USER GUIDE

MAXX DS200 series

Robots and Design

www.stomshop.pro

This manual is used for MAXX DS200 series only.

Please contact the address below for any questions.

Address ▸ Pangyo-ro 700, E-707, Seongnam-City, Gyunggi-do, Korea 13516

Tel ▸ +82-31-708-2684

Fax ▸ +82-31-706-9093

E-mail ▸ maxx@rnd.re.kr

Manual provided with the machine may not include the latest information.

CONTENTS

1. EPNC Installation

Program Installation

To install the program, copy PNC folder. PNC folder is organized as below.

① **<Bin> Folder**

<Bin> contains necessary DLL files to execute Pnc.exe and the program. Below is the description of main executable file and DLL.

File name	Description	Remarks
PNC.exe	PNC Program file	
PComm32.dll	DLL files to manage Pmac Controller	

① **<Config> Folder**

<Config> contains necessary setting files to execute PNC program. Detailed description for each setting file is in “8. Config file” in <Config> folder.

② **<Log> Folder**

<Log> is log files of machine. It is saved in Pnclog.db file. To check contents of log files, special viewer is required.

③ **<NcFiles> Folder**

<NcFiles> is to save NC files for milling. It can be used to copy files from outside.

④ **<Resource> Folder**

<Resource> is to save resources that will be used in the program. It is not allowed to delete or change names of files saved in the folder.

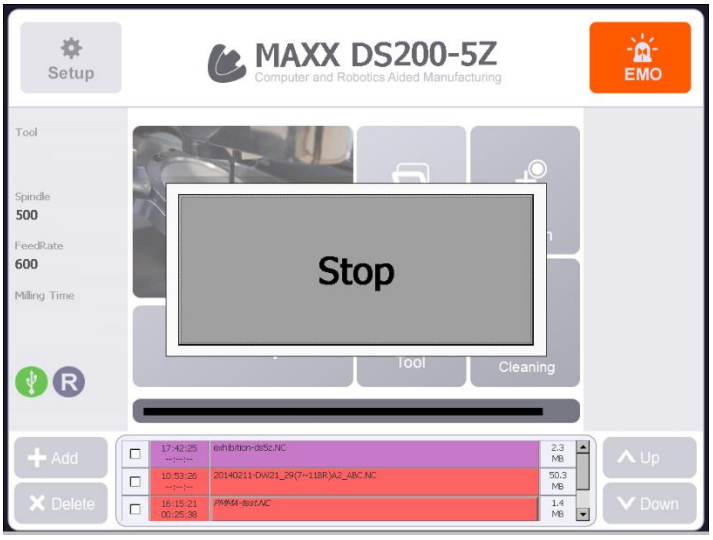
2. Description of EPNC

2.1 Initialization window

Initializing window is shown as below. The machine can be initialized with only one button.

Initial window of EPNC

EPNC starts initializing when “HOME” is clicked.



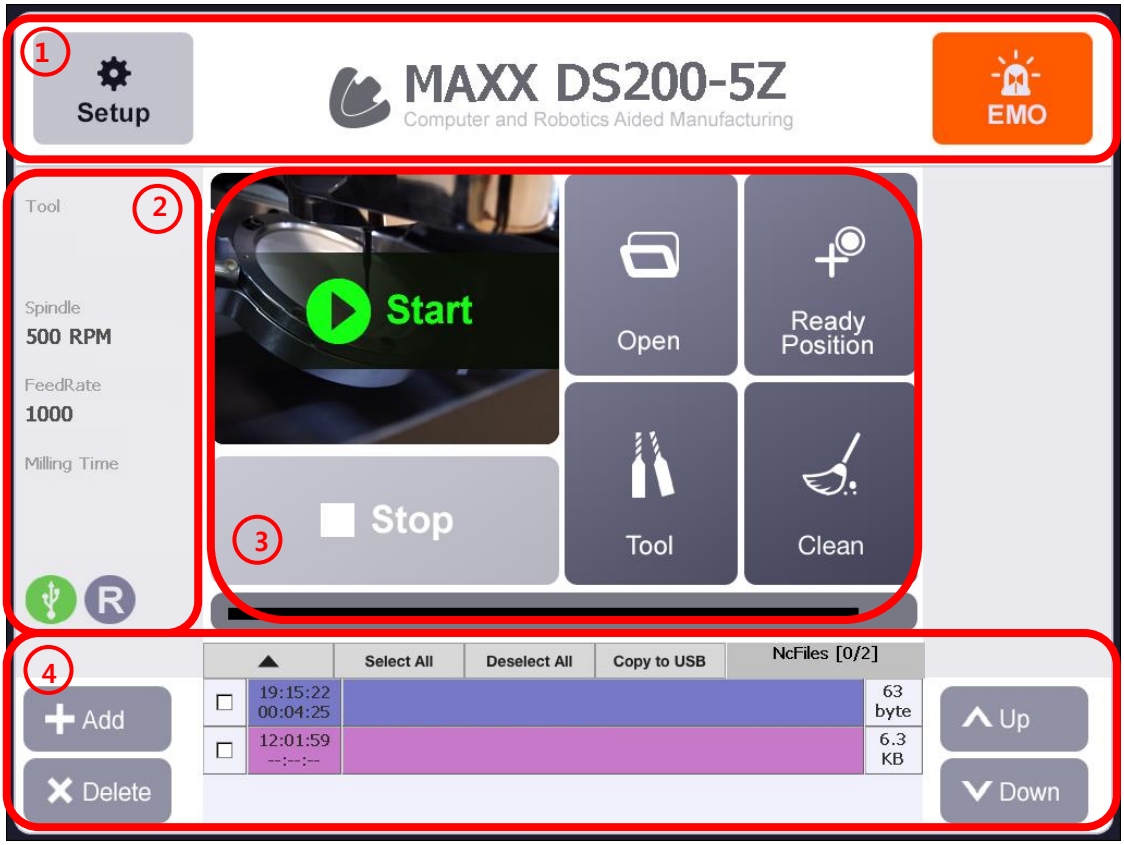
The window flashes for about 15 seconds while initializing.

It stops initializing when “STOP” is clicked.

2.2 Main window

Once initialization is completed, main window will appear.

Name of machine is written at the top and commonly used functions are shown on the main window.



2.2.1 Functions of main window

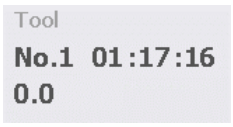
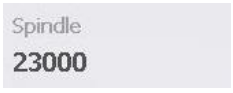
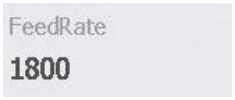
2.2.1.1 Operational setting & EMO

Function of setup and EMO is as below.

	**General setting and changing offset values** - Refer to p.14 Setup
	**EMO (Emergency) Button** - For immediate stop - Emergency stop for hardware. - It changes to “RESET” once clicked 

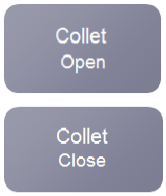
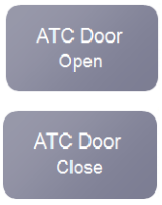
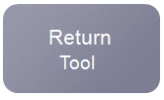
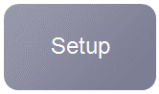
2.2.1.2 Machine status

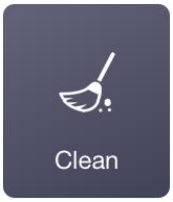
It shows machine's status and activated condition.

		**Tool** - Overall information of tool that is clamped. - Tool number and work hour is shown in [hour : minute : second].
		**Spindle RPM** - Number of spin of spindle per minute (Max: 60,000).
		**FeedRate (mm/min)** - Speed of a tool for a constant working cycle or hour. (moving speed of spindle)
		**Milling Time** - Milling time in [hour : minute : second].
 Deactivated	 Activated	**USB** - It is activated when USB is connected.
 Deactivated	 Activated	**Remote program** -It is activated when remote program is connected.

2.2.1.3 Commonly used functions

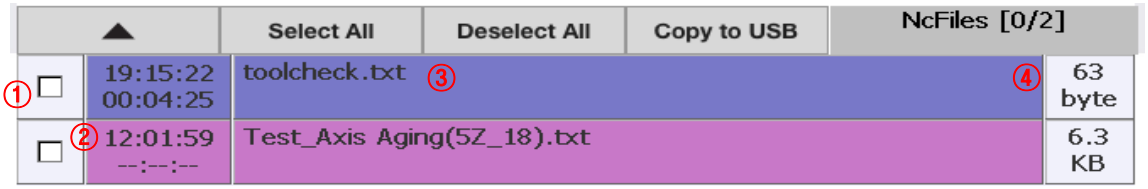
Below is commonly used functions that are essential in operating the machine.

<table>
<tr><td> 
</td><td colspan="2">Start
- Starting NC file.
- Once clicked, it changes to “PAUSE”. 
</td></tr>
<tr><td> 
</td><td colspan="2">Stop
- It is activated after starting NC file.
- Stopping NC file.
- Emergency stop of software.</td></tr>
<tr><td> 
</td><td colspan="2">Open
- Open selected NC file.</td></tr>
<tr><td> 
</td><td colspan="2">Ready Position
- Spindle and stage moves to preset position.</td></tr>
<tr><td rowspan="4"> 
</td><td colspan="2">Tool
- Below functions are shown when “Tool” is clicked.</td></tr>
<tr><td> 
</td><td>* Collet Open / Close
To open and close collet.
Tool number is reset when collet is open.</td></tr>
<tr><td> 
</td><td>* ATC (Auto Tool Changer) (*DS200-4W/4WA)
To open and close ATC door.</td></tr>
<tr><td> 
</td><td>* Return Tool
To return tool that is clamped.</td></tr>
<tr><td> 
</td><td>* Setup</td></tr>
</table>

	**Cleaning Mode** **DS200-5Z** To clean floor of milling room. It is recommended to use with brush to dust inside of milling room. **DS200-4W / 4WA** To operate coolant for cleaning. Connect cleaning hose to spread coolant in milling room. Be careful not to spread it to bellows behind spindle.
	**Milling status** Shows milling status in green (%).

2.2.1.4 NC File

Displays information related to NC file and each functions are as shown below.



① Select file

② Milling start time / milling time

③ NC file name

④ NC file size

⑤ ▲: to show all the list of NC file

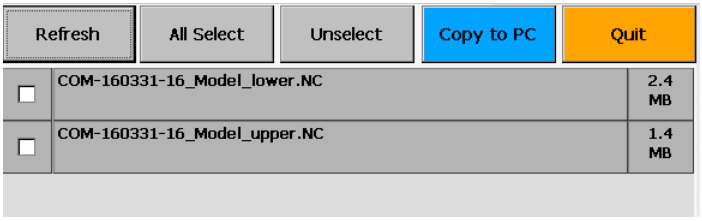
⑥ Select All

⑦ Deselect All

⑧ Copy to USB: to copy NC files to USB

Add

- Adding NC file to embedded PC.



Refresh the list in USB

Select all NC files in USB

Unselect NC file

Copy NC file from USB to embedded PC

Quit

Delete

To delete selected NC file

Up Down	**Move NC file** It is to change order of NC files. **"Up"** moves selected files upwards. **"Down"** moves selected files downwards.

☞ **Compatible USB**

Below is a list of USB that are compatible with the machine OS, Microsoft Embedded CE6.0.

1) FAT16
2) FAT32
3) exFAT

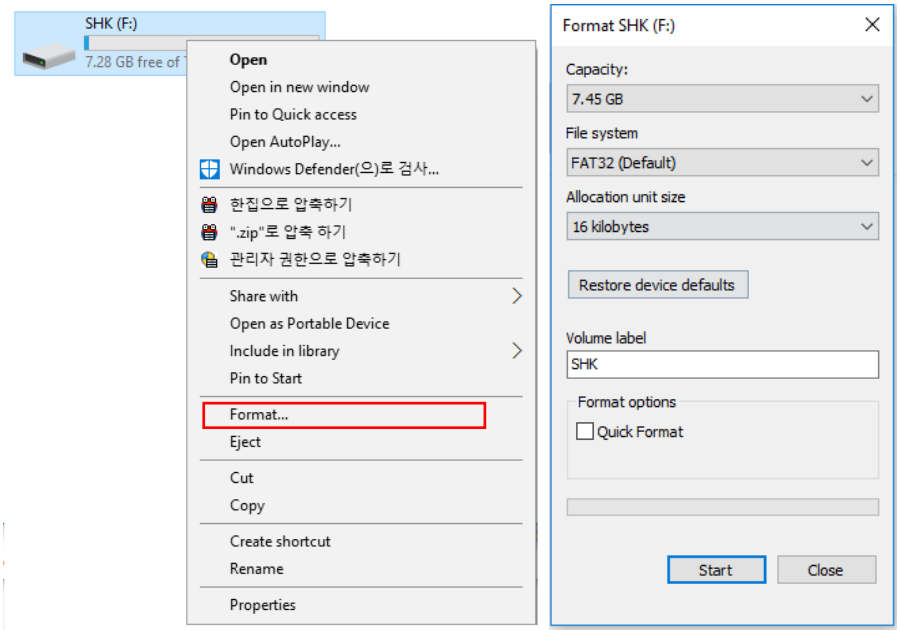
➔ If USB is not compatible with the machine, please change the USB format by following below instruction.

※ How to change USB format

1) Connect USB to computer

2) Click on "my computer"

3) Right click on USB and click "Format (A)"



4) Click "File System" to select the file system that is compatible with the machine.
Do not select "Quick Format" under "Format option".

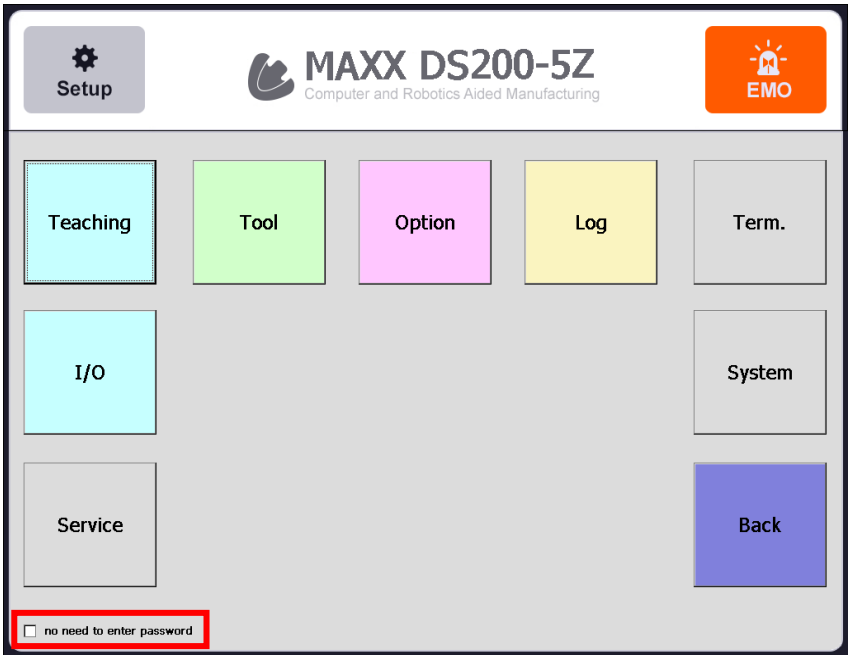
5) Apply changed format.

2.3 Setup

2.3.1 Entering setup

Enter password to enter setup window.

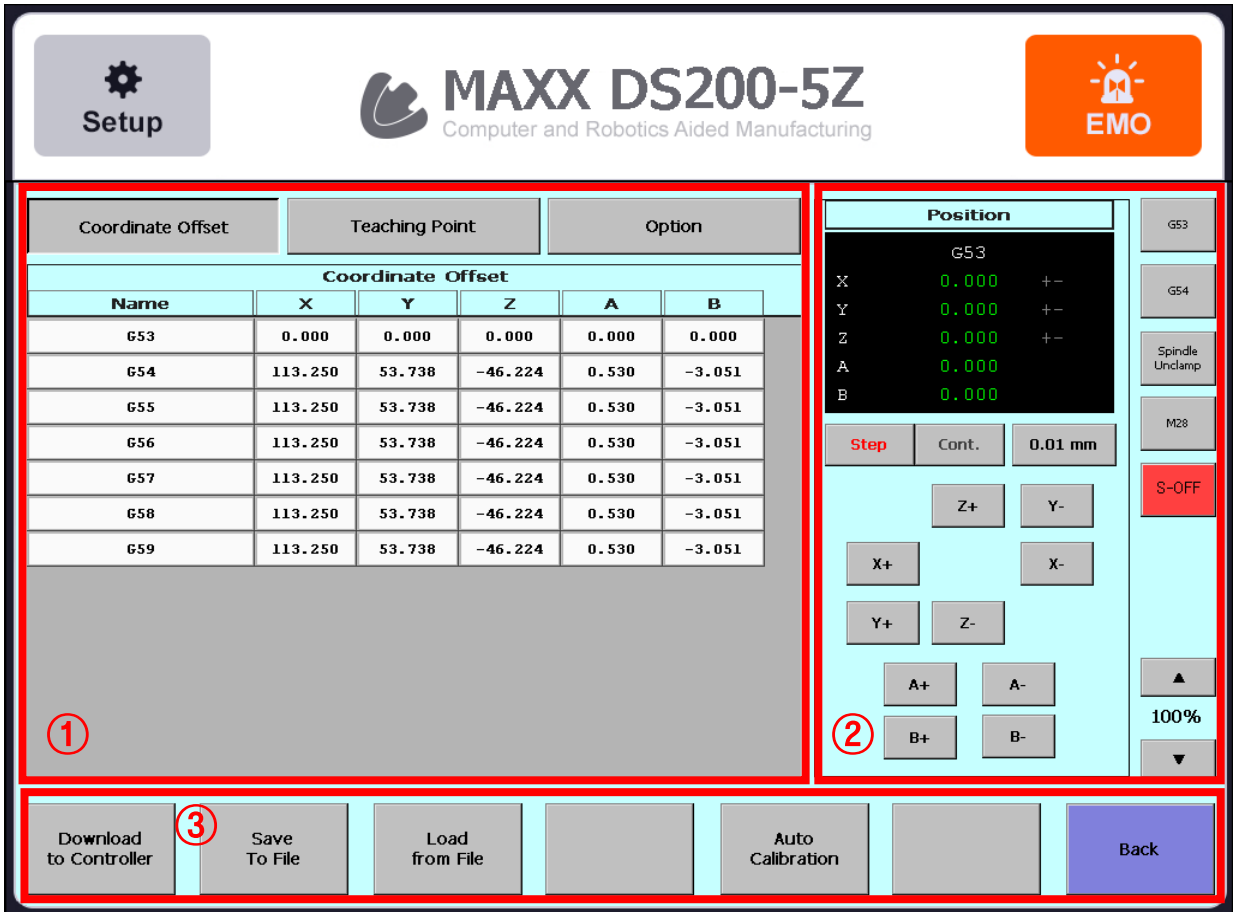
1) User P/W -> ENT (User mode)
2) Manager P/W -> ENT (Manager mode)
3) RND P/W (100321) -> ENT (RND mode)



Click to not enter password when re-entering setup

2.3.2 Teaching

2.3.2.1 Functions of teaching



① **Coordinate and offset display.**
Coordinate offset, teaching point and option.

② **Axis jog and jog shortcut button.**

③ **Opening and saving changed values.**

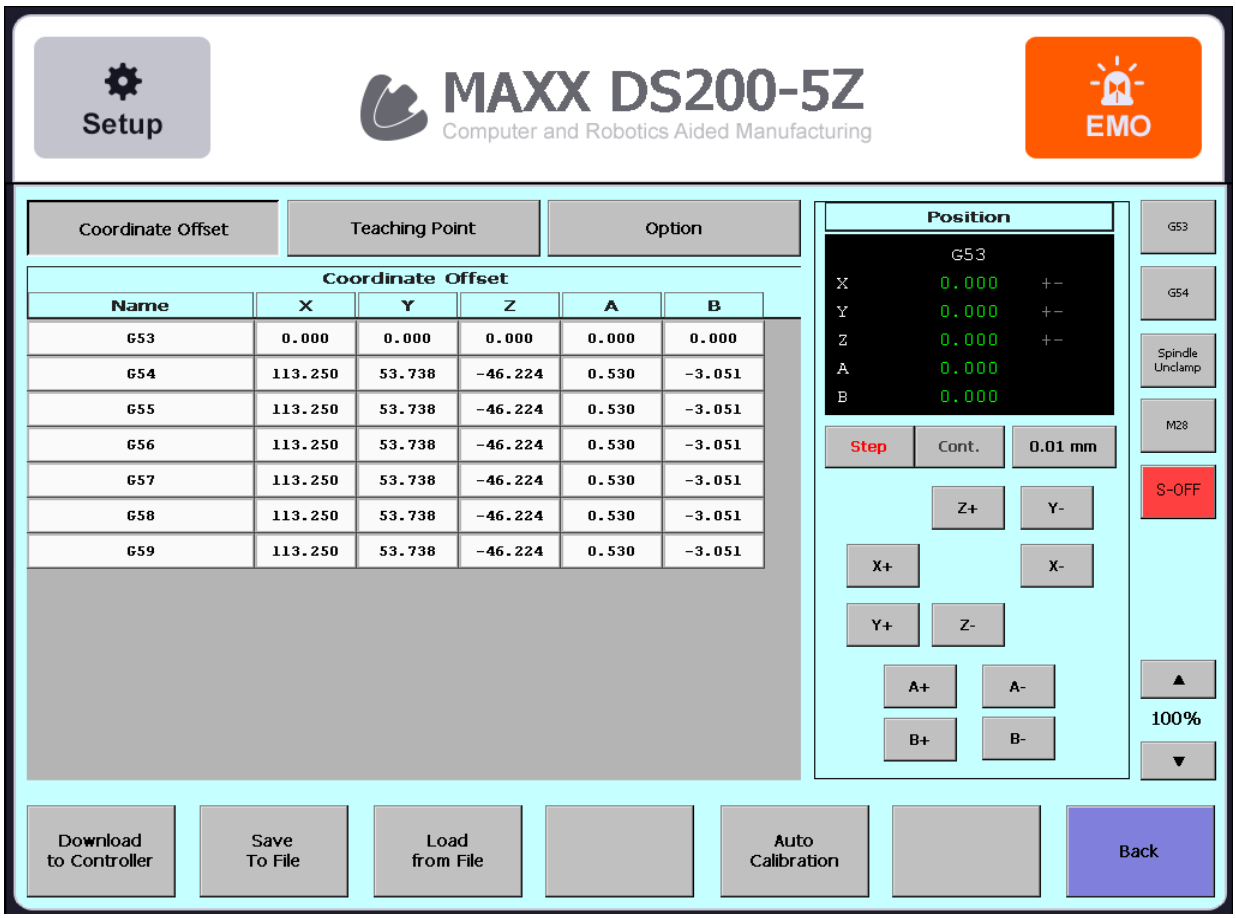
- Download to Controller – save changed coordinates to controller.
- Save To File- save changed coordinates to PC memory.
- Load from File- display coordinates saved in PC memory.
- Auto Calibration (*refer to setup manual)

2.3.2.2 Coordinate offset & Teaching point

To change and save coordinate offset for each axis and tool position.

Coordinate Offset

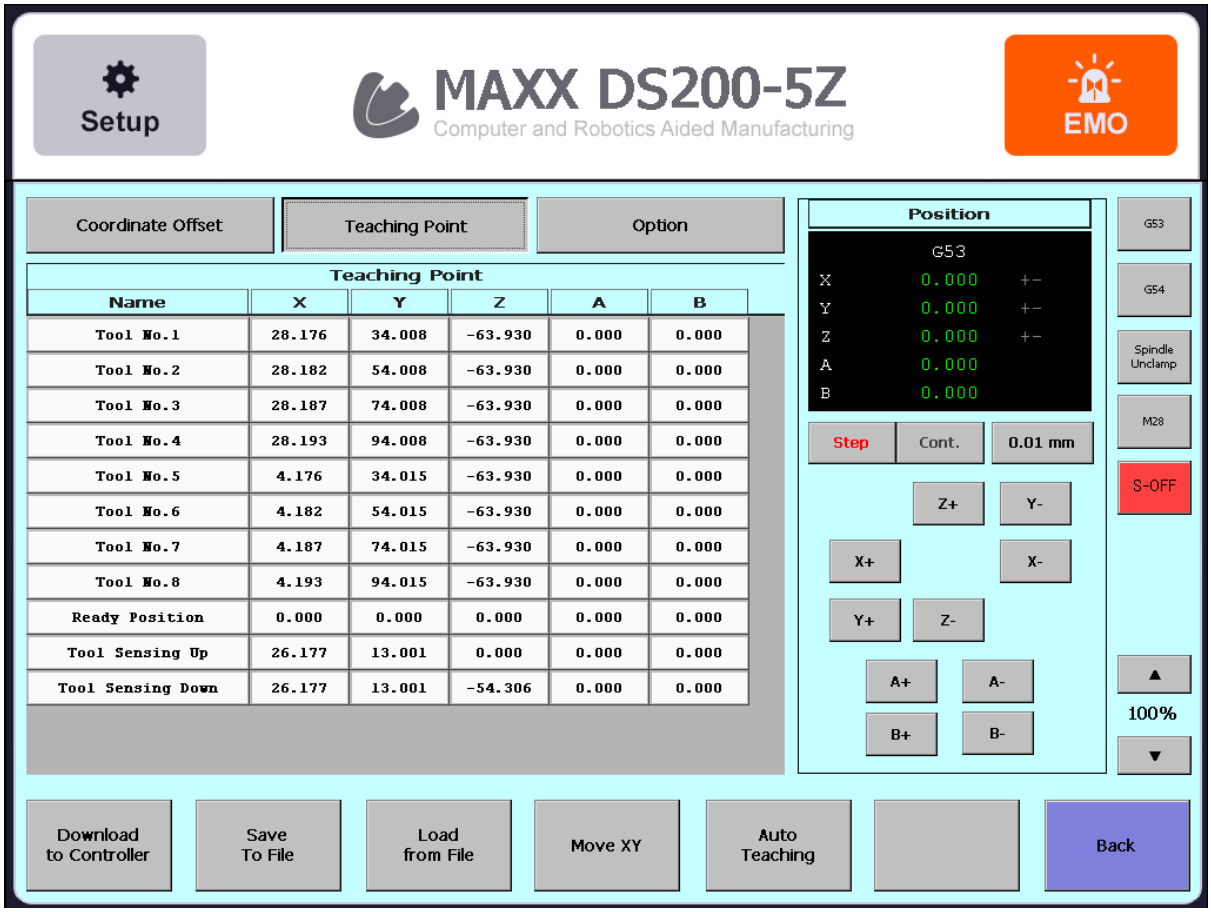
Displays coordinates for each position.



Name	Description	Remarks
G53	Motor coordinates. Coordinates for each axis is based on motor's origin.	
G54-59	Work coordinates	

Teaching Point

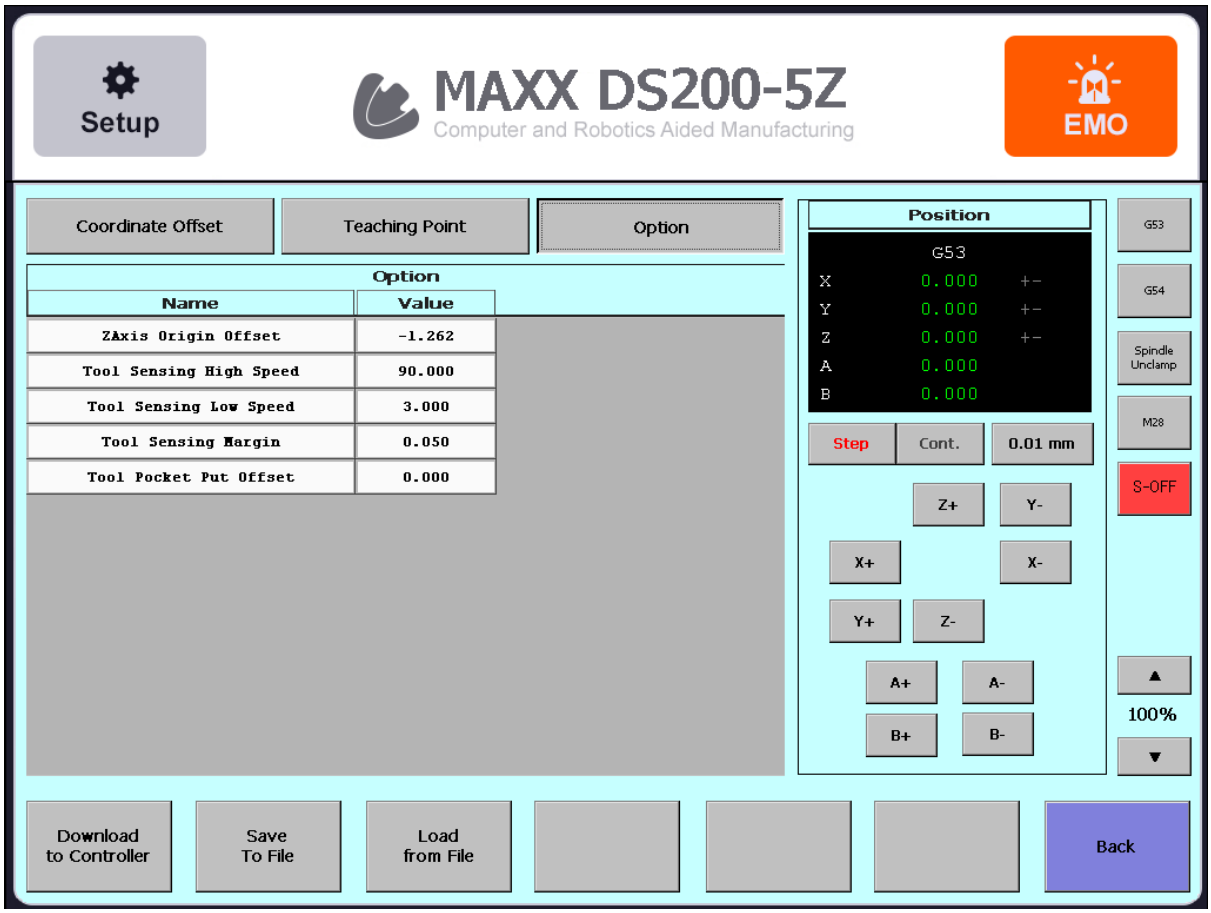
To manage coordinates related to machine operation.



Name	Description	Remarks
Tool1-8	Position of tool 1-8	
Ready Position	Coordinates indicating preset position of spindle	
Tool Sensing Up	Coordinates for tool to move in fast speed to tool sensor	
Tool Sensing Down	Distance between tool and sensor	

2.3.2.3 Option

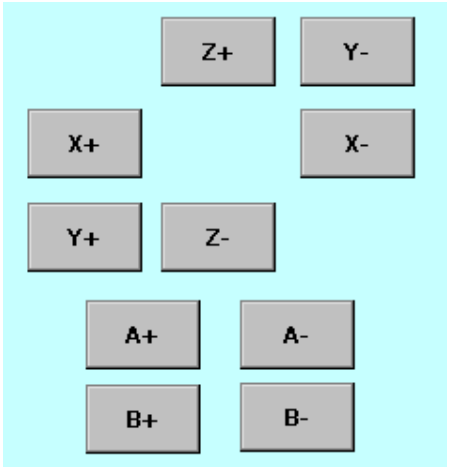
To manage option data.



Option	Description	Remarks
ZAxis Offset Origin	Distance between probing sensor (tool length adjustment sensor) and milling origin in millimeter.	
Tool Sensing High Speed	Highest speed in sensing	
Tool Sensing Low Speed	Lowest speed in sensing	
Tool Sensing Margin	Acceptable range in tool length before and after milling.	
Tool Pocket Put offset	Spindle returns a tool after it moves to Z-axis as offset amount from tool teaching position.	

2.3.2.4 Jog

To move motor by using jog buttons.

	**G53** - Displays coordinates being used (coordinates based on motor) 0.000 - Indicates current coordinates of each axis **+ - (Symbol)** Limit for each axis is shown (indicates direction) It is indicated as **+ - or + -** **+ -** changes into red when it reaches the limit.
	**Step** - To move preset amount only. **Cont. (Continuous**) - To move continuously. **0.01mm** - To decide moving amount in “Step” mode.
	**X±, Y±, Z± (Jog moving button)** - To move stage in X, Y and Z axis direction. Check “Step”, “Cont.”, and “moving amount” before moving. **A/B Axis Jog** - To move A and B axis.

	**G53~G54** - To select coordinates **Spindle Clamp / Unclamp** - To open/close spindle collet **M28** - To operate vacuum or pump required for dry/wet milling. **SRV-OFF** - Servo motor off **▲ 100% ▼** - To control moving speed of jog (Max. 100%)

Move XY. <Teaching Position>

It is to move X and Y axis to selected coordinates.

Click coordinates in “Teaching” and click “Move XY” to move to preset coordinates automatically. Z axis should be at 0.0 before moving X and Y axis.

Jog

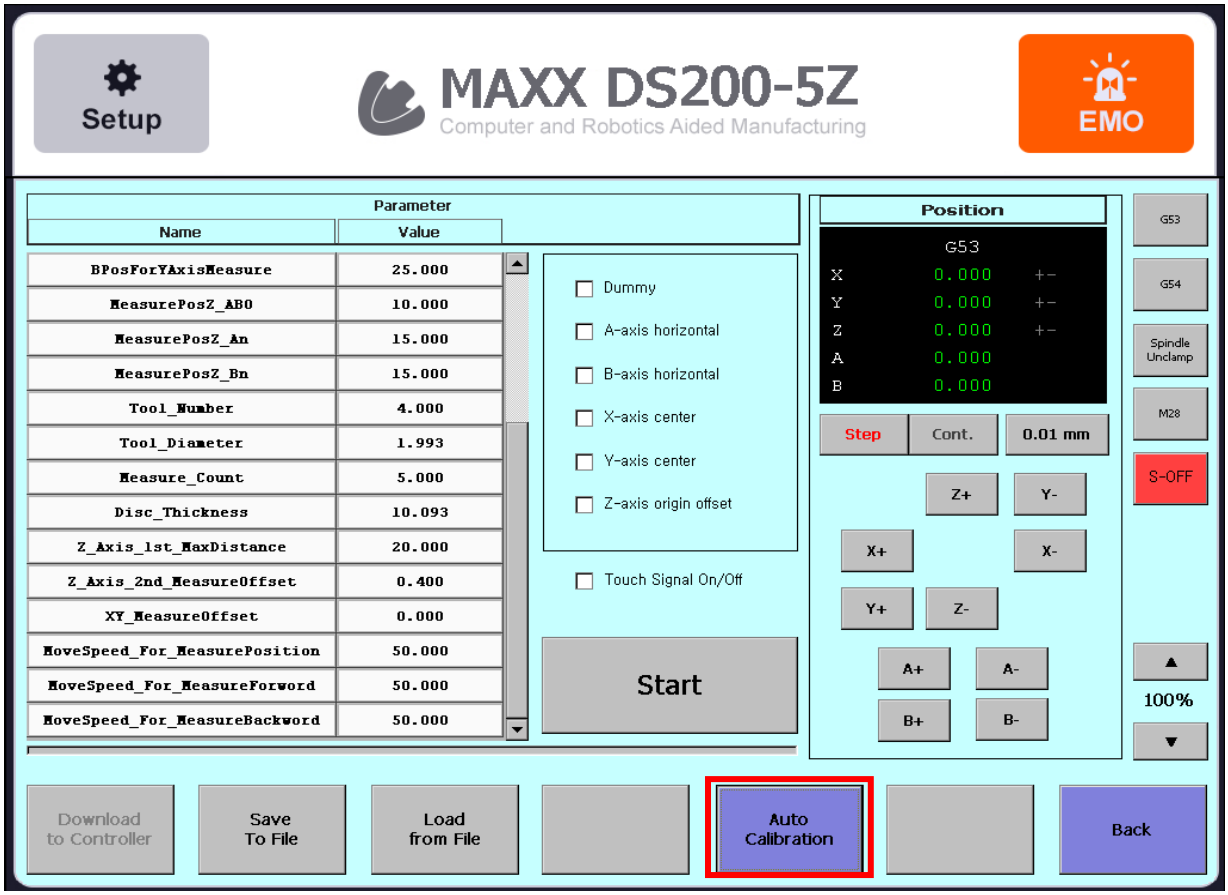
It is to move motor into different position.

1. “Continuous” mode is to move motor continuously.

2. “Step” mode is to move motor only for preset amount. The amount can be set between 1.0mm to 0.01mm.

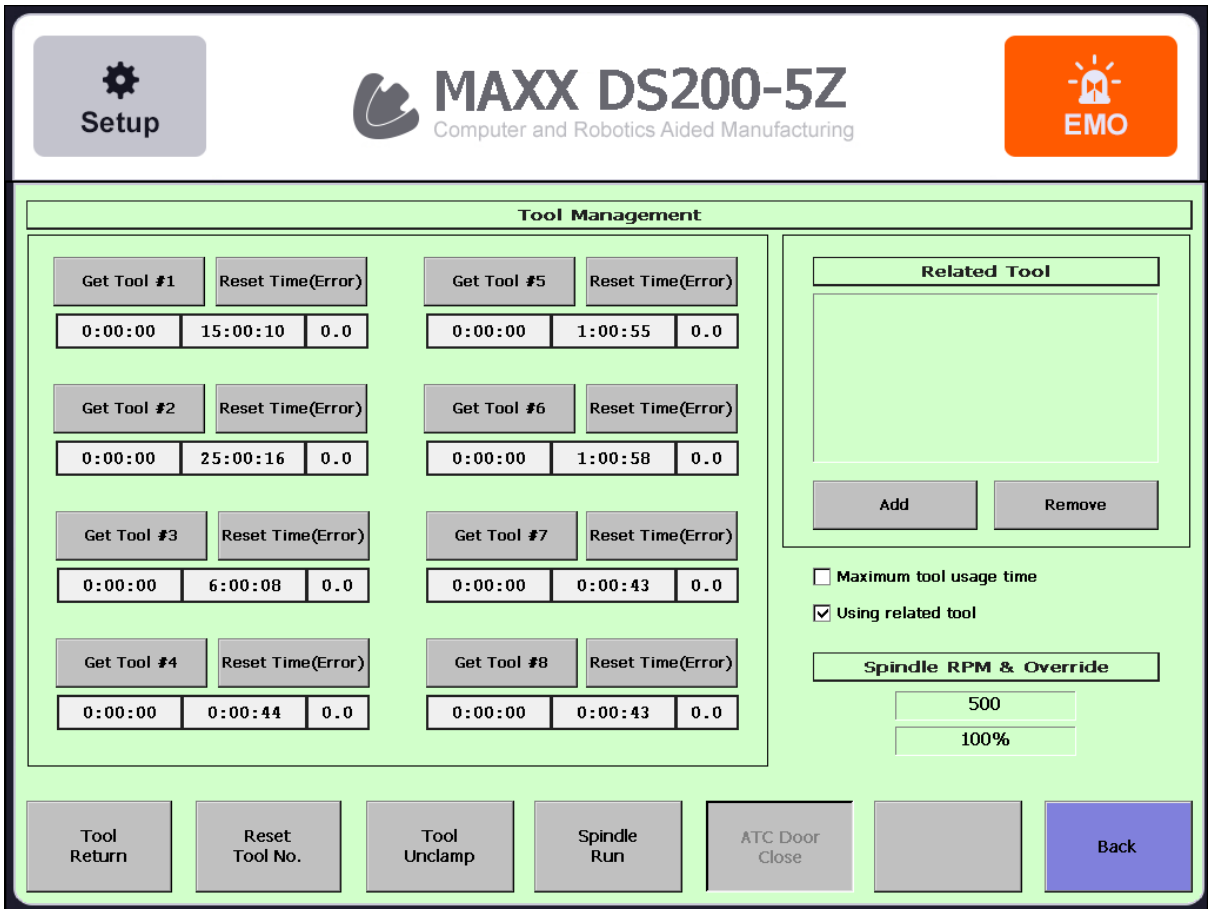
2.3.2.5 Auto Calibration (*Refer to 4.3.4 Auto Calibration)

To calibrate the machine using calibration disc. It takes about 40 minutes to complete automatic calibration.



2.3.3 Tool

To check overall information related tool.



It is to manage tool usage. As shown above, it displays the maximum work time and current work time for each tool in “hour:minute:second” format and its usage in %. If work time for each milling is entered in minutes, it can estimate total number of milling.

When replacing a tool, enter maximum work time and click “reset” to reset usage time to “0:0:0”. “Reset” must be clicked after replacing tool.

Maximum work time is saved in “Config.ini” file in seconds to calculate tool usage.

• Maximum usage of tool

When maximum work time is set to be 100 and maximum usage to be 90, user will receive warning to change the tool after 90 hours.

• Activating maximum usage function of tool

"M710 Macro" and "Tool Change Macro" function can be used when maximum usage function has been activated.

• Counting range of work time

It starts counting work time when position of Z-axis is lower than preset position and spindle is running.

• Related tool

It is to use 2 tool pockets for same sized tool. Enter "M140+M141+" to show that M140 and M141 are the same sized tools so the machine can replace M140 with M141 when its maximum work time is reached.

Instruction

Enter "M140+M141+" to relate tool pocket #1 and #2 at "Edit" and click "Add" button ("+" must be entered at the end).

• Using M710 macro

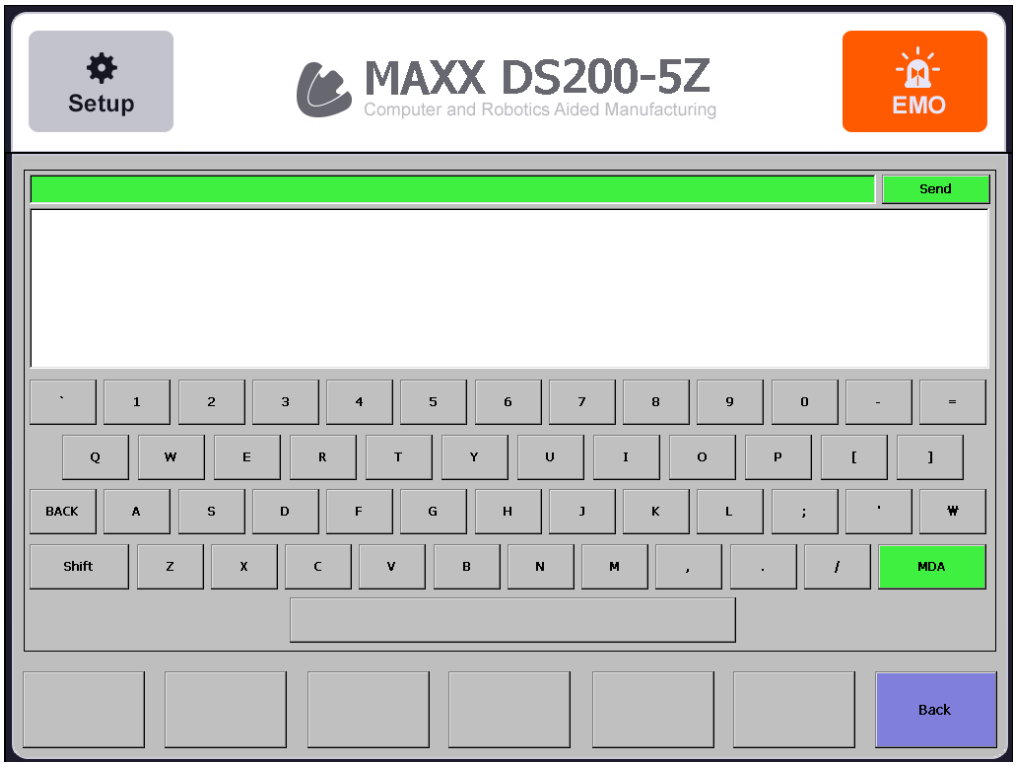
If M710 exists in NC file, it automatically replaces tool with related tool when tool reaches its maximum usage.

• Applying tool change to macro

If NC file order to replace tool and tool is over its maximum usage, it automatically replaces tool with next related tool.

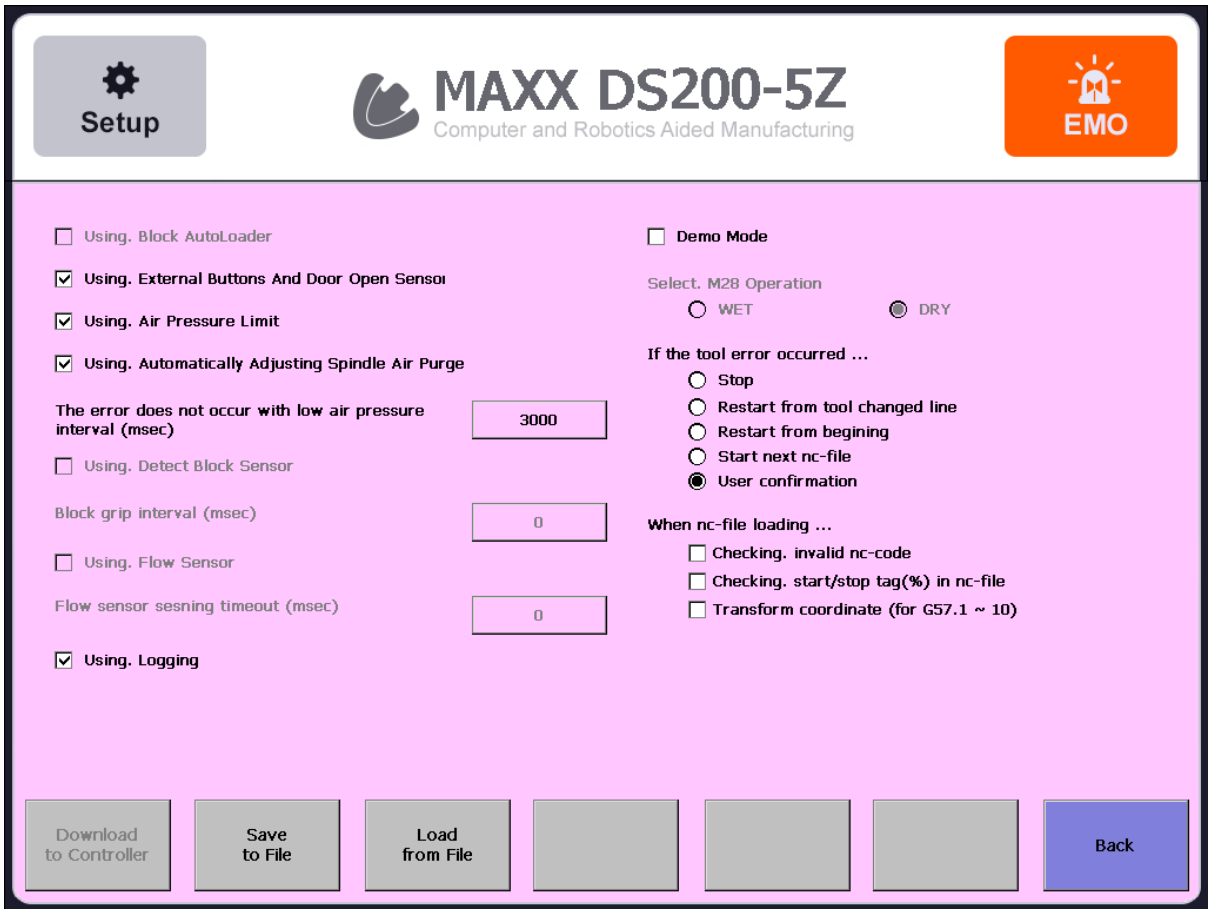
2.3.4 Term

To operate the machine with G-code and M-code.



2.3.5 Option

To activate/deactivate functions of machines.



Option	Description	Remarks
Using. External Buttons And Door Open Sensor	To activate external buttons (DM110 series). To stop milling when door is open.	
Using. Air Pressure Limit	To set air pressure limitation.	
Using Automatically Adjusting Spindle air Purge	To not use purge air when the machine is stopped. It is to control air over-usage.	
The error does not occur with low air pressure interval (msec)	To determine interval time before the machine to recognize shortage of air pressure as an error.	Enter time in msec.
Using. Logging	To records log and error.	
Demo Mode	To start all NC file in the list.	

If the tool error occurred

Option	Description	Remarks
Stop	Stop	
Restart from tool changed line	Restart from tool replacing line before error	
Restart From Beginning	Restart from beginning	
Start next nc-file	Start next NC file	
User confirmation	Require user's confimration before restart of file.	

When NC file loading

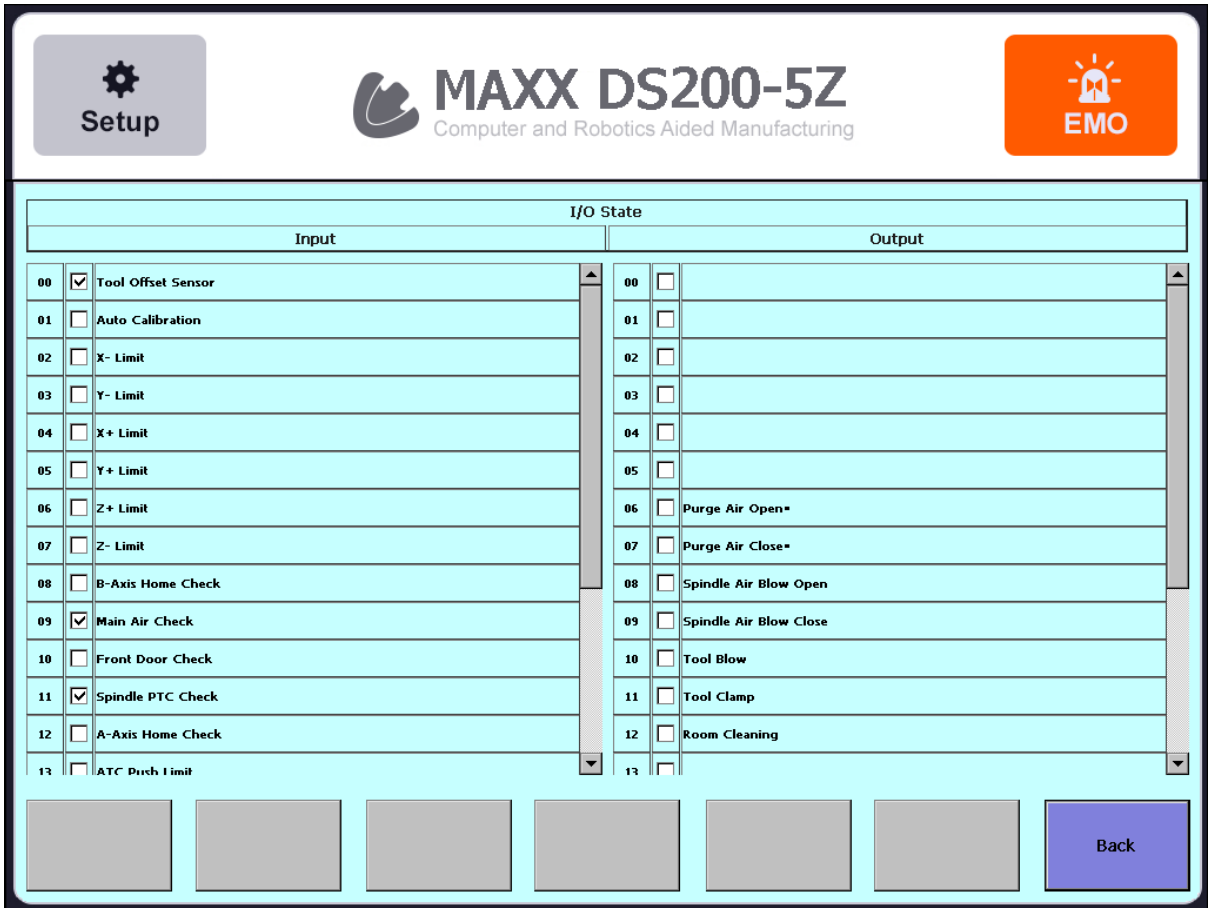
Option	Description	Remarks
Checking. Invalid nc-code	Check NC code error while loading	
Checking. Start/Stop tag(%) in nc-file	Detect the existance of tag(%) at the start and the end of NC file.	
Transform coordinate (for G57.1~10)	Transform coordinates when G57.1~G57.10 exists.	

2.3.6 I/O

I/O (Input / Output unit)

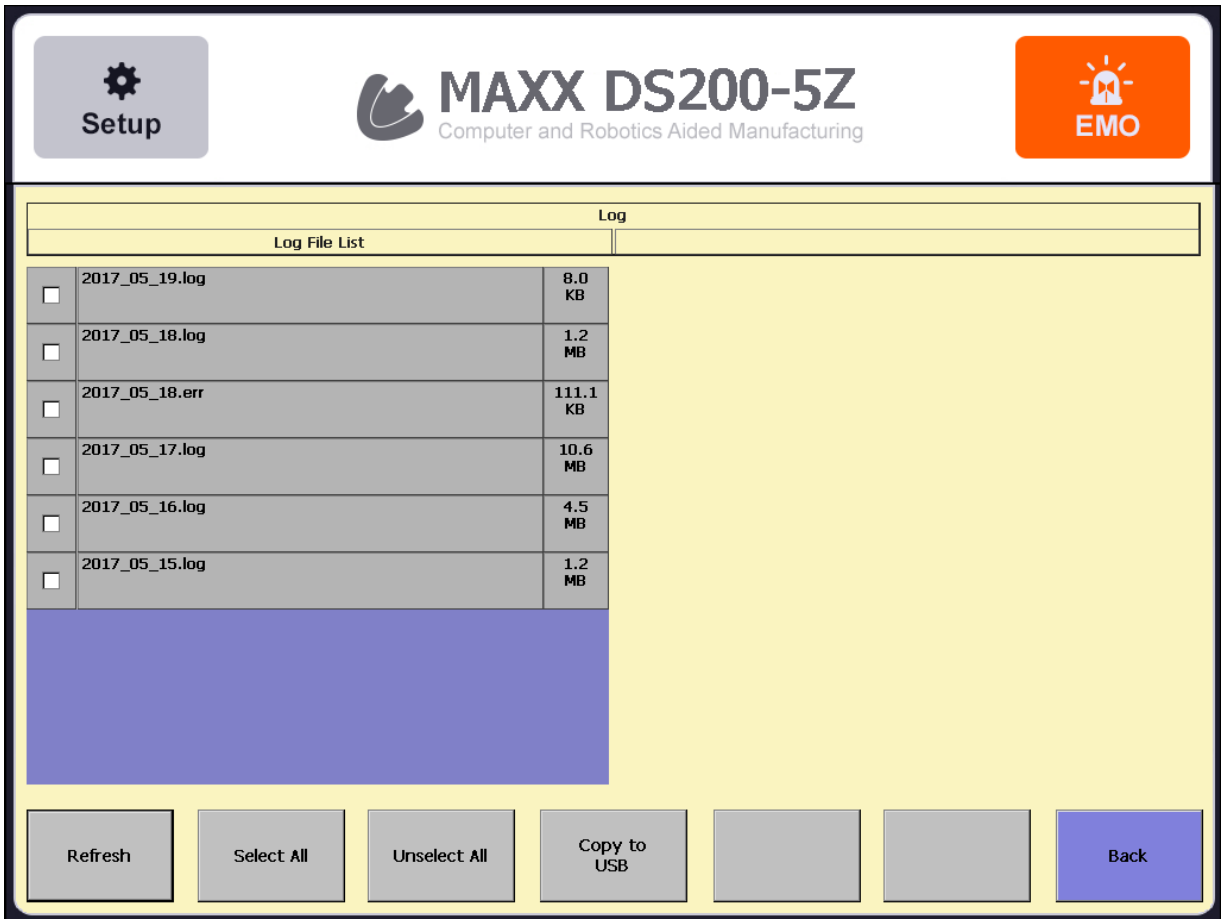
Input : “ v “ appears when sensor is detected.

Output : “ v “ appears when there is port output.



2.3.7 Log

It is records of machine operation. Log is used to investigate error.

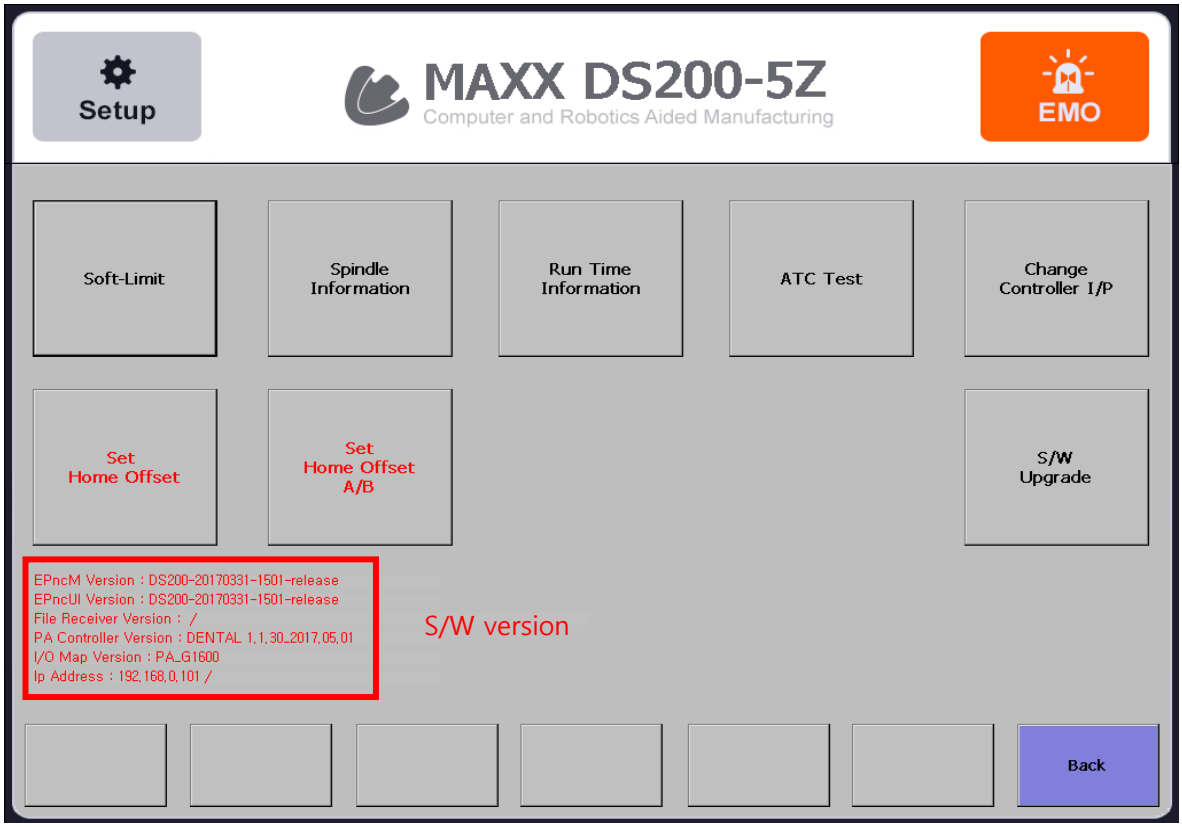


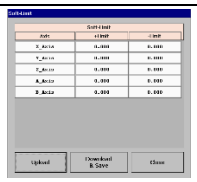
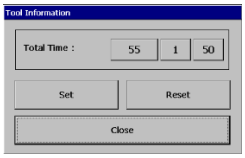
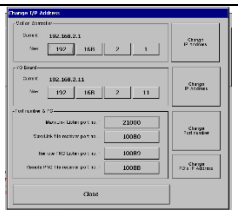
Option	Description	Remarks
Refresh	Refresh log tlist	
Select All	Select all list	
Unselect All	Unselect all list	
Copy to USB	Copy log file to USB	

File type	Description	Remarks
.log	Log file	
.err	Error file	

2.3.8 System

To check general setting and software information.

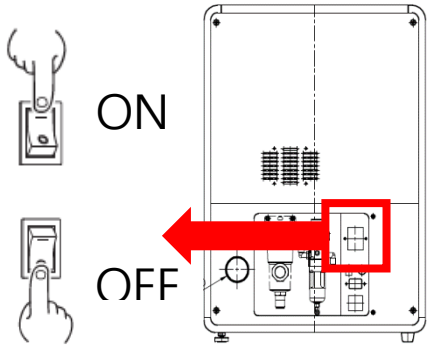


	**Soft-Limit** Limitation for each axis *Do not change without manufacturer's confirmation
	**Spindle Information** Spindle running time
	**Change Controller I/P** To change IP address of controller, I/O board and embedded PC and set Maxx Link port setting.
	**S/W Upgrade** To update EPNC version *Request manufacturer

3. Milling

3.1 Machine power

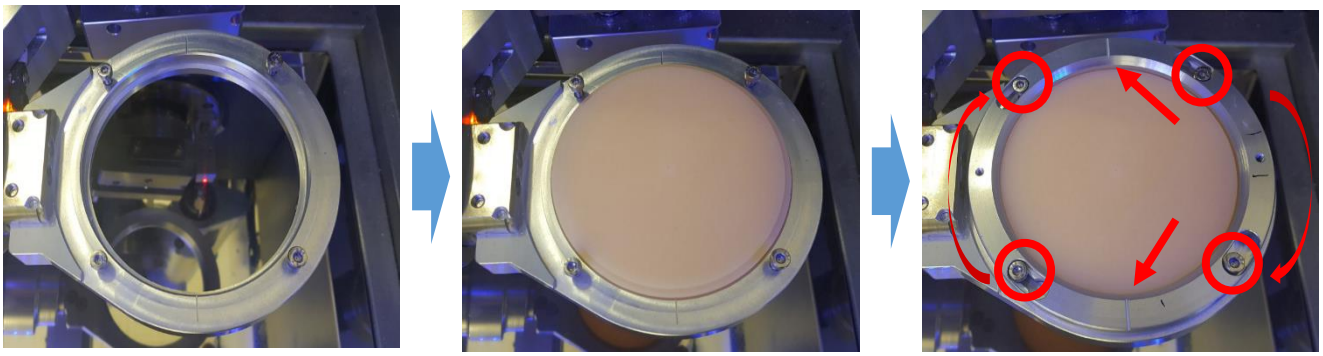
Switch on the machines power switch at the back.



3.2 Milling preparation

3.2.1 Mounting workpiece

3.2.1.1 Disc type (98Φ, step type)

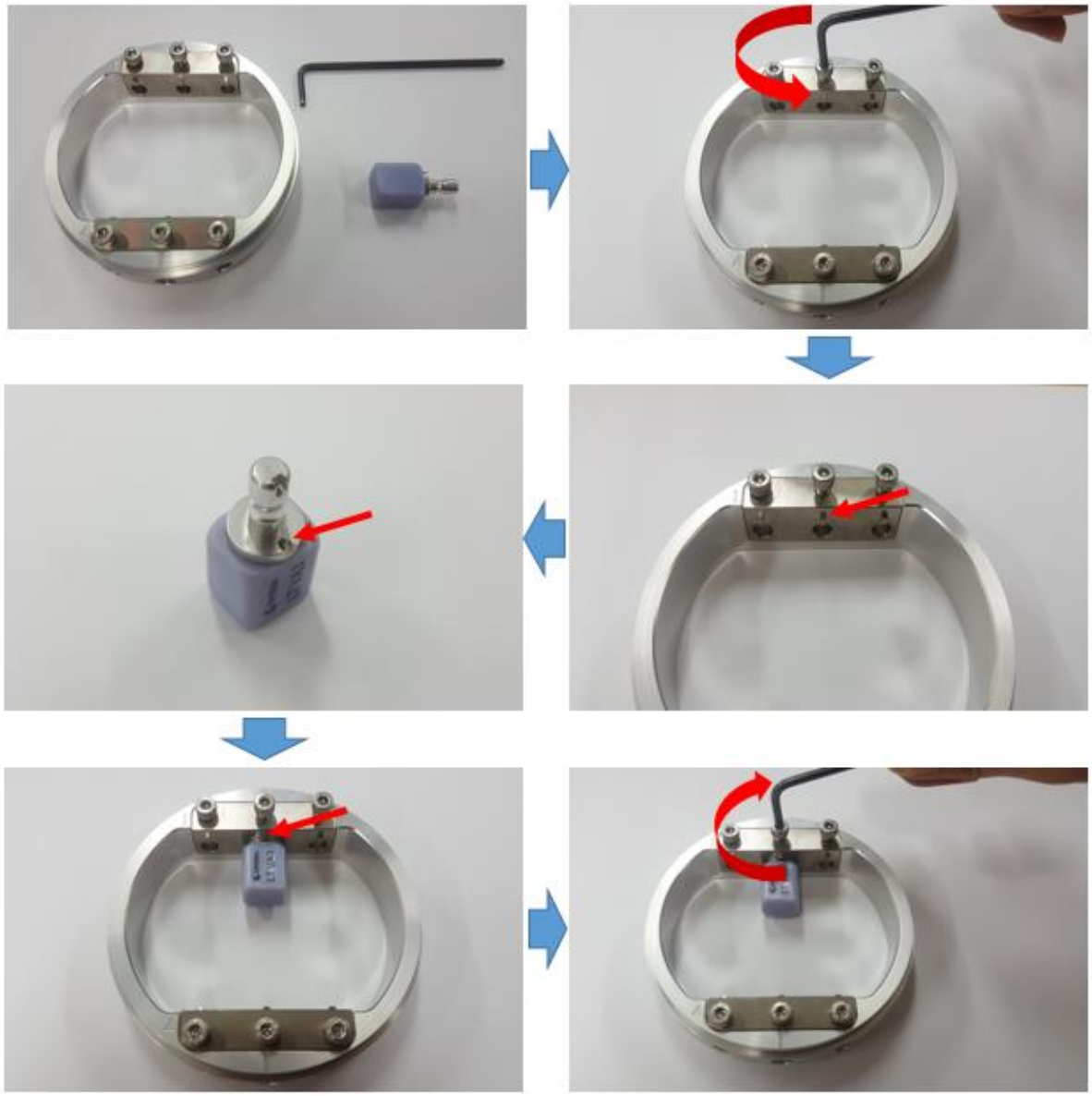


① Mount disc material on disc holder.

② Check the alignment of the workpiece and fix the cover by rotating it in clock wise.

① Screw 4 bots tightly (check if the cover is fixed by attempting to move it with hands).

3.2.1.2 Pin type



① Unscrew bolts of holder (use 3Φ wrench).

② Check female and male of material and put it into a hole.

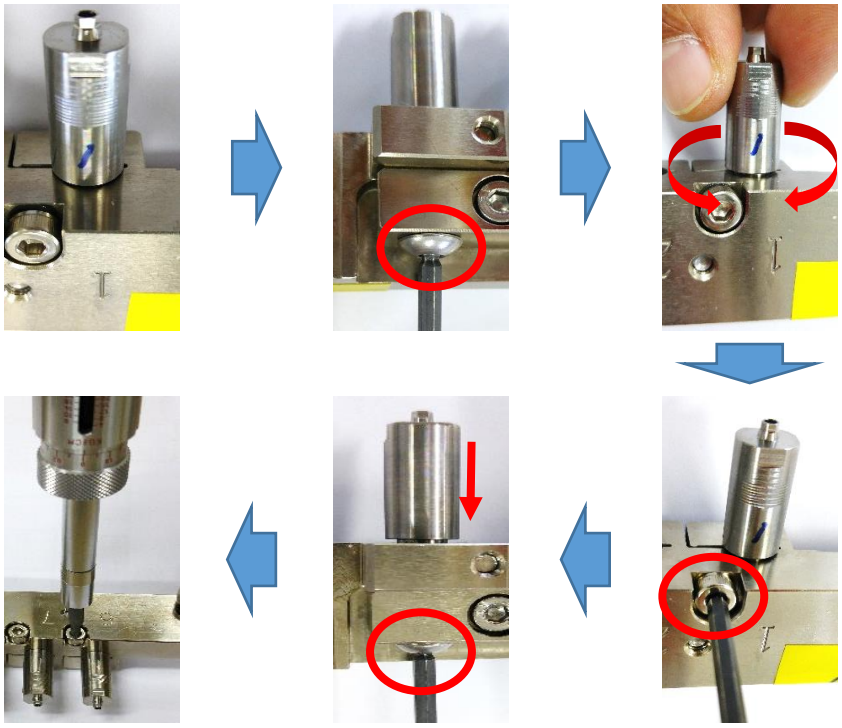
③ Screw a bolt and mount it to DS200 jig.

④ Check if it is mounted correctly and screw 2 bolts tightly.

3.2.1.3 Pre-milled bar (R & D type)

D Type

D-type does not require extra jig.



① Insert a round bar to the desired section of the jig.

② Screw the rear pulling bolt to hold the round bar. However, leave around 1mm of room between the jig and the bar.

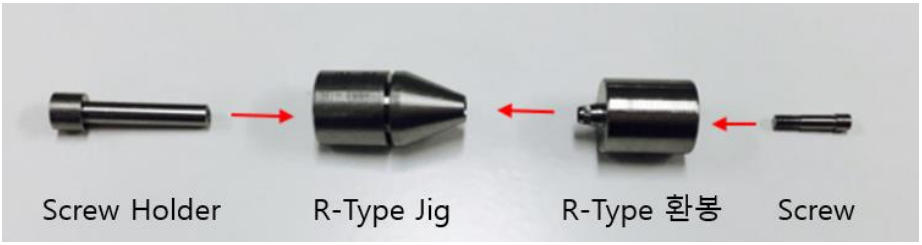
③ Try rotating and pulling the bar to see if the bar is properly attached to the jig.

④ Screw the upper clamping bolt again with the torque wrench. (set the torque wrench to 26kgf-cm)

⑤ Screw the rear pulling bolt until the round bar is completely touching the jig using the torque wrench with the same configuration (caution: Do not over screw the bolt, in which case will hurt the bolt).

R Type

R Type takes more time to be mounted and to select jig. However, it can create more precise result without connector and short post-cleaning process.



① Tighten the screw using the same torque wrench until the round bar does not move.

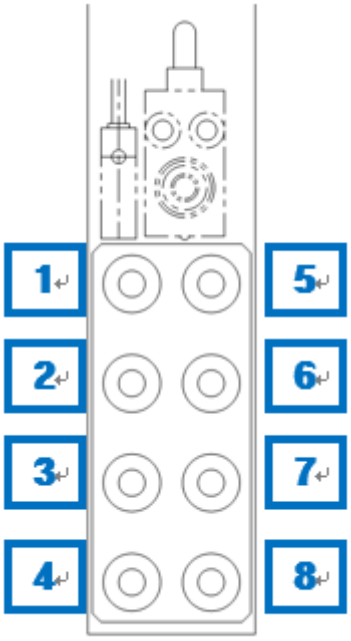
② Insert the assembled workpiece to the jig the same way as the D-type bar (described above).

* Do not leave any room for the workpiece to freely move

* There is no need to use the rear pulling bolt.

3.2.2 Tool

Tool number and M-code is shown as below.



Tool #	M-Code
#1	M140
#2	M141
#3	M142
#4	M143
#5	M144
#6	M145
#7	M146
#8	M147

3.2.2.1 Tool specification

Please refer to tool specification for DS200 series as below.

It is highly recommended to use tools provided from manufacturer otherwise milling quality cannot be guaranteed.

	Coating	Shank	Diameter	Milling length	Total length
ZIRCONIA	DIA	3mm	2mm	20mm	45mm
	DIA		1mm	18mm	
	DIA		0.6mm	12mm	
	NON		0.3mm	2mm	
PMMA, WAX	DIA	3mm	2mm	20mm	45mm
	DIA		1mm	18mm	
	DIA		0.6mm	12mm	
GLASS CERAMIC	DIA	3mm	1mm	14mm	40mm
	DIA		1mm	10mm	
METAL (4W)	TiAlN+CrN	3mm	3mm	15mm	45mm
	TiAlN+CrN		2mm	12mm	
	TiAlN+CrN		1.5mm	10mm	
METAL (4WA)	TiAlN+CrN	6mm	3mm	12mm	50mm
	TiAlN+CrN		2mm	12mm	
	TiAlN+CrN		1.5mm	10mm	

3.2.3 Starting NC file

* Refer to “USER GUIDE” [2. Description of EPNC] > [2.2.1.4 NC file]

4. Maintenance

4.1 Maintenance precautions

<Caution>

Be careful around the milling tool. The milling tool is sharp.

Broken milling tool is dangerous. Use cautiously to avoid injury.

- ✓ This machine is a precision instrument. Perform day-to-day management and maintenance.
- ✓ Carefully clean the milling wastes away. If the machine operates in the presence milling waste it may cause a malfunction.
- ✓ Do not lubricate any location that is not specified in this manual.

4.2 Daily maintenance

4.2.1 Cleaning

1. **Do not use the air blower.**

   This machine is not compatible with an air blower. Milling waste in the machine can cause fire or an electrical accident.

2. **Do not use solvents such as gasoline, alcohol, thinner, in order to perform a warning to clean.**

   Doing so may cause a fire.

3. **Do not use a vacuum cleaner to clean the warning milling waste.**

   General cleaning dust using a vacuum cleaner can cause a fire or explosion. Warning

4. **Do not touch the spindle immediately after milling and the surrounding area has been shut down.**

   Doing so can cause burns.

Must perform below maintenance before and after operating the machine

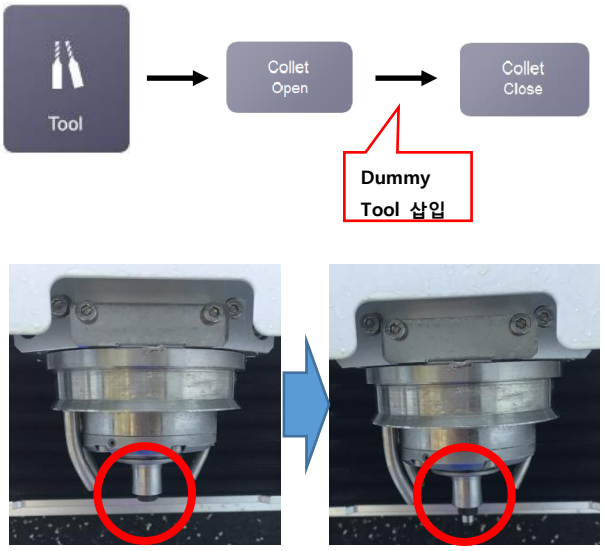
Interval	Scope of maintenance
Before milling	Check if 8 tools are inserted in the right order in tool holder.
Daily	Coolant Tank; Check the filling level using the min / max marking.
	Wipe out the processing chamber at the end of the working day.
	Clean tool pocket
	Clean spindle collet
	Clean the Spindle collet, ATC Tool holder

4.2.2 Dummy tool

Preserve the machine with dummy tool inserted.

AIM

- ✓ To prevent oil left in spindle collet to become solid
- ✓ To prevent particles to go inside spindle.



How to insert dummy tool

① Click “Tool” > "Collet Open”

② Insert tool with hand (hold dummy tool as it may fall out by air)

③ Click “Collet Close”

4.3 Periodic maintenance

Periodic maintenance to ensure full performance of the machine.

Interval	Scope of maintenance
Weekly	Check coolant nozzles and clean
	Check filter regulators and clean
	Check that the suction bellows are working properly.
	Clean the Spindle collet in accordance with these Operating procedures.
	Change the coolant.
	Change the coolant filter.
Monthly	Check the condensate tank on the filter regulator and drain the condensate if necessary.
	Clean the outer surfaces of the machine.

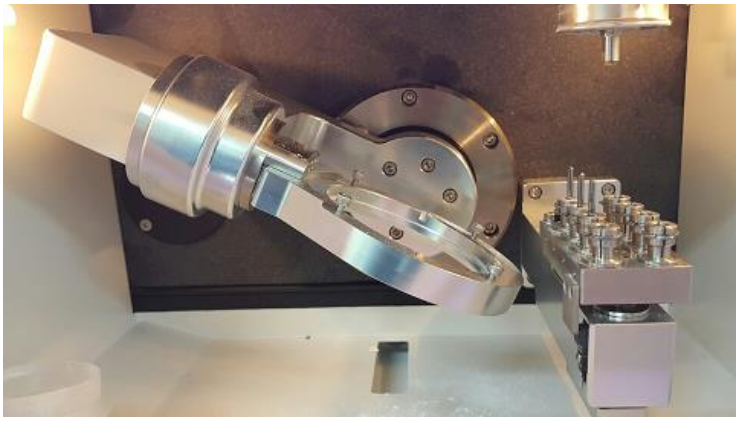
4.3.1 Replacing consumable parts

Replacing tool pockets

*Replace tool pocket when:

- ✓ Inclination occurs with tool stored inside
- ✓ Tool pocket becomes loose
- ✓ Push Limit Error occurs frequently even though tool teaching is performed

① Position the machine as below (move "-" A axis).



② Unscrew set-screw using 2mm wrench.

③ Replace tool pocket and tighten it until it does not move.

4.3.2 Spindle warm-up

To stabilize the spin of spindle, spindle warm-up is needed.

When to perform spindle warm-up

- ✓ After installing the machine
- ✓ When the spindle units are replaced.
- ✓ When the machine was not used for a prolonged period
- ✓ Before using the machine in low room temperature

Check before performing spindle warm-up

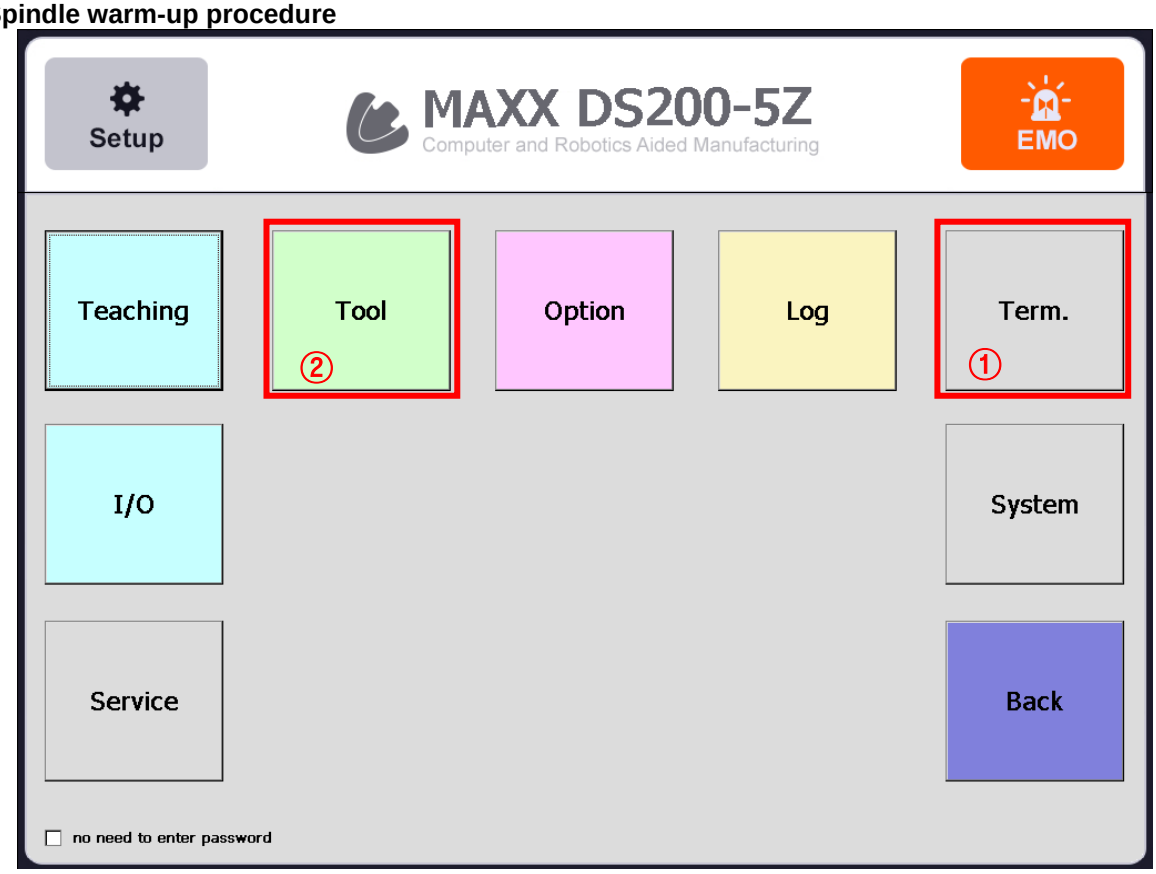
- ✓ Check if spindle RPM & Override value is changed correctly.
- ✓ Tool must be clamped in collet (perform “Get tool #1”).

**When there is no tool in ATC:

“Tool unclamp” > put tool into spindle collet manually > “Tool clamp”

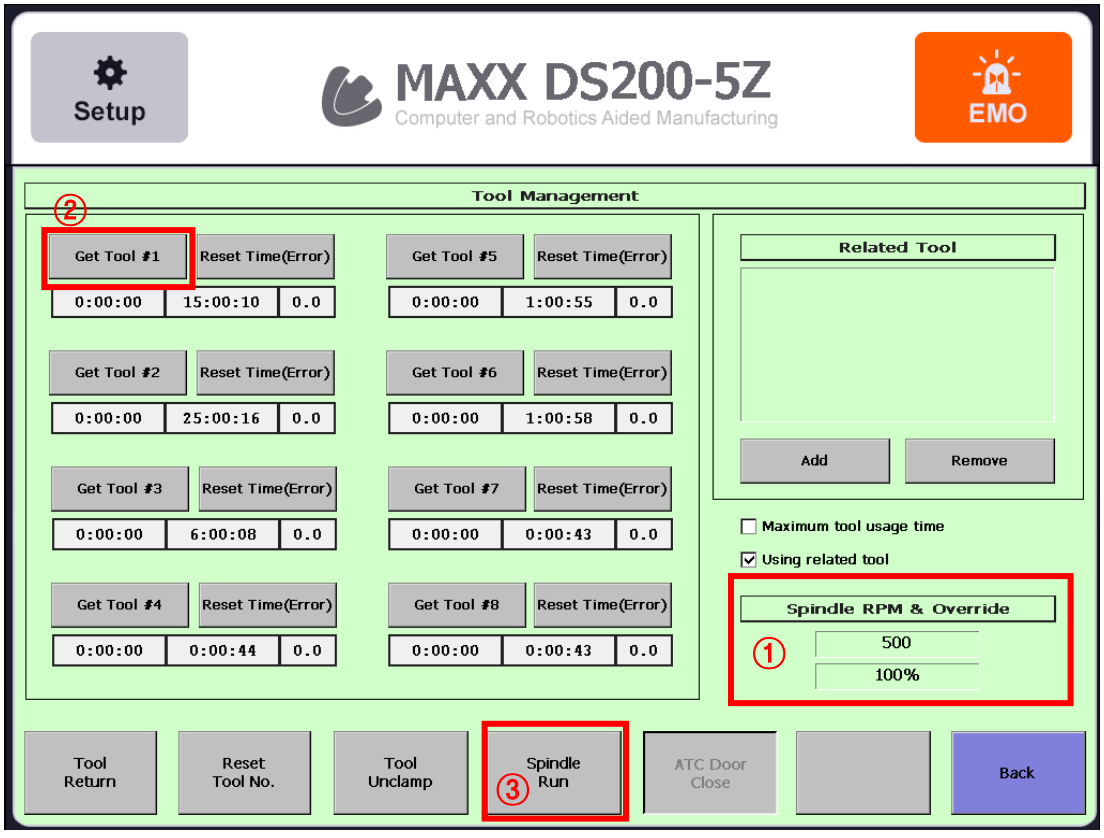
- ✓ Perform spindle warm-up by following below schedule

Spindle warm-up procedure



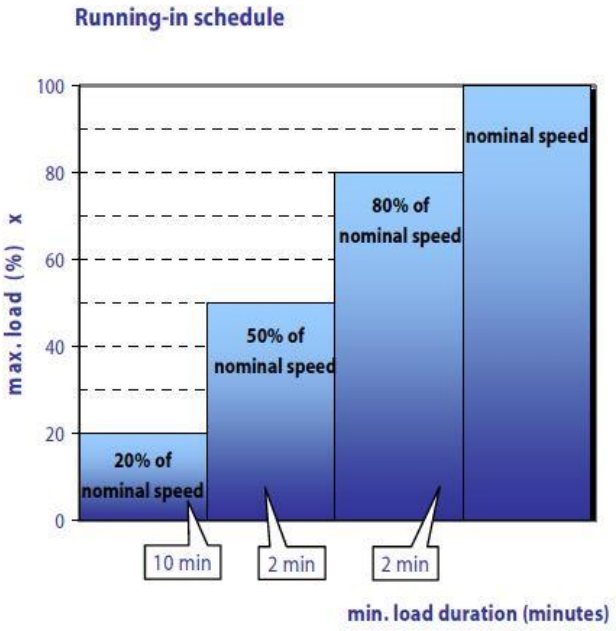
1. Click “Term”

2. Enter RPM and click “SEND”



3. Check RPM in “Tool” and click “Get Tool #1” then “Spindle Run”

Spindle warm-up schedule



① Commission the spindle by running it with a tool fitted (no machining) for approximately 10 minutes.

② The maximum speed is 20% of the rated speed of the spindle.

③ Allow the spindle to run for approximately 2 minutes at a maximum of 50% of its rated speed.

④ Operate the spindle for around a further 2 minutes at a maximum of 80% of its rated speed.

⑤

The spindle is now ready for operation.

Never operate the spindle without a shank this can result in:

- ✓ the clamping system being damaged by the centrifugal forces
- ✓ the clamping system becoming displaced
- ✓ the balance of the spindle being adversely affected and the bearing will be damaged.
- ✓

WARNING

Observe the maximum speed of the selected tool specified by the tool supplier and the maximum speed of the spindle. Always select the lowest specified speed for machining. Selecting the wrong speed could destroy the spindle or the tool and the fragments could be thrown outwards. This can result in serious injury to the whole body or death.

Daily startup

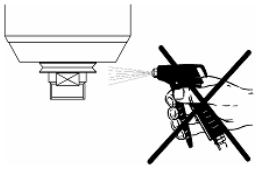
When starting up each day, we recommend operating the spindle for approximately 2 minutes with a tool fitted (no machining) at a maximum of 50% of the maximum speed to pre-heat and preserve the grease lubrication on the bearings. This brings the spindle up to its operating temperature.

4.3.3 Spindle cleaning

Daily cleaning and maintenance

To ensure that the spindle functions safety and accurately, all contact surfaces of the spindle, the mount for the spindle, the tool mount and the tool holder must be clean.

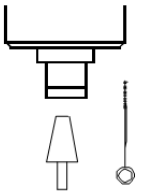
Before beginning work each day, check that all surfaces have been thoroughly cleaned and are free of dust, grease, coolant, residue from precious operations and metal particles and that there is no visible damage.



Do not use compressed air, ultrasound or steam jets to clean the spindle. This can allow dirt to get into the bearings.

If the spindle is equipped with sealing air, always turn this on when cleaning.

Use only a clean, soft cloth or a clean, soft brush for cleaning.



Clean the inner taper of the spindle shaft. The inner taper must be free of chips and impurities.

Clean the tool taper with a clean, soft cloth or a clean, soft brush.

Clean the collet and the collet mount with a clean, soft brush.

Apply a light film of grease to the taper of the collet after cleaning. This improves the slippage and increases the clamping force of the collet.

! Only use the collet grease from the service set. No other lubricating grease or oil may be used.

Checking Collet Gap

① Check if collet automatically fall out when it is open

② If collet does not come out smoothly, separate collet and clean and grease taper of collet and spindle before re-assemble.

③ Check purge air supply: check and adjust to 1.0 ~ 1.5 bar

4.3.4 Auto Calibration

Aim of calibration

Process to match coordinates of milling machine and operation coordinates.

Type of calibration

Auto calibration	Measure origin position using 98Φ Calibration Disk
Multi Origin Auto Teaching	Measure D-Type abutment coordinates (3)
	Measure R-Type abutment coordinates (3)
ATC Auto teaching	Measure ATC Tool Pocket coordinates

1. Auto Calibration

① Install calibration disc

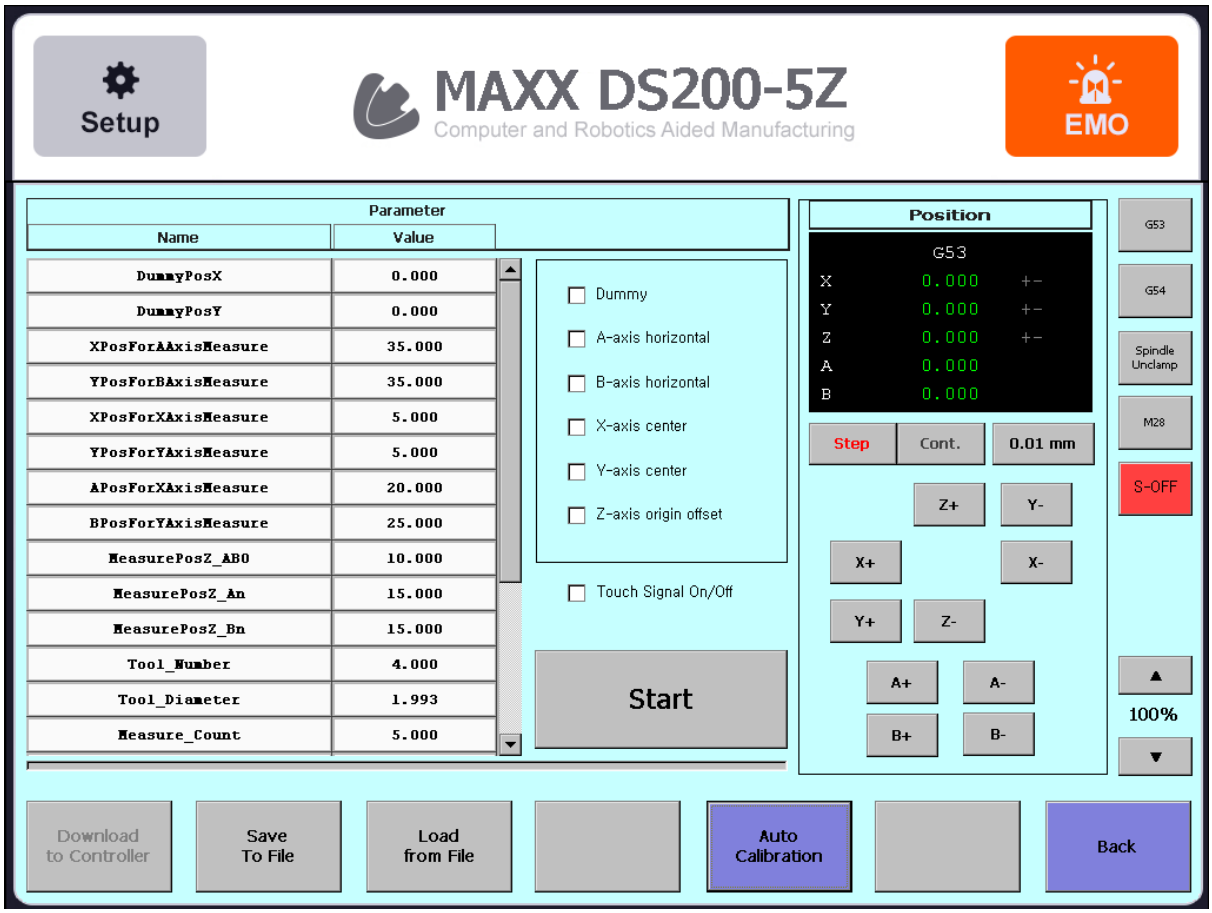
Check thickness of calibration disc before installation and insert it to parameter

② Put calibration tool

ATC Door Open: MAIN WINDOW > TOOL > ATC OPEN

Open ATC from “Setup” then install at entered tool pocket position in parameter.

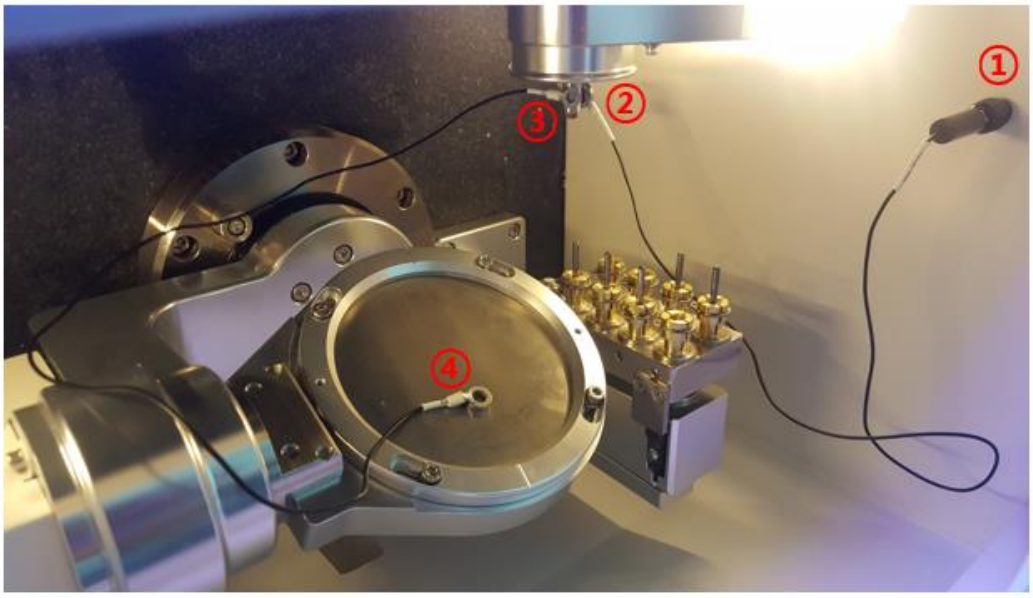
③ Auto Calibration: Setup > Teaching > Auto Calibration



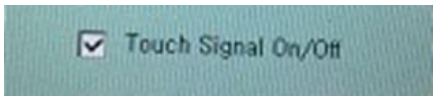
Check before start

- ✓ Disk Thickness: enter calibration disk thickness
- ✓ Tool_Diameter: enter calibration tool diameter
- ✓ Too_Number: enter tool pocket number for calibration tool

  **Click Name and double click on value to enter number
- ✓ Click “Save to File” to save the data.
- ✓ Test connection (refer to next page)



After connecting calibration cable, “V” will appear at touch signal on/off as below.



“V” should only appear when calibration cable magnet is touching calibration disc.

④ “Calibration completed” window will appear when it is completed.

- ✓ Setup > Teaching > Coordinate offset
- ✓ Click “Download to Controller” and “Save to file” to save changed coordinates.

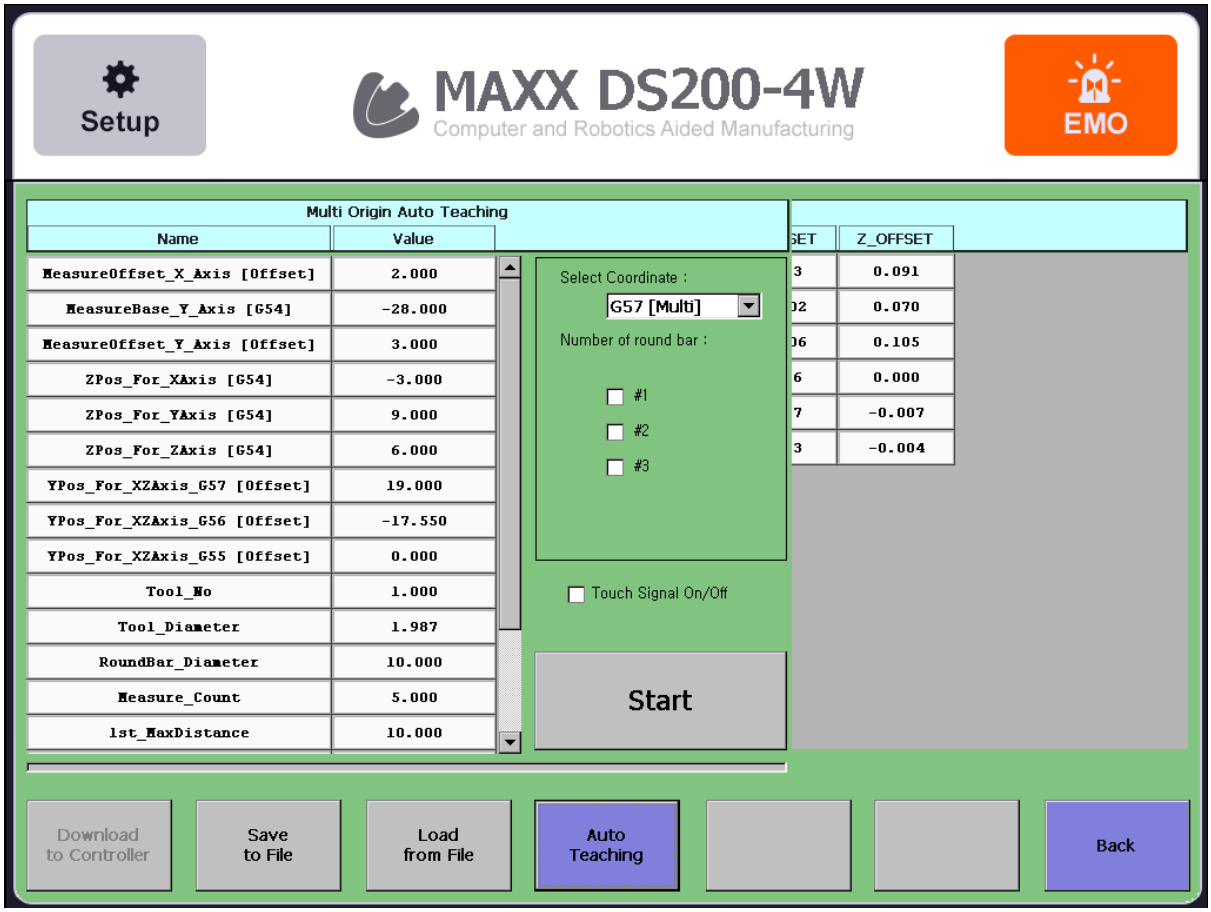
2. Multi Origin Auto Teaching (*DS200-4W / 4WA)

<D-type>

① Mount D-type pre-milled bar *refer to [3.2.1.3 Pre-milled bar]

② Put calibration tool *refer to [4.3.4 Auto calibration]

③ Auto Teaching: Setup > Multi Origin > Auto Teaching

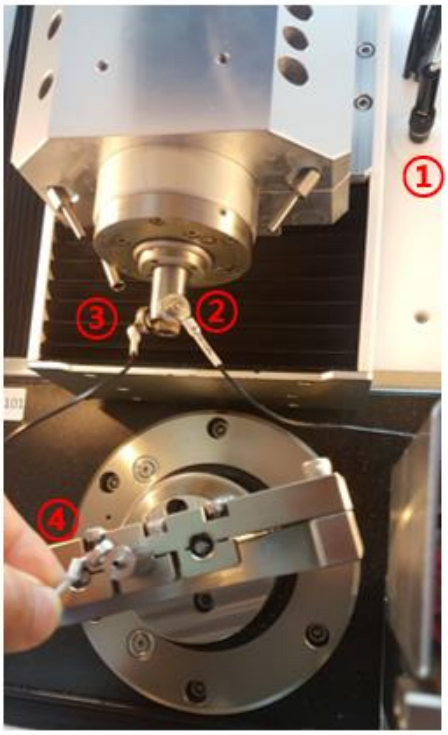


④ Start menu

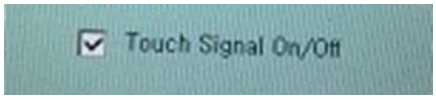
Select coordinate	D-type uses G57.
Number of round bar	Select pre-milled bar position. D-type can use up to 5 positions.
Touch signal on/off	When calibration tool touches pre-milled bar, signal “v” will appear.

Check before start

- ✓ D-type uses G57 [Multi]
- ✓ Check calibration position #1~#5
- ✓ Disk_Thickness: enter calibration disc thickness
- ✓ Tool_Diameter: enter calibration tool diameter
- ✓ Tool_Number: enter tool pocket number for calibration tool
  **Click Name and double click on value to enter number
- ✓ Click “Save to File” to save the data
- ✓ Test connection as below



After connecting calibration cable, “V” will appear at touch signal on/off as below.



“V” should only appear when calibration cable magnet is touching calibration disc.

⑤ Remove ③ and ④ and click “Start”.

⑥ “Calibration completed” window will appear when it is completed.

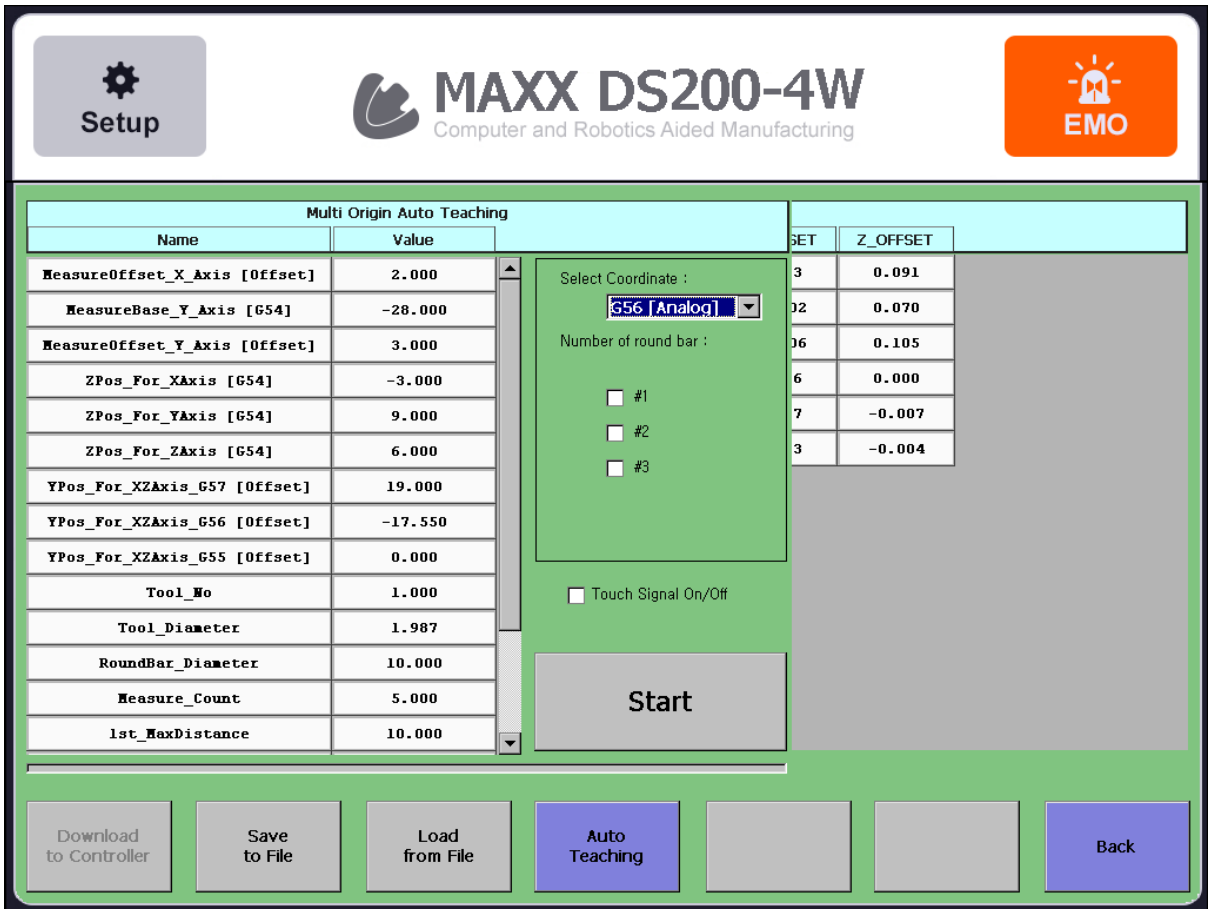
- ✓ Setup > Multi Origin
- ✓ Click “Download to Controller” and “Save to file” to save changed coordinates.

<R-type>

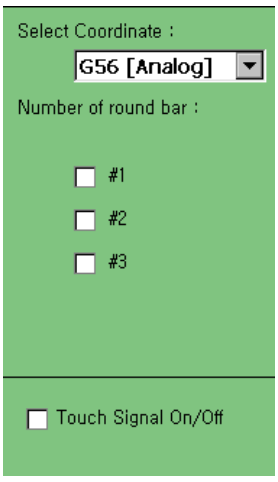
① Mount R-type pre-milled bar *refer to [3.2.1.3 Pre-milled bar]

② Put calibration tool*refer to [Auto Calibration]

③ Auto Teaching: Setup > Multi Origin > Auto Teaching



④ Start menu



Select coordinate	R-type uses G56 [Analog]
Number of round bar	Select pre-milled bar position. R-type can use up to 3 positions.
Touch signal on/off	When calibration tool touches pre-milled bar, signal “v” will appear.

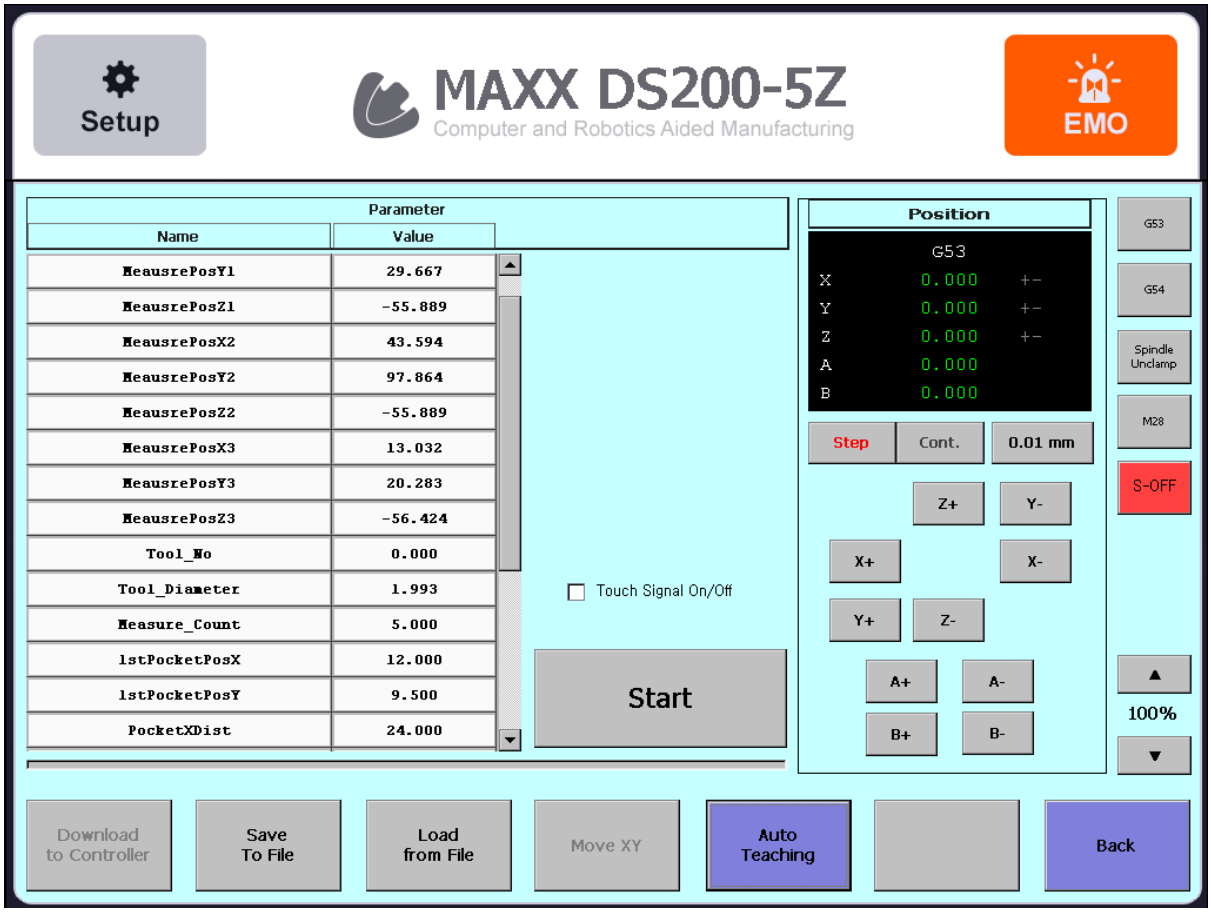
⑤ Refer to <D-type> for next procedure.

3. ATC Auto Teaching

To automatically perform teaching of tool pocket X and Y coordinates.

Setup > Teaching > Teaching Point > Auto Teaching

① Auto teaching



Check before start

- ✓ Tool_Diameter: enter calibration tool diameter
- ✓ Tool_Number: enter tool pocket number for calibration tool
  **Click Name and double click on value to enter number
- ✓ Click “Save to File” to save the data
- ✓ Test connection (*refer to Disk calibration)

② “Calibration completed” window will appear when it is completed.

- ✓ Setup > Teaching > Teaching point
- ✓ Click “Download to Controller” and “Save to file” to save changed coordinates.

4.3.5 Collet tightening

Milling will cause the collet to become loose, making milling bur to fall out easily.

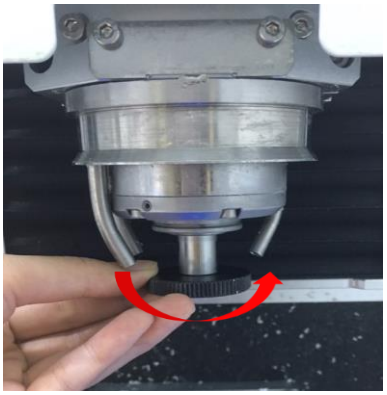
Collet must be retightened periodically.

Interval

Once a month, or when the total work time of the spindle exceeds 200 hours (with slight variation depending on the work situation)

Procedure *refer to 4.2.2 Dummy Tool

After installing milling tool, use spanner (spindle maintenance kit) to tighten the collet in anti-clockwise.

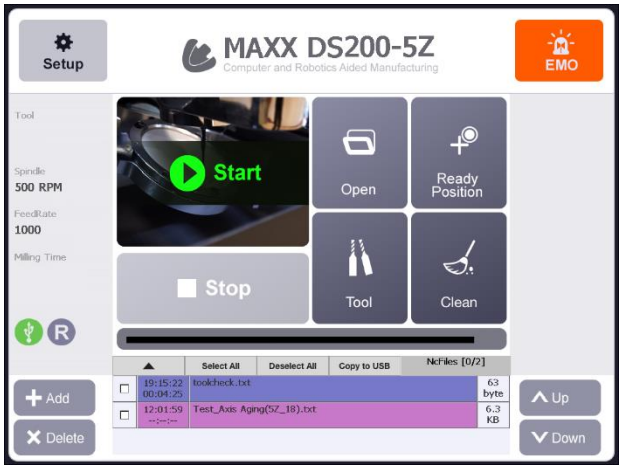


5. Trouble shooting

5.1 Restart milling

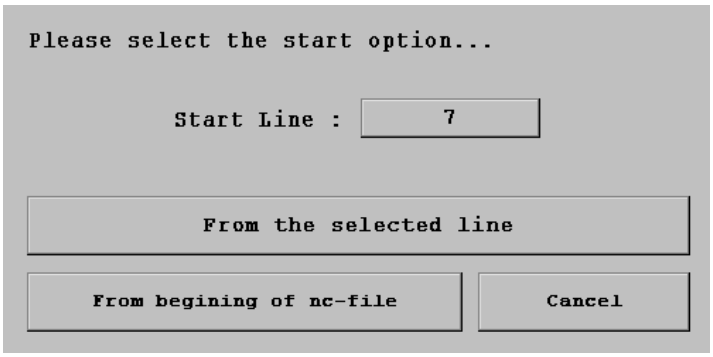
How to restart the machine when it was stopped due to an error

① Stopped during milling



② After error reset, click “START”

③ A window requesting for restart method appears



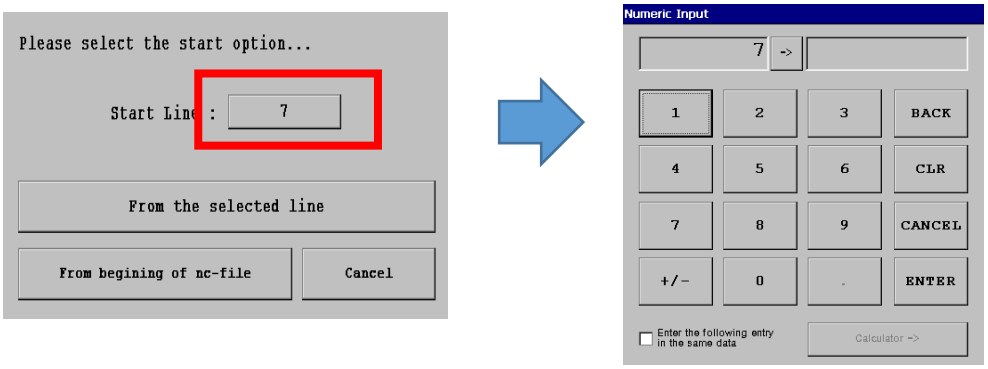
Start Line	Selected NC-file Ex) 7 = 7th NC code line no.
From the selected line	Re-start from start line
From Beginning of nc-file	Re-start NC file from the beginning
Cancel	Close the window

***Restarting when tool broken error occurs during milling**

For example) After milling Φ2, the machine successfully performed tool change from Φ2 to Φ1 but Φ 1 break during milling and "tool broken" error occurs at the return of the tool.

☞ When you click on "from the selected line", milling will restart from recent tool change line.

* How to change "start line": change line number by entering number



*Finding start line

① Open NC file with notepad

② View > Status bar (shows current prompter position)

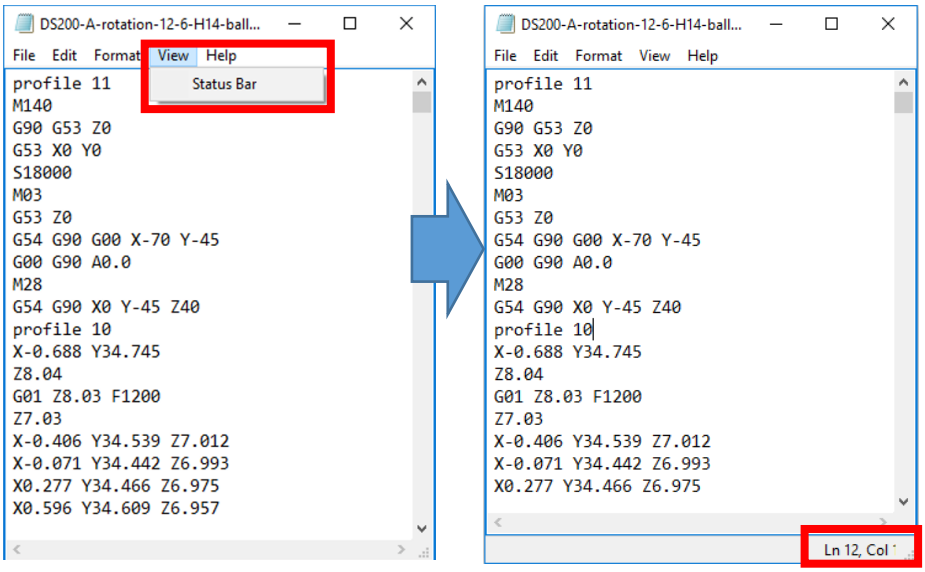
③ Edit > find

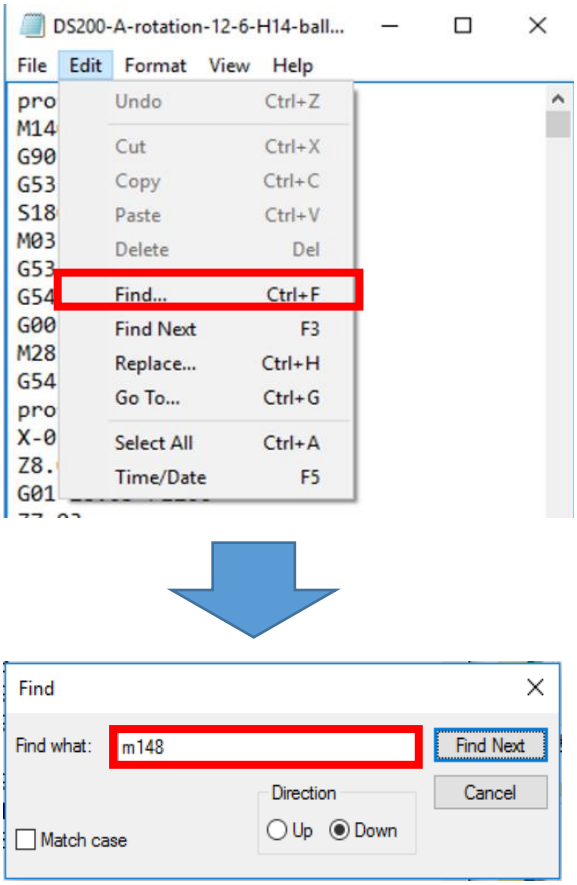
**Tool select and pickup macro: #1 = M140, #2 = M141, … , #8 = M147

④ Search M141

⑤ NC code for #2 replacement is 13346

⑥ Enter 13364 at selected line





5.2 Air pressure sensor setting

Please follow below instruction to pneumatic sensor.

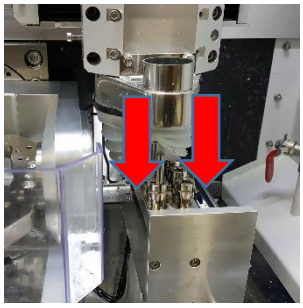
	① Press S for 3 seconds.
	② Press △ once.
	③ Press S three times.
	④ Set value to 0.500 by pressing △ or ▽

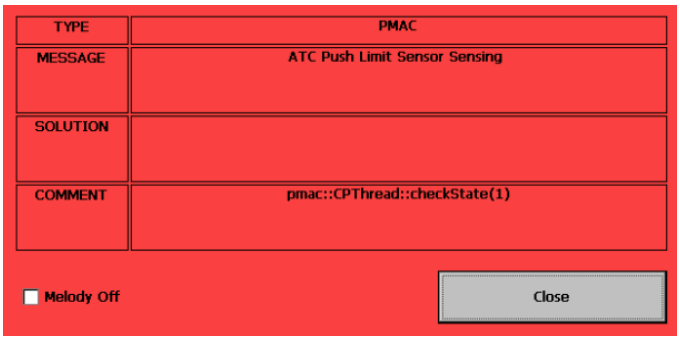
	⑤ Press  once to move to next step.
	⑥ Set value to 0.000 by pressing  or 
	⑦ Press  twice to go back to ③. ⑧ Press  for 3 seconds to go back to initial screen ①.

*SMC – Model: ISE30A

5.3 Removing ATC push limit

When performing Tool Get and Put, an error can occur due to many reasons that makes spindle to push ATC.





① Close Error message

② Move spindle manual that is pushing ATC

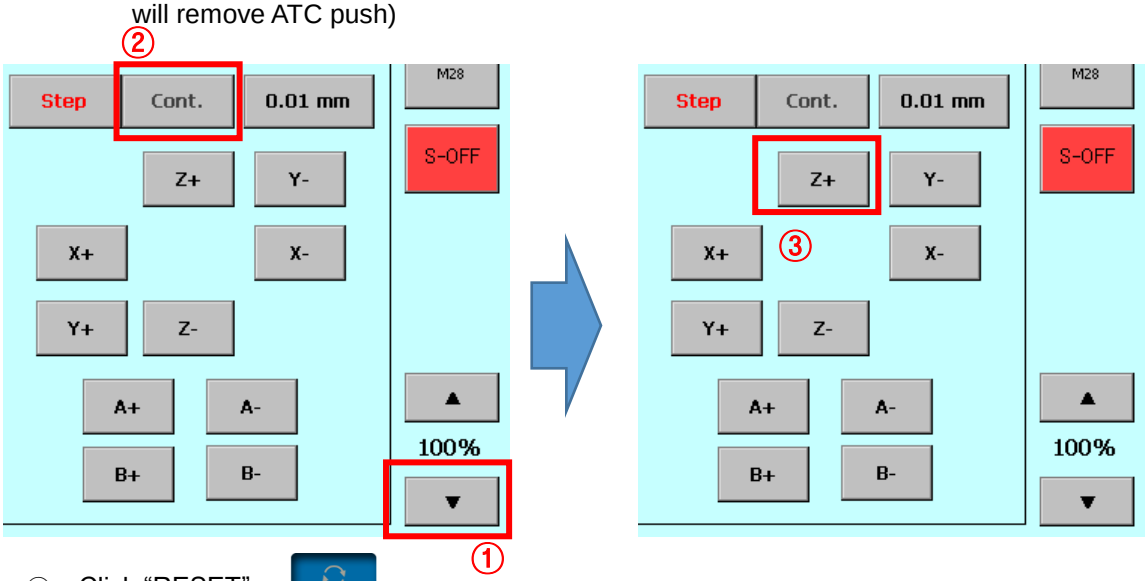
**How to move Z-axis (vertical direction of spindle)

Setup > Teaching > Jog

A. Change axis moving speed 100% > 50%

B. Change Step > Cont.

C. Click Z+ to move spindle (continue to check and move spindle only for the amount that will remove ATC push)



③ Click “RESET” 

④ Check collet

⑤ If there is a tool clamped, perform “RETURN TOOL” 

If there is no tool clamped, perform “READY POSITION”