Roland MDX-540

CNC Mill
Roland MDX-540
Roland MDX-540

Facts

- 500mm (X) x 400 (Y) x 155 (Z) travel
- 550 mm x 420 mm table size
- Maximum of 20kg workpiece
- 0.001mm mechanical resolution
- 0.05mm repeated accuracy
- 400 – 12000 RPM spindle speed
- Automatic tool change (4 stock places)
- Optional 4\textsuperscript{th} axis
Safety Rules / make sure !!!

- Make sure your workpiece is secured
- Do not change tools when spindle is moving
- Never touch moving spindle
- Always keep your hands away from the moving spindle
- No towels near moving parts
- No measuring or marking when machine is running
- Only clean the machine when turned off
- Never change the safety systems
Safety Rules

- Safety goggles
- No baggy clothing
- No open hair
- No jewellery
  - Including watches and rings
- No gloves
Terms

Tools used in CNC Milling
Terms

Machine

4th Axis
Table
Safety off
On/off Switch
Tool changer
Z-Sensor
Handreel

Roland MDX-540
**Terms**

Workpiece holding

- Bold down bolt
- Step block
- Hold down plate
- T-Nut
- Vise
Terms

Tool holding

- Shank
- Shank nut
- Tool
- Collet
Terms

The on switch can be found at the front panel of the machine.
Tools

Tools used in CNC Milling
Flat or square end mill

- Used for cutting flat or stepped items
- Prone to fail due to very small and brittle corners
Ball end mill

- Used for cutting curved surfaces and wavy shapes
- Most common used
- Usually stronger than flat end mills
Engraving Tool

- Used for engraving
Flutes
Cutting etches

Flute
Tool Limitations

- **End Mill Limitations**
  - Can’t mill hole or feature smaller than tool diameter
  - A large mill will leave corners unmilled
    - Use smaller diameter tools for reaching corners
  - Can’t mill a wall taller than length of tools reach.
    - Need longer fluted tool or long reach tool
Tool Holding

How to securely hold your tools
Tool holding

Tools

Wrench (19)

Spanner wrench
Tool holding

To open the tool holder put the spanner wrench with your left hand and secure the shank with the wrench in your right. Then rotate the wrenches towards each other.
Tool holding

Insert the collet in the shank nut. Make sure the collet is correctly placed.

Insert collet into nut

- Make sure its well seated

Engage the tabs securely.
Tool holding

Insert the tool as far as possible in the collet without inserting the cutting surface.
Tool holding

To secure the tool holder put the spanner wrench with your right hand and secure the shank with the wrench in your left. Then rotate the wrenches towards each other.
Tool holding tips

Select the shortest and largest diameter tool for better rigidity. Also use a multi flute tool for greater rigidity and better finishing.
Workpiece Holding

How to secure your workpiece
3 Axis Workpiece Holding

For 3 axis workpiece holding you want to use the T slot table. Unscrew everything from the table and clean it. Also clean the bottom side of the T slot plate.
3 Axis Workpiece Holding

Place the T slot plate on the table and align the 4 holes.
3 Axis Workpiece Holding

Use the flat heat screws to secure the plate on the table. Fittings tools can be found in the drawer bellow.
3 Axis Workpiece Holding

The tool sensor will be placed in the front right corner and secured using the three polyamide screws.
**Terms**

The 2 options we have to secure our workpiece are step blocks and a vise. In the following we will explain both.
3 Axis Workpiece Holding

When using the step block prepare the nuts as pictured below. Make sure the “steps” are on the side of the t-nut.
3 Axis Workpiece Holding

Place your workpiece and slide the nut in from the side. Place the step blocks on the side.
3 Axis Workpiece Holding

Make sure that the screw is placed as close to the workpiece as possible.
3 Axis Workpiece Holding

It is important that the hold down plate is flat on the workpiece or angled towards it.
3 Axis Workpiece Holding

A vise can be helpful when securing smaller workpieces. It can be secured on the table using screws.
3 Axis Workpiece Holding

Insert the t-nut in the slots and secure the vise. Make sure to use a washer!
3 Axis Workpiece Holding

Make sure the workpiece is inserted flat and securely.
How to straighten your workpiece/vise
Workpiece straightening

Secure your workpiece and install the dial gauge as shown in this picture. Note: This will only work if your workpiece side surface is properly machined.
Workpiece straightening

Adjust the arm so that the needle almost touches the workpiece. Then use the handwheel to move the table until the needle touches.
Workpiece straightening

You can see the dial indicating how far in you are. Do not go too far or the dial will show a constant reading.
Workpiece straightening

Now move the table in the x direction and see how the dial is changing. Adjust the alignment by lightly hitting the workpiece with a rubber hammer.
Vise straightening

The same procedure can be applied to the vise. Install it and then move the dial needle along the back side.
Vise straightening

The fastest way to straighten the vise is to securing one side and only slightly securing the other. This will create a rotation point.
The shown method will straighten your workpiece/vise. The pros of this are:

- X-Axis is parallel to part edge
- Less danger of cutting outside your part
- Less extra material needed (if zero position is correct)
MDX-540 Handwheel
Handwheel

The handwheel can be helpful in working with the machine. This is the base after starting the machine. Hit enter to start the initialization.
Handwheel

After the initialization you will be prompted with this view. There is a lot of information on the screen, so let's go through it step by step.
Handwheel

After the initialization you will be prompted with this view. There is a lot of information on the screen, so let's go through it step by step.
Handwheel

After the initialization you will be prompted with this view. There is a lot of information on the screen, so let's go through it step by step.
Handwheel

After the initialization you will be prompted with this view. There is a lot of information on the screen, so let's go through it step by step.

- **Position**
- **Movement Speed**
- **Spindle Speed**
Handwheel

After the initialization you will be prompted with this view. There is a lot of information on the screen, so let's go through it step by step.

Position
Movement Speed
Spindle Speed
Speed Override
Handwheel

There are also buttons, let's see what we can do with those.
Handwheel

The X, Y, Z buttons can be pressed to select a axis, the axis can then be moved by rotating the wheel. The speed is selected by pressing the RATE button. Be careful not to hit something!
Handwheel

The A button is only helpful when the 4th axis is installed. Press it to rotate the axis using the wheel.
Handwheel

The OVER RIDE button can be used during cutting to override the speeds and feeds selected in the software.
Handwheel

An important button is the MENU one. Press it to get to more options.
Handwheel Move

The first menu is the move menu. Selection VIEW and pressing ENTER will move the table to the front.
Handwheel Move

If you select USER-XY the machine will move to the XY 0 position that you selected in the software
Handwheel Move

Selecting MACHINE-XY will move the machine to its own 0 position.
Handwheel ATC

Pressing the MENU button again will take you to the second menu, which is used to control the automatic tool changer (ATC).
Handwheel ATC

Selecting a tool on the left and confirming with ENTER lets the machine change the tool. Make sure the compressor is on!
Handwheel ATC

Selecting return will put the tool back in its place in the ATC stock.
Handwheel ATC

Mag. Cover will open/close the cover to insert tools manually.
Handwheel ATC
Meas. Length will measure ALL the tools in the ATC. Make sure the compressor is turned on and the sensor cable is connected!
Handwheel ATC

Force release does what you would expect. Make sure the compressor is on and that you hold the tool so it does not fall down.
Handwheel

Selecting Return to main will take you back to the base view.
Handwheel

When you get an error message it can be deleted using the CLEAR button.
Roland V-Panel

The interface to the MDX-540
V-Panel

V-Panel is the software that controls the machine. When you open it you will see this screen. Let's see what's on there.
V-Panel

This indicates which tool holders are filled and what tools have been selected to be in each holder.
V-Panel

Here you can see which addons are installed on the machine right now. The only one that might grayed out is the rotary axis unit (more on that later).
V-Panel

Time that has passed since the start of the job.
V-Panel

Overrides can be used to speed up or slow down the different speed during cutting. In general the cutting settings will be provided by the Makerspace.
V-Panel

There are two options: Machine and User Coordinate System. Each show you the relative position of the tool tip relative to the selected system.
V-Panel

This box indicates how much distance is to travel during an operation (e.g. line till endpoint). This can be helpful in preventing collisions when moving slowly.
V-Panel

Status tells you what the machine is doing right at the moment. Error messages will also be displayed here.
V-Panel

These numbers tell you the speed and feed right at the moment.
V-Panel

If you have an Error you can clear it here. While cutting you can pause/stop the job. You can stop the spindle to securely check your work. The Move to VIEW allows you to get the part to the front to check the results.
V-Panel

The Load indicator gives you information about how high the load on the tool is. This is helpful to prevent breaking tools. If this value is too high, reduce the feed (cutting speed).
V-Panel

There are also some more settings on the top. Let's dive into the important ones.
V-Panel Cut

Pressing the Cut button will open this window. If you have a cutting file you can load it here. You can also start your cut directly in the SRP Player.
V-Panel Cut

Open your part, select it and press output to start the cut.
V-Panel Base Point

The Base Point option is really important and will be used for every job. We will explain how to use this later.
V-Panel Move Tool

If you press the Move Tool option you can move the spindle around using the software instead of the handwheel.
V-Panel Move Tool

We have two options. Either moving the spindle by pressing the buttons on the left, or move to a specific coordinate using the buttons on the right.
V-Panel Setup

Most of the settings in the Setup tab are not needed to be dealt with. The Makerspace will provide all the presets for you.
V-Panel Preferences

If for any reason your unit system is switched to inches you can change it back to mm in the Presences tab.
V-Panel Magazine

One of the most important tabs is the Magazine tab. It is used to store all the data of the tools installed. How to change a tool will be explained later.
V-Panel Attach

Attach can be used to change out tools. This is working together closely with the Magazine option.
Tool Change

Installing tools on the MDX-540
Tool change

After your tool was secured in the collet and shank there are different ways to go. We will explain the safest one here.
Tool change

Turn on the machine and in the ATC menu select the Magnetic Cover option. The cover should now open. When it is open turn off the machine.
Tool change

After you turned off the machine you can insert tools in the free spaces. The ATC positions are labeled bellow.
Tool change

After all the tools are inserted you can turn machine on again. Now we have to take care of the software.
Tool change

Turn the machine and the compressor back on. Now select the Magazine tab. All the positions where a tool was installed are now white.
Tool change

Use the dropdown to select the tool that was inserted in each position. Double check your work.
Tool change

Now it is time to measure the length of each tool. Make sure to connect the sensor cable to the tool measuring sensor.
Tool change

The simplest way to measure the tools is to select the Measure All option. The machine will do all the work for you.
Tool change

After the measuring is done you can close the Magazine window and go on with the next step.
The first thing to do before cutting
V-Panel Base Point

Before starting a job it is important to set your 0 position. This is done in the Base Point tab of V-Panel. Make sure a tool is loaded and the length of the tool was already set.
V-Panel Base Point

To start you can move the spindle and table around. Control the speed using the selection at the bottom. You can also use the handwheel. Be careful not to crash into something on the table.
V-Panel Base Point

Move the spindle so it is located above the center of your workpiece. It might be helpful to mark the center beforehand.
V-Panel Base Point

Now select the set XY Origin option and press apply. This is the first 0 Position you have to set before cutting.
V-Panel Base Point

Connect the sensor cable to the z sensor. And place it on top of your workpiece. Then move the spindle with an tool installed over the sensor.
V-Panel Base Point

Double check the connection and press “Start Detection” to measure the z height. If the detection was successful you will get a pop up telling you.
SRP-Player

Easy to handle CAM
SRP Player

SRP Player is a simple software to create the cutting path for your part. Open it and it will prompt you with this window.
SRP Player

The first thing we want to do is to navigate to File – Preferences and adjust the setting for the machine. Explicitly one want to select/deselect the rotary axis here.
SRP Player

We will use the software going from top to bottom on the right hand side tabs. First use the Open button to load an .stl file.
SRP Player

The Software will preview your part and its dimensions. If they are not correct you can adjust them. Also the rotation can be changed beneath.
SRP Player

The first thing we want to do is to go through the questions under the “Type of Milling” tap. This will heavily influence the results so take care.
SRP Player

The software is also asking if you want to machine a cylindrical workpiece or a block workpiece.
SRP Player

When selecting “Block workpiece” and “Cut top and bottom” the machine will ask you to rotate the workpiece. This is best done with the rotary axis. More on that later.
SRP Player

Selecting the “Add support to model” options will create taps to hold your piece while machining. These have to be manually removed after machining. This can be helpful when using the rotary axis.
Next we will create the tool path. Select your materials on the top and add your workpiece dimensions below. The values on the right will tell you the minimum workpiece size.
SRP Player

After all the dimensions are added the “Create Tool Path” button is clickable. Click it and wait for the process to finish.
SRP Player

The created paths can be found on the right. They are divided in two kinds.
SRP Player

Roughing is a process to create the rough shape of your part. A lot of material will be milled away here, but some will be left.
SRP Player

The leftover material will slowly be removed during the **Finishing**. This process is way slower to create a nice surface finish.
SRP Player

Both process can be further defined by expanding them. One thing the software usually does is leaving some material left around your piece. You can see the dark green box is smaller than the light green one.
SRP Player

To change this navigate to Top Surface – Modelling Form and increase the margin bellow.
SRP Player

Clicking “Apply” in the edit section will create a pop up asking you if you want to delete the path. Click “Yes”.
SRP Player

You can see the dark green box now is bigger than the light green one. Be careful not to machine in the vise using this method.
SRP Player

The software is preloaded with available tools. Using this tool library the software will auto select a tool. You can change this by pressing on “Flat”/”Ball” and then using the dropdown.
SRP Player

After pressing “Apply” and “Close” you will get back this view. You can see that the tool path was not completely created. Select the “Create Tool Path” button to put the new settings to action.
SRP Player

This process might take a second.
SRP Player

The preview window will offer a rough time estimate. Pressing the “Preview Cutting” button will provide you with a preview of your results.
SRP Player

Check the preview and if needed go back to the “Create Tool Path” tap.
SRP Player

The final thing you want to do is to adjust the tool list. To do so click on the “Edit magazine” button. Make sure the list is the same as the one in the V-Panel Software.
SRP Player

If the list is correct press the “Start Cutting…” button. The software will ask you once more to install the tools. Make sure that the cutting length of each tool is at least as long as indicated on the right and all tools are measured.
SRP Player

Next we need to select the position of our zero point according to the one we used in the V-Panel set up. Make your selection and click “Next”.

![Image of SRP Player interface]
SRP Player

While cutting the SRP Player will tell you which process you are working on and give you a time estimate of the left over time. We found this to be wrong most of the time.
SRP Player

More interesting is the output of the V-Panel software. You can see a lot of stuff. Let's go through it one more time.
SRP Player

Here we have the current position of the spindle. This might be helpful in estimating how much further the spindle needs to go (e.g. z direction).
SRP Player

We also have a speed and feed reading.
SRP Player

“Distance to Destination” will tell you much there is to go (e.g. end of straight line).
SRP Player

Error messages will be put here. In this instance the machine is working nominally.
SRP Player

Last but not least you can override the speed and feeds here. Be careful with this to not damage the machine or tool.
3 Axis Milling Walkthrough

Hardware Walkthrough

1. Cut your workpiece to size
2. Tool Preparation
   1. Select tools
   2. Secure tools in collet
   3. Insert tools in ATC
3. Workpiece Holding
   1. Vise
      1. Straighten Vise
      2. Secure workpiece
   2. Step blocks
      1. Secure workpiece
      2. Straighten workpiece
3 Axis Milling Walkthrough
V-Panel Walkthrough

1. Tools
   1. Adjust tool magazine according to ATC
   2. Measure tool lengths

2. Zero Point
   1. Measure XY zero point
   2. Measure Z zero point using the z sensor
3 Axis Milling Walkthrough

SRP Player Walkthrough

1. Import .stl file
2. Check model size and orientation
3. Answer type of milling questions
4. Create Toolpath
   1. Check toolpath
   2. Make adjustments
   3. Recreate Toolpath
5. Preview Cutting
   1. Double check results
6. Start Cutting
   1. Change magazine in SRP Player (check length)
   2. Note zero position
   3. Start cut
4 Axis Milling

How to use the 4th axis on the Roland MDX-540
Installation

First you want to uninstall the z-sensor and the t-slot table. To do so unscrew the screws indicated bellow.
Installation

Make sure that the table is clean and free of chips.
Installation

Next we need to install the base plates. Note that they have indicator pins.
Installation

There are corresponding holes for the pins in the table. Slide them in and make sure the lay down flat.
Installation

The orientation is correct when a t-slot was created between the plates. Use the 6 provided M8x20 screws to secure the plates on the table.
Installation

The drive unit and tailstock will be installed next. Make sure the bottom surface is clean. Also note that the drive unit has a stopper on the bottom left.
Installation

Both units are secured on the base plates using the t-nuts that are preinstalled. The drive unit can be secured by tightening the M8x20 screw that holds the t-nut. Do not tighten the tailstock yet.
Installation

Next the z-sensor is installed behind the drive unit. Use the predrilled holes to do so.
Installation

The cable from the drive unit will be installed on the left side of the machine. There is only one connector that will fit. Make sure to secure the plug using the screws on its side.
Zero Position

To set the zero position while using the 4th axis we need the following tools.

- Y-Origin Sensor
- Detection Pin
- Live Center
Zero Position

Install the detection pin in an collet and make sure the it is reaching out 25 to 35 mm. Then insert it in ATC position 1.
Zero Position

Next we need to install the y-origin sensor. Please make sure to follow the next steps in the same order to avoid damage to the machine.
Zero Position

Install Y-Origin Sensor.
1. Loosely tighten the clamp.
2. Place live center in tailstock and push against sensor.
3. Tighten clamp.
4. Turn livestock clamp ½ turn.
5. Tighten retaining knob.

Orient so that the end with these holes is positioned close to the chuck.
Orient so that the two holes are level.
Zero Position

When you open the V-Panel software you can see, that the 4th Axis was automatically recognized.
Zero Position

To set the zero position navigate to Options – Detect Center of Rotation. Make sure the listed points are true.
Zero Position

On the machine it will look like this. Note the sensor cable is installed on the z-axis sensor.
Zero Position

The toolholder with the sensor pin is installed in the ATC at position 1.
Zero Position

After the height of the pin was measured, V-Panel will ask you to install the sensor cable on the y-origin sensor.
Zero Position
This should look like this!
Zero Position

After some time the software will ask you to rotate the tool by hand. This part is important for good results.
Zero Position

If everything went according to plan you will get the following message. But we are not quite done yet.

---

Detection of the axis of the center of rotation is complete. Detach the sensor cable from the Y-origin sensor.
Zero Position

First we need to set the Y axis at the center of rotation.
Zero Position

Next install a tool and measure its length as you would normally do.
Zero Position

After this set the z-origin at the center of rotation.
Zero Position

Make sure to connect the sensor cable to the z-origin sensor.
Zero Position

Now mount your workpiece in the holder by using the clamp. Make sure the workpiece is as centered as possible.
Zero Position

Insert the center drill bit and move it close to your workpiece. Make sure it is not touching. Then secure the center to the t-slot using the screw.
Zero Position

In the “Options” menu select “Drill Workpiece” follow the instructions on the screen to create a hole at the center of rotation.

1. Mount the workpiece on the rotary axis unit.
2. Remove the live center and replace it with the center drill.
3. Click [Rotate].
4. Bring the center drill into contact with the workpiece and drill the hole.
5. After milling has finished, click [Stop].
Zero Position

To move the tool in the workpiece use the screw on the side.
Zero Position

A hole will be created. Make sure it is as wide as the center drill (5mm) and at least 5mm deep.
Zero Position

The x-zero position for the 4th axis is set above the live center. Install your stock and move the spindle above this position and set the x origin here using v-panel.
4 Axis SRP Player

How to use the 4th axis on the Roland MDX-540
First we need to manually let the software know that we use the 4\textsuperscript{th} axis. Go to Preferences – Cutting Machine and Check the ZCL-540 Rotary Axis Unit.
Settings

Next load the part as you would usually do. Either by drag and drop or using the “Open” button.
Settings

The rotating axis is shown by the dashed line. As you can see we need to use to tools on the right to rotate the part.
Settings

This is a better orientation for our part. Let's now go through the steps as we did with 3 axis machining.
Settings

The questions are the same. The only difference can be found when selecting the workpiece. Here we have two options that make sense.
Settings

For this part we want to use a cylindrical workpiece. The process for an block workpiece will be explained in a second.
Settings

For both it might make sense to add supports that can later be cut. To do so check the box. We now can do some refinements. Click on edit.
Settings

As you can see the supports reach over the end of the part which is not ideal. Edit the values on the right to make them smaller or bigger. You can also drag the supports around by clicking on its edge.
Settings

After adjusting them one can see that the results are better. Make sure that the supports are not to small or they might break during machining.
Settings

If you use a block workpiece choosing the option “Cut top and bottom” will turn the workpiece after finishing one side and then do the other. This will save you a second workpiece holding. The rest is the same.
Settings

Next we will need to insert the data of our workpiece on the right.
Settings

As always the next step is to create the toolpath. Again click on edit to see some further details.
Settings

You can see that the software did not select the best tool for this operation. A ball end mill will be better for this curved surface. You already know how to change the tool.
Settings

Make sure to change it for both operations (at least the finishing).
Settings

Click on “Preview Cutting” to get a preview of what the machine will do. This might take a second to calculate.
Settings

If you are happy with the results go to the next step. If not, go back to the “Create Tool Path” tap.
Settings

At last go through the “Start Cutting” process as you normally would. Double check, that the cutting surface of your tool is at least L mm long.
Settings

From here on it's all the same as with 3 axis operations.
4 Axis Milling Walkthrough

Hardware Walkthrough

1. Cut your workpiece to size
2. Tool Preparation
   1. Select tools
   2. Secure tools in collet
   3. Insert tools in ATC
3. Workpiece Holding
   1. Install 4\textsuperscript{th} Axis
   2. Run zero position selection (y&z)
   3. Install Workpiece
   4. Select x zero position
4 Axis Milling Walkthrough
V-Panel Walkthrough

1. Tools
   1. Adjust tool magazine according to ATC
   2. Measure tool lengths

2. Zero Point
   1. Double check zero position
      1. Move to XY-Zero
4 Axis Milling Walkthrough

SRP Player Walkthrough

1. Import .stl file
2. Check model size and orientation
3. Answer type of milling questions
4. Create Toolpath
   1. Check toolpath
   2. Make adjustments
   3. Recreate Toolpath
5. Preview Cutting
   1. Double check results
6. Start Cutting
   1. Change magazine in SRP Player (check length)
   2. Note zero position
   3. Start cut
Safety Rules / make sure !!!

- Make sure your workpiece is secured
- Do not change tools when spindle is moving
- Never touch moving spindle
- Always keep your hands away from the moving spindle
- No towels near moving parts
- No measuring or marking when machine is running
- Only clean the machine when turned off
- Never change the safety systems