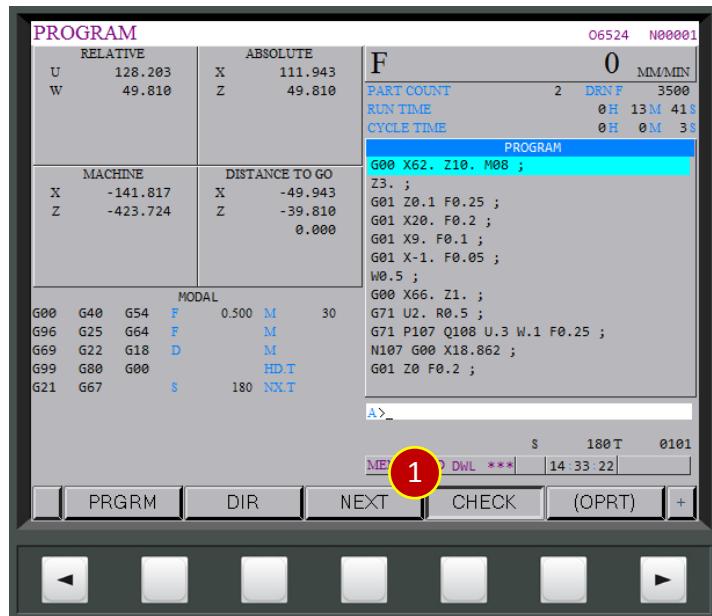
(4) Press 【←】 to return to previous menu



## Manual

## 7.4.2 Program AUTO Run

- (1) Press 【CHECK】 to display coordinates and program
- (2) Switch 【RAPID OVERRIDE】 override knob to 25%
- (3) Press 【S.B.K】 (It's active when the button is illuminated)
- (4) Press 【CYCLE START】 to run the program
- (5) Watch the tool moving and when the tool is about 50mm left to the workpiece, press 【FEED HOLD】 immediately to stop the tool
- (6) Check if the position of the tool and workpiece agrees with the dimension displayed via the program coordinates  
e.g. Program Absolute Coordinates Z= 39.810 , about 40mm left to the workpiece measured by vision



## Manual

- (7) Press 【CYCLE START】 to continue the process
- (8) When the tool gets closer to the workpiece (about 20mm ),  
press 【FEED HOLD】 to pause the feed
- (9) Switch 【RAPID OVERRIDE】 override knob to 0F position
- (10) Check if the position of the tool and workpiece matches the size of program coordinates  
e.g.

Program Absolute Position      Distance to go

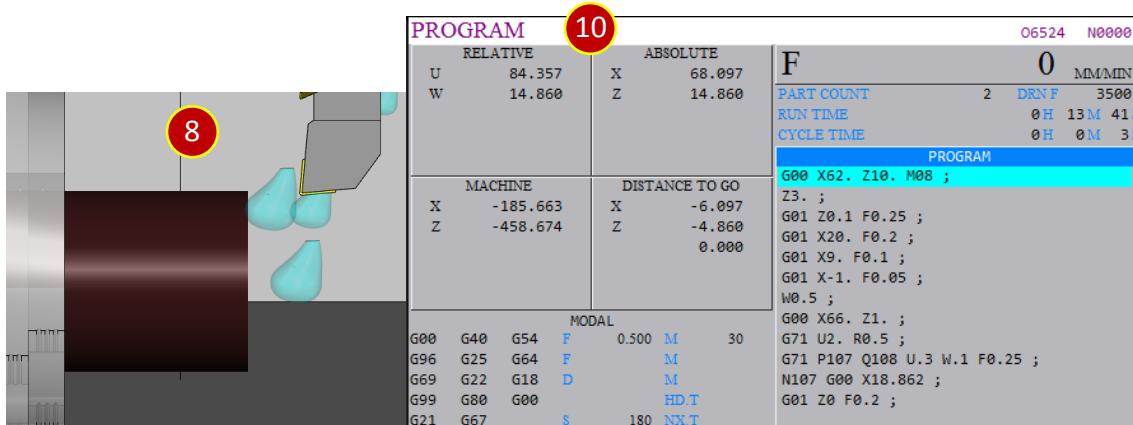
X= 62.000      X= - 0.000

Z= 14.860      Z= - 4.860

- (11) Press 【CYCLE START】 to continue programing till the cutting process is done.

\*Caution:

- (a) During the process of trial cutting,  
as long as the process is within the approaching range of the tool and workpiece,  
keep Rapid Traverse at F0 position for the sake of safety
- (b) Press 【FEED HOLD】 first whenever sensing any impropriety,  
check the alarm details later



## Manual

## 7.5 Machine Alarm

When the error occurs, alarm codes will display on the controller panel

## 7.5.1 ALARM Clear

- (1) When the controller panel displays “ALM”
- (2) Check the alarming code and messages, then clear the errors according to its messages
- (3) Press 【RESET】 to clear the alarm after getting the errors messages
- (4) Press 【HISTORY】 in 【MESSAGE】 to check history record
- (5) History record area



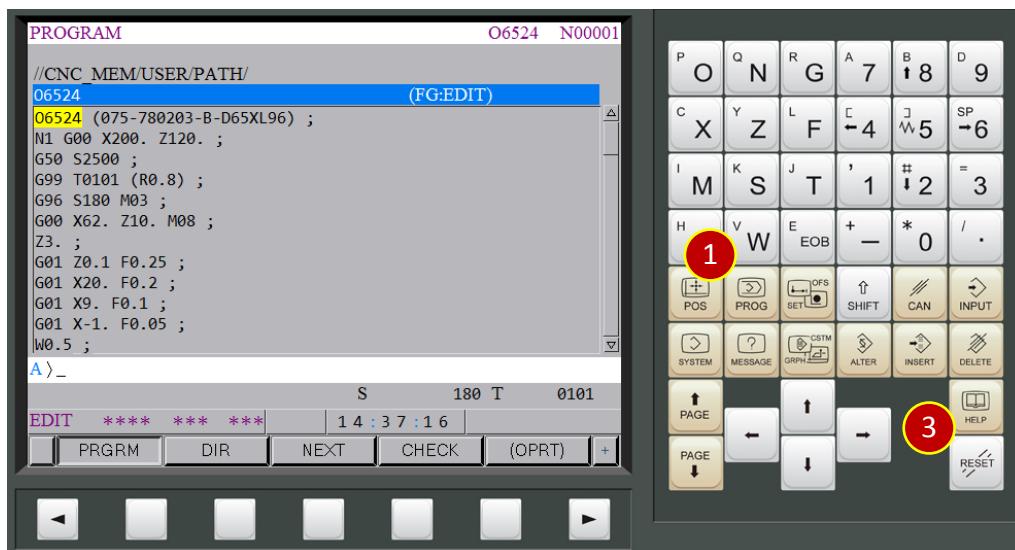
## Manual

### 7.5.2 Resuming AUTO Run

When the program is interrupted,

or trying to resume AUTO Run from the alarm notification window:

- (1) Press 【PROG】 to return to the program display window
- (2) Switch to 【EDIT】 mode via Modal Select
- (3) Press 【RESET】 and move the cursor back to the beginning of the program
- (4) Reselect the required operation mode  
e.g. 【AUTO】 mode
- (5) Make sure the cursor is at the beginning of NC code  
and press 【CYCLE START】 to AutoRun



# CNC Simulator

## CAD/CAM System

### Training Equipment



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

5F-3, No.1 Keji Road, Dali Dist., Taichung City 412028, Taiwan

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



RenAn Website