



ROLAND MDX-540 WITH ROTARY AXIS

A BEGINNERS GUIDE

BY RUBEN HOLSBEKE

GUIDED BY PROF GABRIELA CELANI

CONTENTS

- 1 STARTING THE MACHINE.....3

- 2 PREPARING OF THE WORKPIECE.....5
 - 2.1 CYLINDRICAL OR BLOCK WORKPIECE.....5
 - 2.2 CUTTING AREA.....6
 - 2.3 PLACING THE WORKPIECE.....7

- 3 SRP PLAYER SOFTWARE.....11
 - 3.1 MODEL SIZE AND ORIENTATION.....11
 - 3.2 TYPE OF MILLING.....12
 - 3.3 CREATE TOOL PATH (CYLINDRICAL WORKPIECE).....12
 - 3.4 CREATE TOOL PATH (BLOCK WORKPIECE).....16
 - 3.5 PREVIEW CUTTING.....19
 - 3.6 PERFORM CUTTING.....19

- 4 PREPARING THE MACHINE.....20

- 5 DURING RUNNING OF THE MODEL.....24

- 6 ERRORS.....25

1 STARTING THE MACHINE

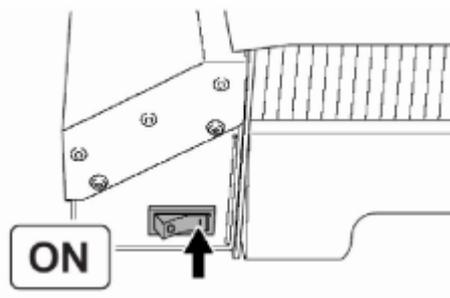
1) Start the compressor: Plug in the red electricity plug (230V) and pull the red button.



1) Plug in the handypanel. (To remove the handypanel it is important to not pull the plug out without pulling the security which is around it!)



2) Start the machine by pushing the button at side of the front panel. Nothing happens: check if the emergency stop is not pressed and the electricityplug is connected.



3) Press enter on the handy panel or front panel, the machine will initialise.



When there is still a tool present in the toolholder at the starting of the machine, the tool will have to be manually removed: go to **6 ERRORS**

4) If all went well, the following screen will appear on the handypanel



5) Open Vpanel for MODELA PRO II, it can be found on the desktop:



6) A window opens, click [OK].



2 PREPARING OF THE WORKPIECE

2.1 CYLINDRICAL OR BLOCK WORKPIECE

There are 2 ways to mill a workpiece:

- Rotational milling: proces that uses the rotational axis in the milling of a **cylindrical workpiece**. The tool will always be perpendicular on the top of the circle going in the x-direction while changing the depth of cutting: the z-direction.
- Plane milling: proces that mills a **block workpiece** in a certain angle in x and y direction. From top and/or bottom or any given angle.

Choose cylindrical: if it is evenly distributed around a central axis and not much material is spilled in a cylindrical way. It's less time consuming to make the workpiece in a cylindrical shape, then to mill the excess material away. For example:

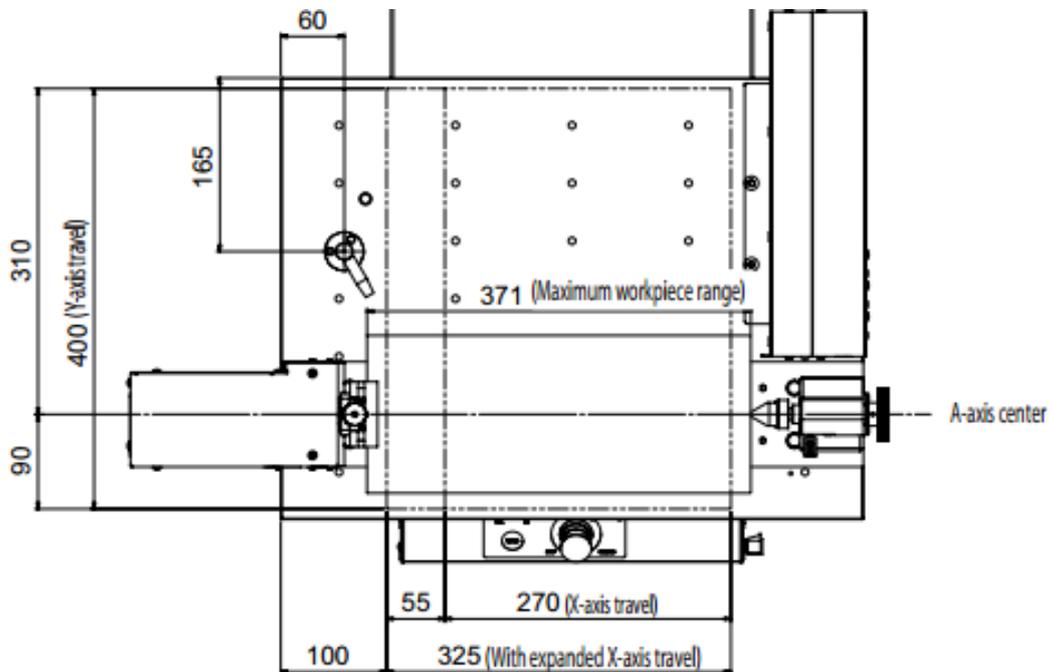


Choose block if your highrise building is more distribitued in one direction than the other and thus a cylindrical workpiece would waste a lot of material and time. For example:



2.2 CUTTING AREA

Follow the text in this paragraph to calculate the dimensions of your workpiece.



1) what is the x-dimension?

The maximum size of the workpiece is 371mm in the x-axis. But the actual cutting size in safe mode is smaller, because of the clamp and the centerpiece, it is 270 mm.

The SRP player software will automatically add the safe zones to the workpiece, so you don't have to take care of that in your design.

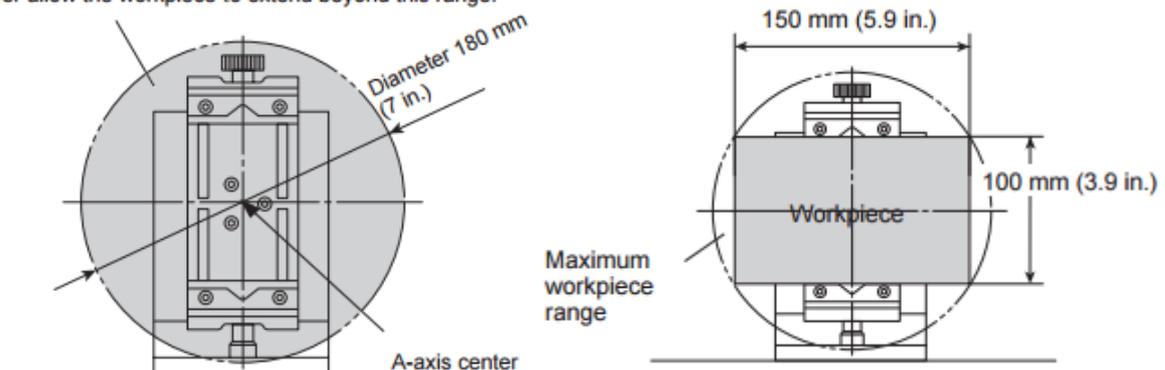
What is the minimal length (in X-direction) of my workpiece?

The dimensions that will be added to the length of your geometrie is **84,51mm**, so make sure that your workpiece has that length added, longer lengths are no problem.

2) what is the y and z-dimension

Maximum workpiece range

Never allow the workpiece to extend beyond this range.



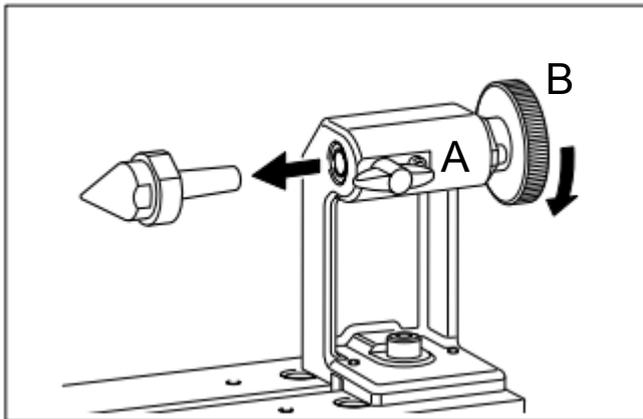
The maximum dimensions in depth and height of the workpiece are shown in the left and right figure. Never exceed the range shown in the figure on the left under any circumstances. This can be even further restricted when a long tool is used, because a long tool maybe can't extend far enough in the z-direction.

You can check that by extending the longest tool in the z-direction to the maximum and see if the tool doesn't touch the workpiece.

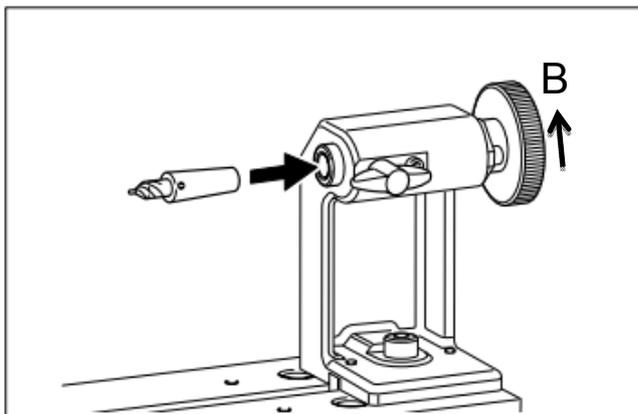
Figure on the right shows that the maximum thickness of the workpiece cylindrical or block can only be 100mm or less. At that thickness the depth is max 150mm with a block workpiece.

2.3 PLACING THE WORKPIECE

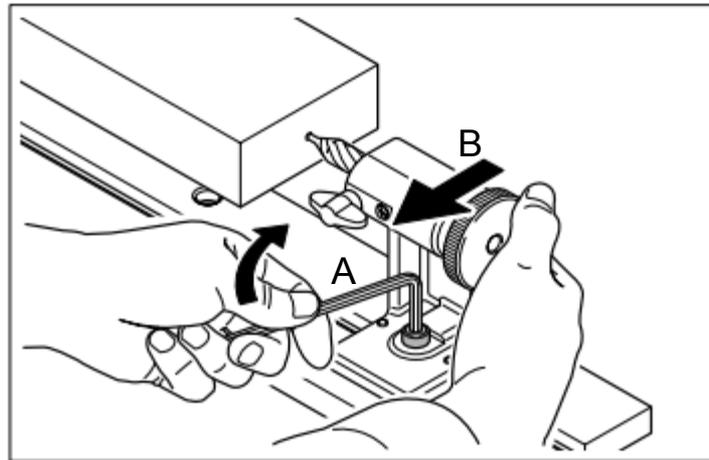
- 3) Draw a point on the middle of the right side of the workpiece
- 4) Pull out the Live Center: loosen the knob A retract the spindle by turning the knob B in the direction shown on the figure to make it easier detaching the spindle.



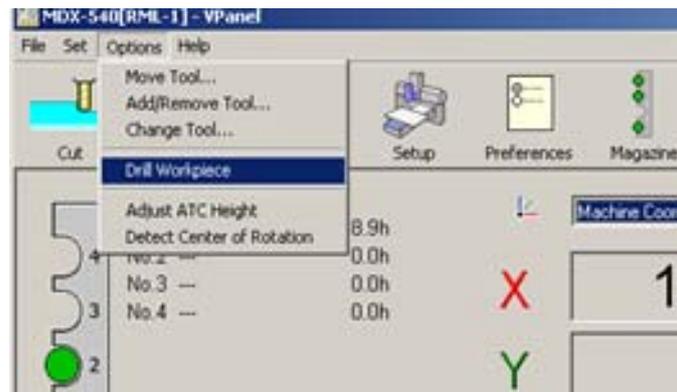
- 5) Now put in the Center Drill and turn back the knob B in the direction shown on the figure so the drill is fastened.



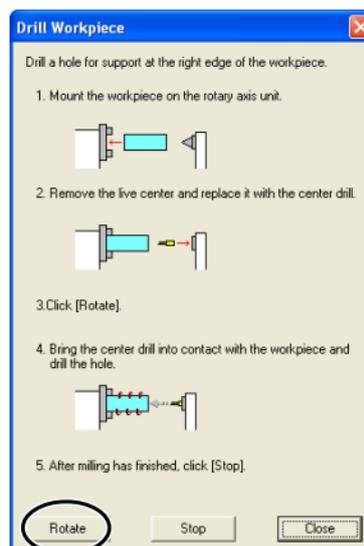
6) Loosen the retainer screw (A) by turning in the opposite direction as on the figure. Then slide the tailstock against the workpiece with the drillpoint touching the center that is drawn on the workpiece. Fasten the retainer screw (A); now in the direction of the figure, while holding pressure on the workpiece; by pulling in the direction of the arrow (B). If necessary you can turn the knob a little further to tighten the workpiece.



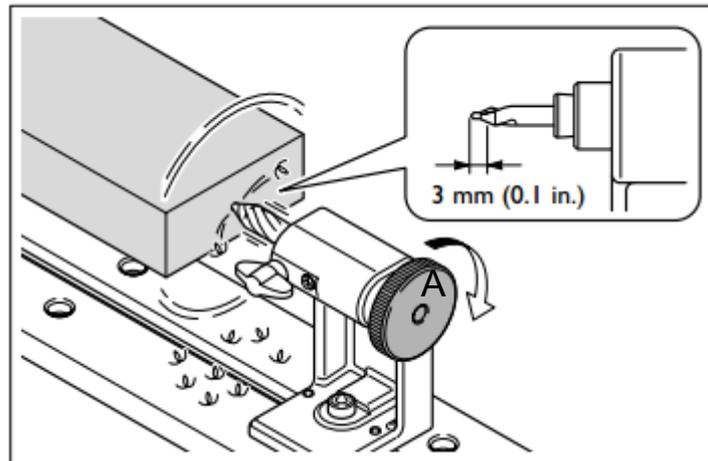
7) In the Vpanel program, go to [Options] and click [Drill workpiece].



8) The following window opens. Click [Rotate].



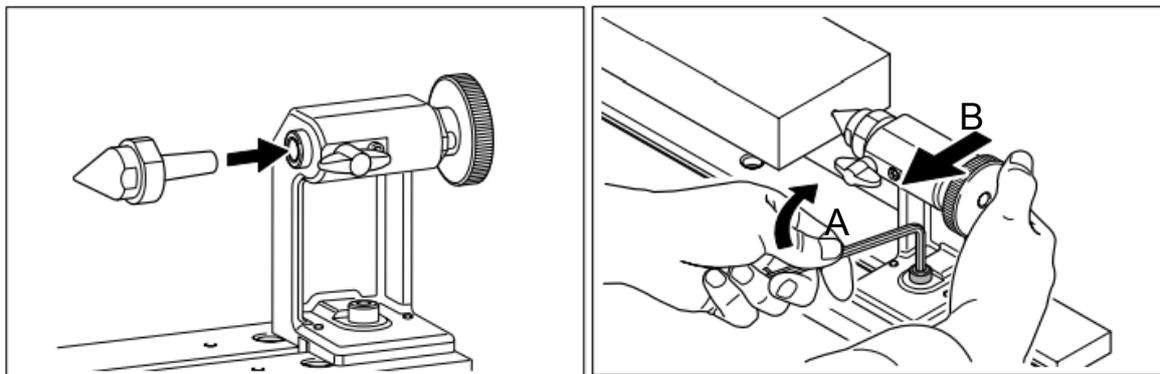
9) Turn the knob (A) in the direction on the figure to drill the hole, slowly turning until it reaches a hole of 3mm depth.



10) Click [Stop] and [Close].

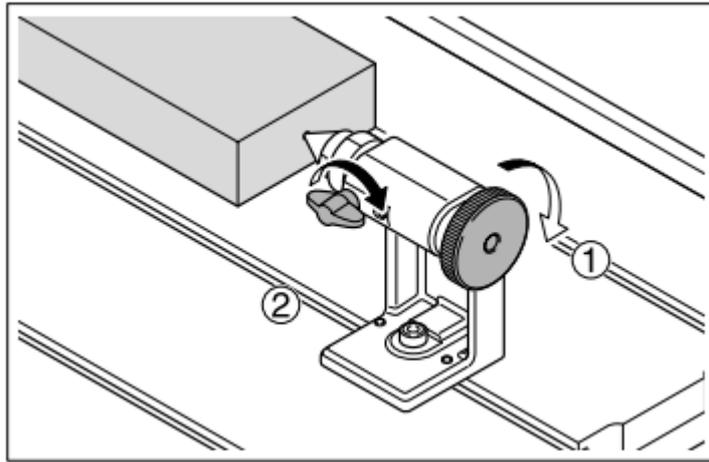
11) Loosen the retainer screw to slide the tailstock to the right. Pull out the Center Drill, like the Live Center was removed in 4).

12) Put in the Live Center, slide the tailstock back to the workpiece with the Live Center in the drilled hole. Fasten the retainer screw(A) (like on the right figure), while holding pressure on the workpiece (B).



13) (1) Turn the knob in the direction of the figure more or less half a turn, until you get a fastened workpiece. Not too hard, it must be able to turn around and not warp (bend). Not too soft and the workpiece will not be cut precise.

(2) Fasten the retainer knob. This will enable the knob (1) to unturn and loosen.



3 SRP PLAYER SOFTWARE

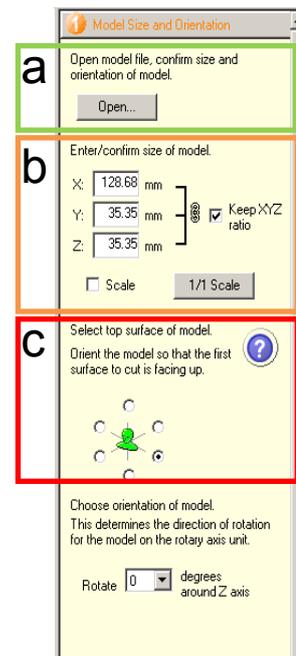
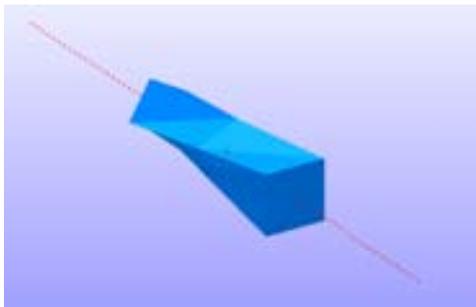
1) Open SRP Player Software



3.1 MODEL SIZE AND ORIENTATION

1) [a] Open the .stl-file of your geometrie by clicking on “Open...”
(Be sure the .stl-extension is selected at extension-type)

2) [c] Position the geometrie with its length-axis in the direction of the red striped x-axis



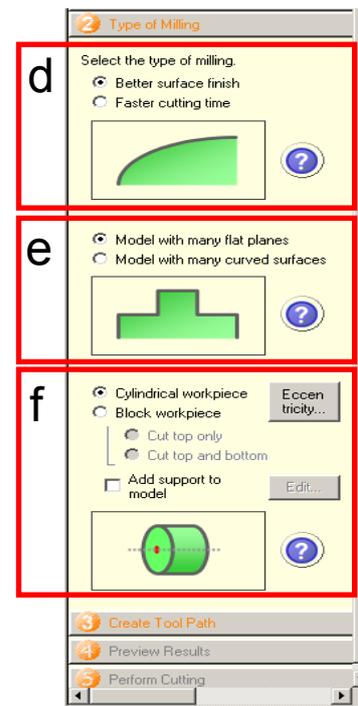
3) [b] Scale the object to the desired size by changing it in one direction, the other directions will keep the ratio, unless “Keep XYZ ratio is unclicked”. To go back to the original dimensions click on “1/1 scale”.

4) Click on [2] Type of Milling].

3.2 TYPE OF MILLING

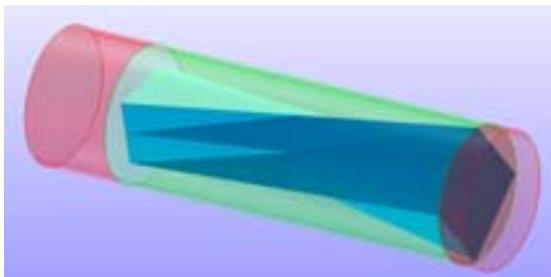
- 1) [d] The quality of the milling is selected: faster time or better surface finish? **Choose Faster cutting time.**
- 2) [e] Choose what fits best for your geometrie: **flat planes** or **curved planes.**
- 3) [f] You click the type of workpiece that was chosen in [2.1](#) **CYLINDRICAL OF BLOCK WORKPIECE.**
- 4) Click on “3) Create Tool Path”

Continue with 3.3 if you chose a cylindrical workpiece, 3.4 if you chose a block workpiece.

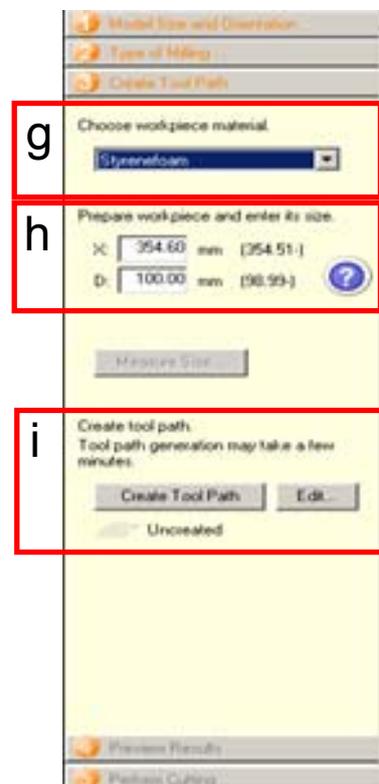


3.3 CREATE TOOL PATH (CYLINDRICAL WORKPIECE)

- 1) [g] In the list you choose your workpiece material. This is important because the rotation speed, cutting speed,... depend on the material.
- 2) [h] Give in the size of your workpiece material. X is length, D is diameter. You have to give in at least the minimum value that is given. The red zones on the model are the safety zones of the workpiece.

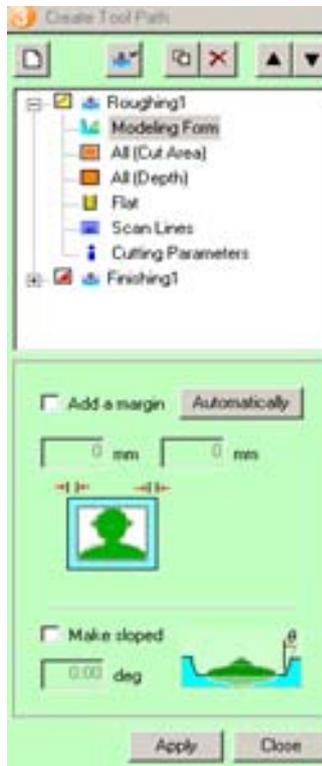


- 3) [i] Press “Edit...”, this will open the tool path options.

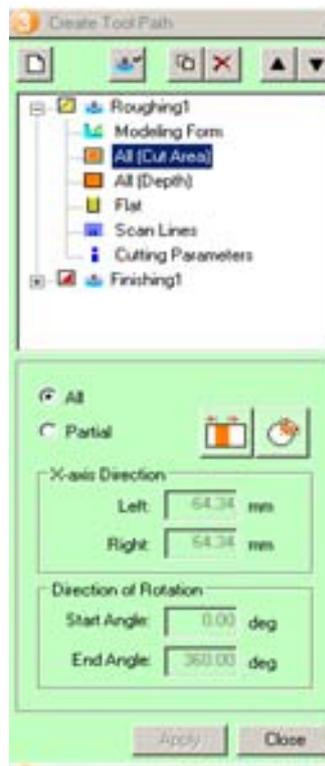


4) Tool path options

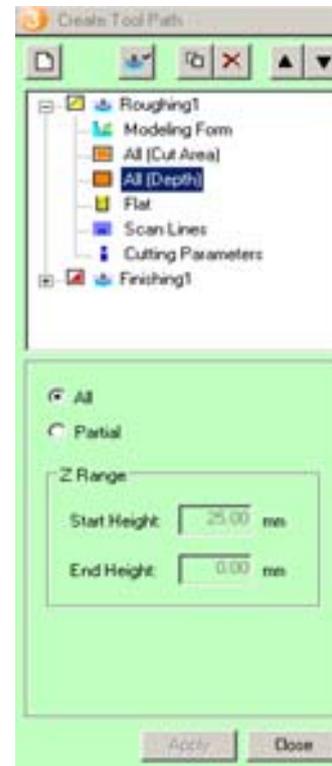
Every geometry will have different settings for the tool path, adjust where is needed with the help of the following list.



a)



b)



c)

a) Modeling Form

Adding a margin is the amount of unmilled material you want in the x-direction of your part.

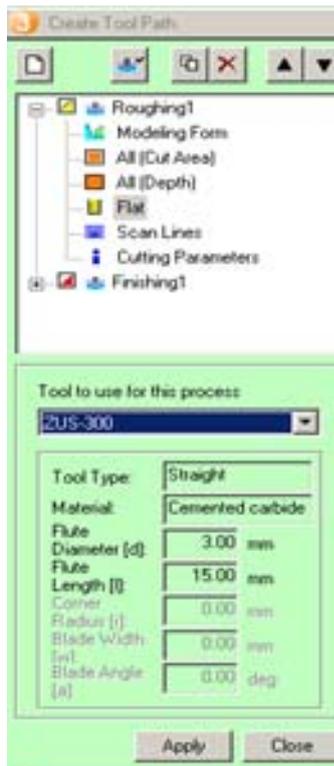
Adding a slope to the sides can make the desired length of the tool shorter. This is important when there is no long tool present but you still have to cut deep.

b) Cut Area

In this window, you can choose to not let the machine mill a surface along the x-axis. Or an angle of your cylinder. This could be needed when you want to mill a part of the surface with another tool.

c) Depth

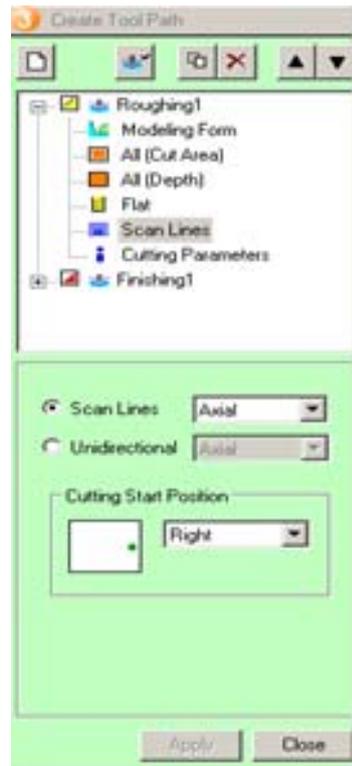
Here you can specify the depth that has to be cut. This can come in handy when you want to use another tool to mill in different depths



d)

d) Flat

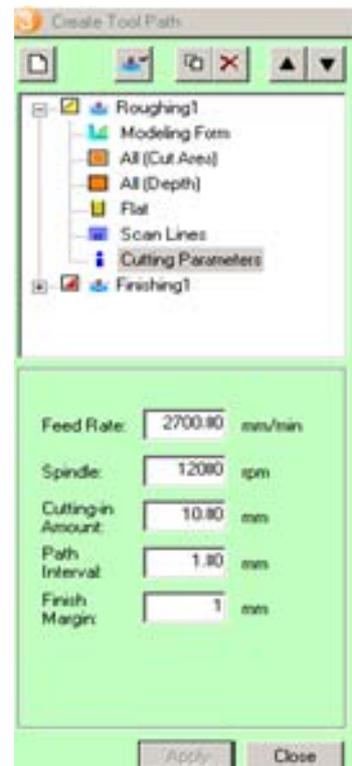
The tool with which the specific tool path has to be run is chosen here. More information about the different tools can be found in appendix 1.



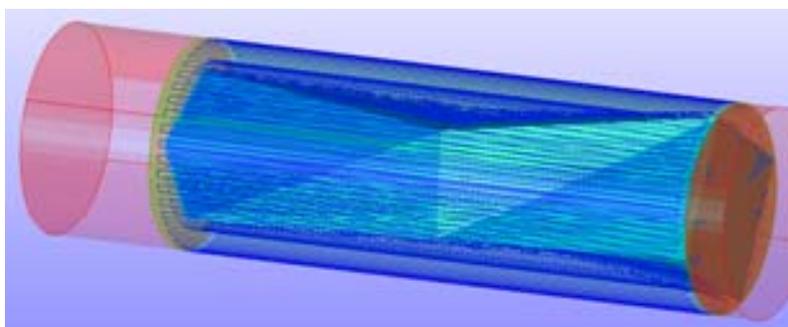
e)

e) Scan Lines

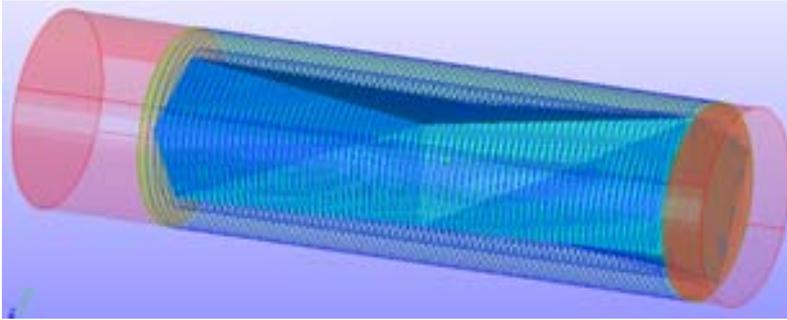
The toolpath of the milling can be set in four ways: Axial and Circumferential with and without respect for unidirection. The axial scan lines will mill along the axial of the workpiece before rotating. The circumferential scan lines will do a whole rotation before moving a step along the axis. See the figures for a better understanding. If unidirection is respected the the toolpath lines are always performed in the same direction, this takes more time.



f)



Axial



Circumferential

The cutting start position lets you define at which position the process must start.

f) Cutting parameters

These parameters are automatically set according to the workpiece material and the tool.

Feed Rate: is the speed with which the drill moves along the path (millimeters per minute)

Spindle: is the rotation speed of the spindle (rotations per minute)

Cutting-in Amount: the depth of the drill with which it cuts the material.

Path Interval: the distance between the toolpaths, this has to be more narrow than the diameter of the tool.

Finish Margin: the thickness of the remaining material surrounding the model, in the roughing process it is important to leave some remaining material so that it can be removed in the finishing process where it will be 0 thickness.

5) Click [Close].

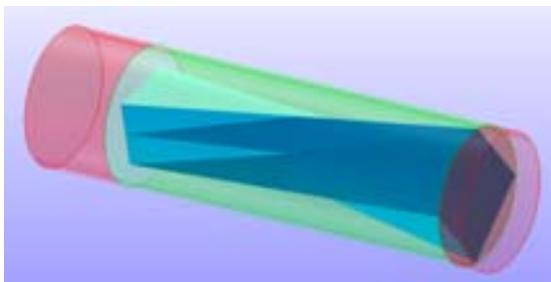
6) Click [Create Toolpath].

7) Click [4) Preview Results].

3.4 CREATE TOOL PATH (BLOCK WORKPIECE)

1) [j] In the list you choose your workpiece material. This is important because the rotation speed, cutting speed,... depend on the type of workpiece material.

2) [k] Give in the size of your workpiece material. X is the length along the axis, Y is the depth, Z is the height. You have to give in at least the minimum value that is given. The red zones on the model are the safetyzones of the workpiece.



3 Create Tool Path

j Choose workpiece material.
 Styrenefoam

k Prepare workpiece and enter its size.
 X: 300.00 mm (270.01-)
 Y: 70.00 mm (70.00-)
 Z: 70.00 mm (70.00-)

Measure Size...

l Create tool path.
 Tool path generation may take a few minutes.

Create Tool Path Edit...

Uncreated

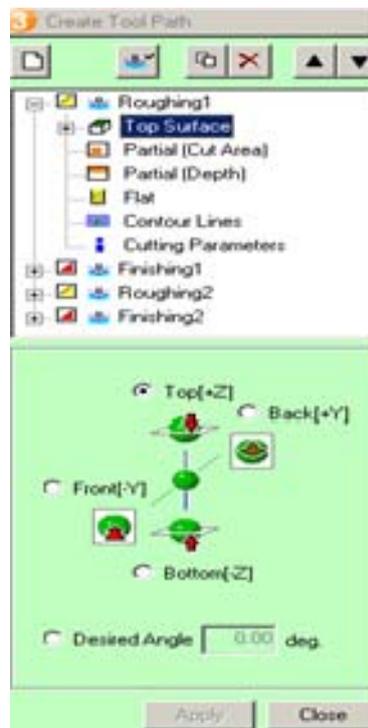
3) [l] Press “Edit...”, this will open the tool path options.

4) Tool path options

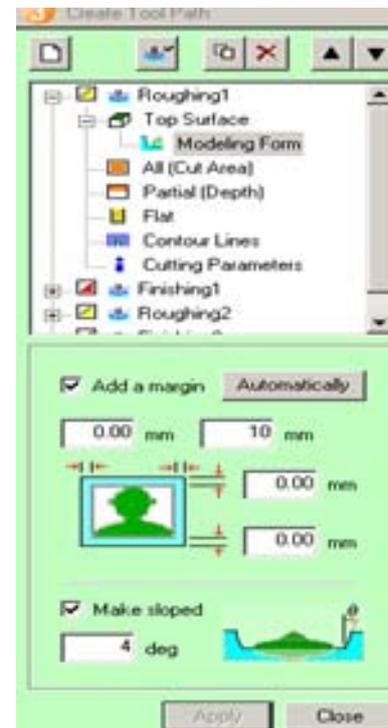
Every geometry will have different settings for the tool path, adjust where is needed with the help of the following list.



a)



b)



c)

a) Roughing and Finishing

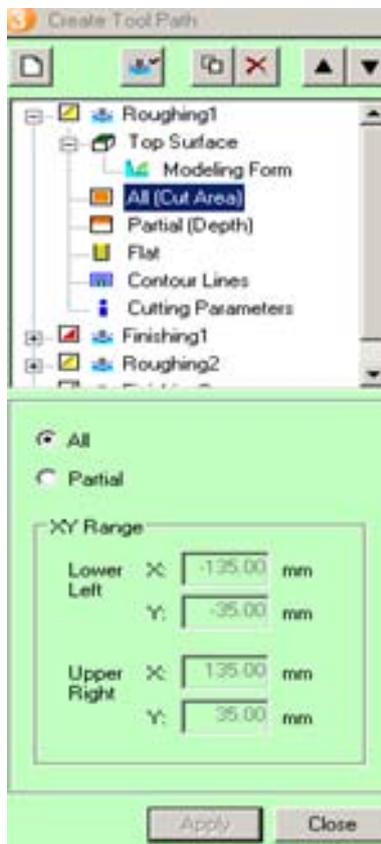
These are organising names of the toolpaths. Standard the program will make for each side a roughing and a finish. You can only do a finish when you did a roughing. The roughing will do a fast removal of the excess workpiece material until it reaches the shape of you model with a finish margin, the surface will be rough. The finish will slowly and accurately remove the finish margin to have a smooth finish.

b)Top Surface

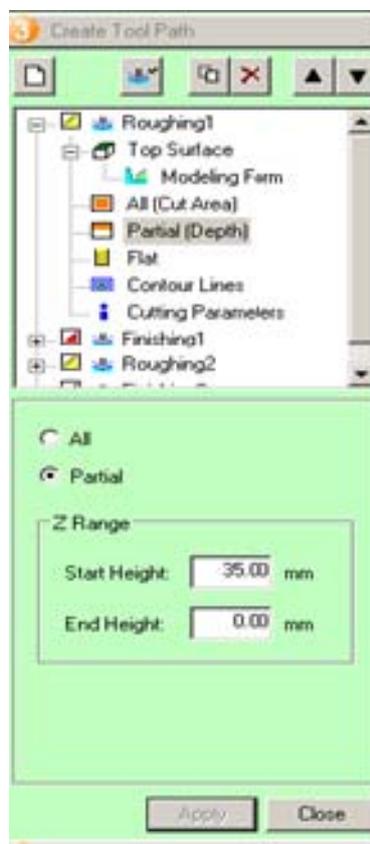
Is the orientation of the milling in relation to your part. Choose Top(+Z) to let the tool mill from above. You can also choose an angle in which the tool has to mill the part in “desired angle”. This last one can be important if you have a hole in your part that doesnt go straight but in an angle.

c) Modeling Form

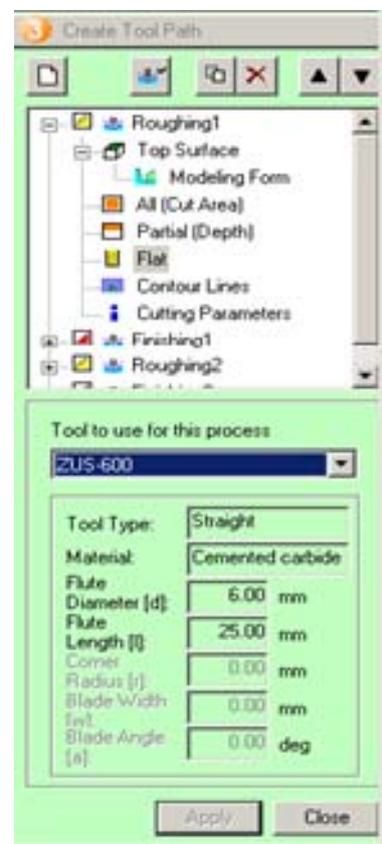
Adding a margin to the workpiece means it will cut along that margin the whole depth of the workpiece. That means that the workpiece would fall out of the clamp because it doesn't support itself anymore. Thus this should be avoided. Also giving the part a slope is not usable.



d)



e)



f)

d) Cut Area

The user can choose to mill all the surface, or just a certain rectangular range. This can for example come in handy if you want to finish parts of the surface with a different tool.

e) Depth

Only milling a certain depth can be set here. Like the previous paragraph, different tools can be utilised for different depths.

f) Flat

The tool with which the specific tool path has to be run is chosen here. More information about the different tools can be found in appendix 1.



g)

g) Contour Lines

Down Cut

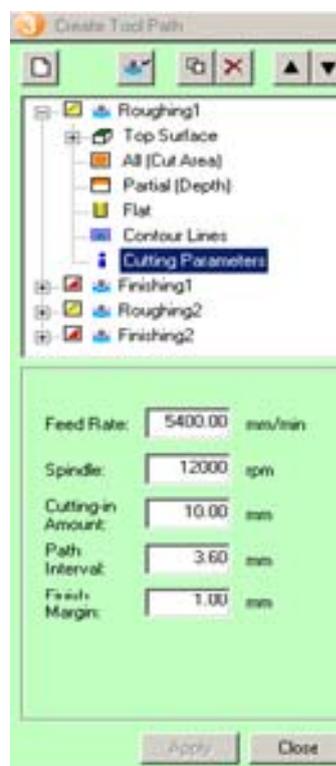
This is a cutting method in which the direction of tool rotation is opposite to the direction of tool movement (transfer).

Up Cut

This is a cutting method in which the direction of tool rotation is identical to the direction of tool movement (transfer).

Choose Up Cut.

h) Cutting Parameters



h)

See at **3.2 CREATE TOOL PATH (CYLINDRICAL WORKPIECE)** paragraph f).

5) Click [Close].

6) Click [Create Toolpath].

7) Click [4] Preview Results].

3.5 PREVIEW CUTTING

1) This window is very important. It will show you after clicking “Preview Cutting” how the model will look like after the milling process so adjustments can be made. Also here can be seen what the estimated cutting time will be, so you can compare different cutting tool paths and there effect on the cutting time.

2) Click [4) Preview Results].

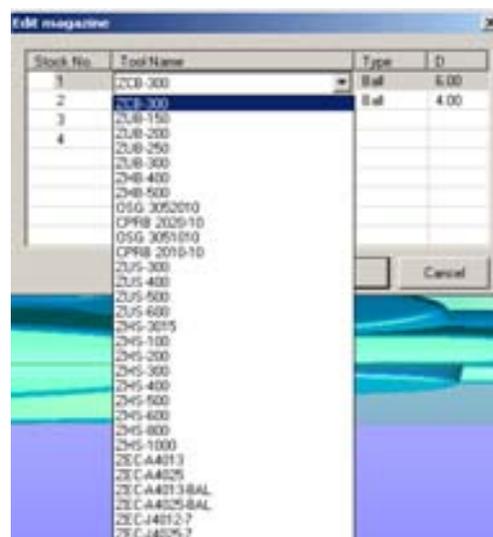
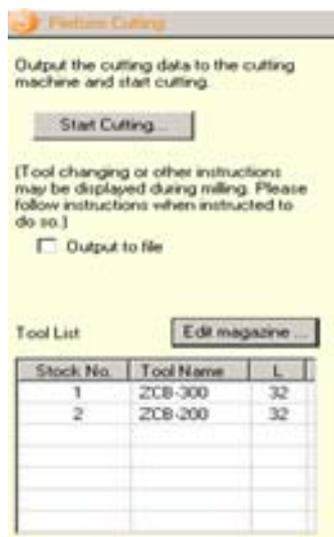


3.6 PERFORM CUTTING

1) The tools in the magazin have to correspond with the chosen tools of your toolpaths. You can do that by clicking [Edit magazine].

2) After that choose the tool out of the list that is in the exact location (Stock No.) as the one that is in the CNC-machine.

3) After all the tools are set in the right place it will be possible to press the button [Start Cutting]. Make sure that Output to file is **not** checked.



4)Go to next paragraph: 4 Preparing the machine.

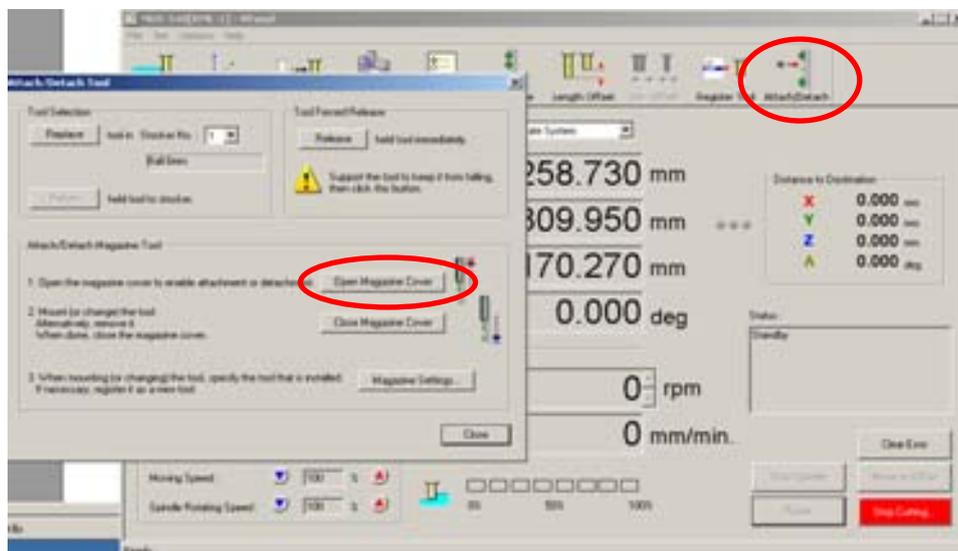
4 PREPARING THE MACHINE

1) The next window opens. The drills in the magazine of the cnc-machine have to be longer than the length given at L. This is very important! Failing to do so will result in a collision with the toolholder and the workpiece damaging it.



If the tools are already measured by hand, compare the values and if ok, go to point 5) if not ok: choose in your toolpath for longer tools.

2) Tool length check-up: Open the machine and measure the length of the drillpiece. You can do that by clicking on [Attach/Detach] and then on [Open Magazine Cover].



3) Slide out the tools out of the plastic holders. And measure the part L. If they are not long enough, use longer tools in your toolpath.



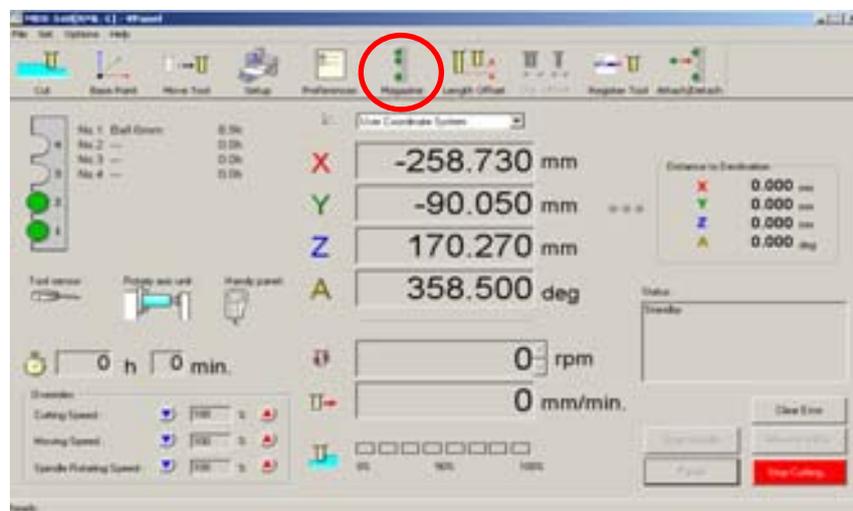
4) Slide the tools back in.

5) In the Vpanel: click on [Close Magazine Cover] and [Close].

6) In SRP Player, click on [Avançar].



7) The tool length should already be measured with the machine, if so skip to 11). If not: click on [Magazine].



8) Click on [Measure all] or only measure one tool by clicking on [Measure] at the no. of the tool.



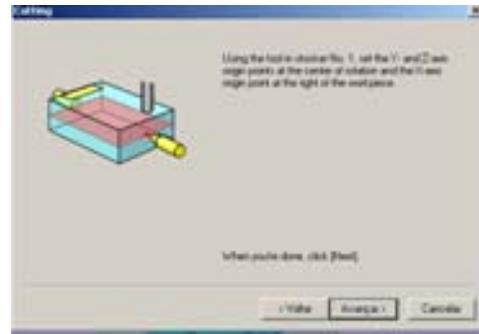
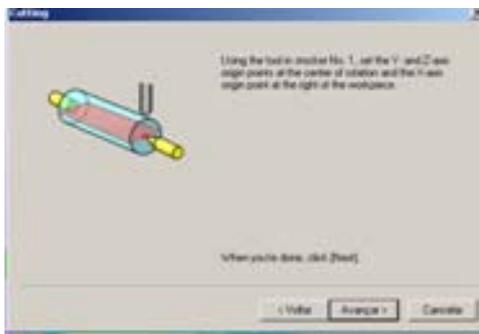
9) Plug in the sensor cable into the Z0 sensor. Click [OK]



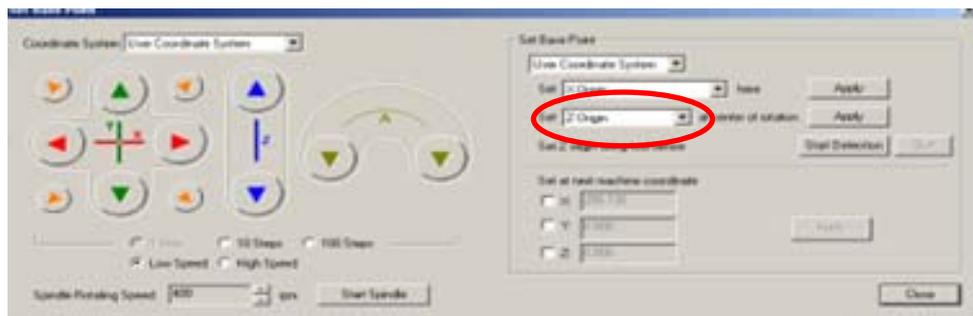
10) After measuring: click [OK]. Detach the sensor cable.

11) Now, we are going to set the user coordinate system. This is for the machine to know where the workpiece is situated.

Left is in case for a cylindrical workpiece, right is in case of a block workpiece.



12) The Y and the Z origin are already set correctly at the center of rotation. If they were changed for any reason: Click [Base Point]. Select [Z origin] at center of rotation, click [Apply], Select in the same field [Y origin] at center of rotation and click [Apply]. The Y and Z origin are now set.

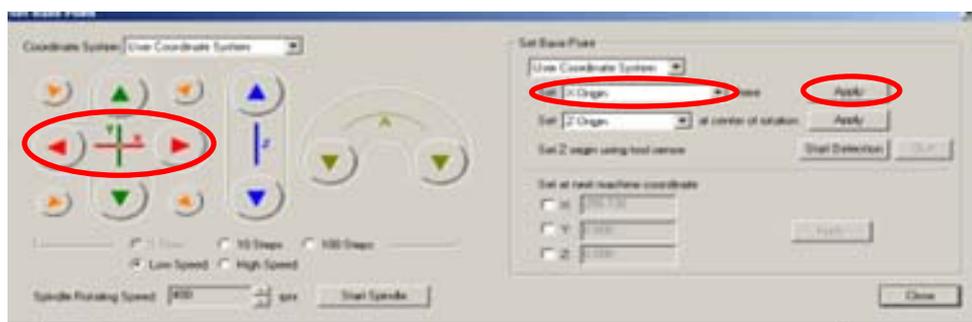


13) Now we are going to set the X point. You can use:

- The Handy Panel: Click on [X] (this activates the x-direction); turn the knob.



- Vpanel: Click [Base Point]; click the arrows of the x-direction.



to set the drill tip at the place asked by the SRP Player window; in this example right side.

Select X origin and click [Apply], see figure above. The user coordinates are set.

14) Click [Avançar] on the window at SRP Player.

15) Now all the settings are set. Click [Avançar].



16) The machine will start cutting.

5 DURING MILLING

It is possible to pause the milling of the workpiece. And to have a look to the workpiece when the front window is opened. All the time the computer and the machine must keep running, otherwise the process will stop completely and it is impossible to go on from the point where it stopped.

- 1) Click [Pause] in the VPanel. The movement of the spindle stops, but it keeps spinning.
- 2) Click [Stop Spindle] in the VPanel.
- 3) Now you can open the front window and inspect the workpiece. It's also possible to rotate the workpiece; or to move the spindle. Watch out to not collide with the workpiece.
- 4) Press [Resume] in the VPanel.

If something is going wrong or will go wrong: press the **EMERGENCY BUTTON**.

The progress will be lost, the proces cannot be continued.

6 ERRORS

6.1 STARTING THE MACHINE WITH THE TOOL STILL IN THE TOOL HOLDER