

Precix 3-Axis CNC-Mill (Router)

Setup, Control and Error Handling

Check Safety Guidelines posted in the Milling Lab !



© digitalwerkstatt D-ARCH 2006



1. HIL B41 (The CNC-Milling Lab)

- 1. Switchbox, at the side
- 3. The Mill
- 4. Webcam <http://millcam.ethz.ch>
- 5. Vacuum-Unit

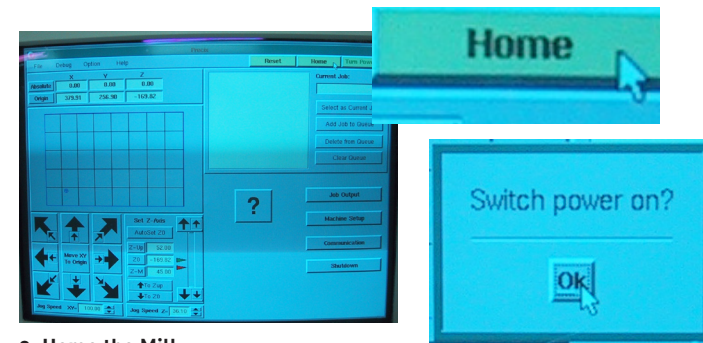


2. Computer (Precix)

EMERGENCY SHUTDOWN
USE WITH CAUTION !

2. Switching on the Mill and homing it

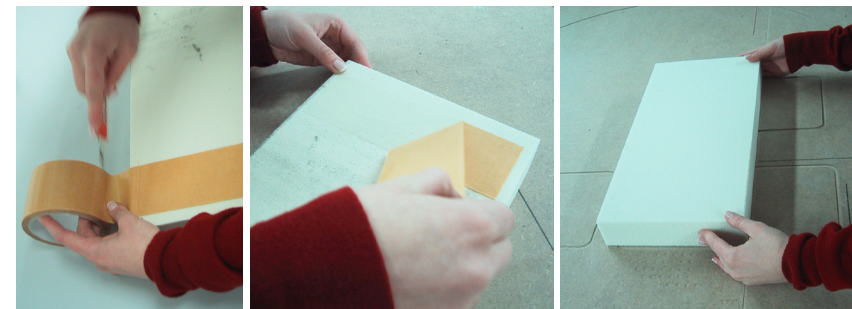
- 1. Switch on the Mill's power-supply:**
At the Switchbox, flick all switches towards the red dot.



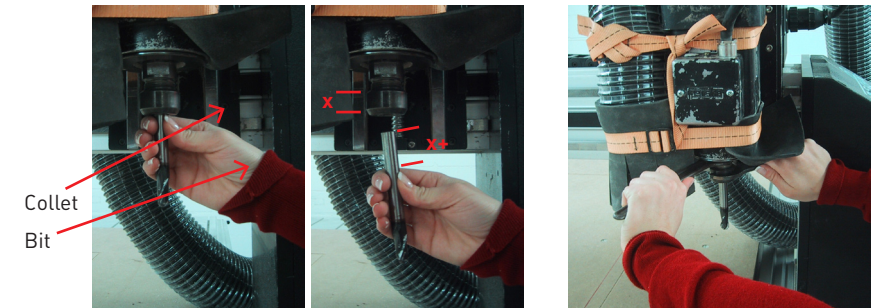
- 2. Home the Mill:**
On the Mill's computer, hit the "Home"-Button in the top right corner of the window, when prompted "Switch power on?", click "OK" - **be aware that the Mill moves !**
- 3. Homed:**
The Mill has now found it's Hardware Origin.

3. Getting your material on the table

- 1. Clean the table:**
Be sure that the table is properly cleaned from dust and debris, otherwise your material will not stick properly.
- 2. Taping:**
Get a good amount of double-sided tape on the bottom side of your block, rub it on.
- 3. Positioning:**
Place your block on the milling table, making sure that is oriented correctly - the longer side of the milling table (X-Axis) corresponds to the X-Axis in Surfcam. Press it well against the table, if in doubt, step on it !



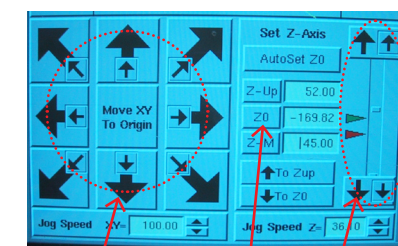
4. Mounting a Milling Bit



- 1. The bit:**
There is a multitude of different milling bits available. In the lab, we stock 12mm, 6mm and 3mm bits, both with a flat tip (Endmill) and a round tip (Ballnose). There are two types of collets, 12mm and 6mm diameter. Choose the right bit (as specified in SurfCam) and the fitting collet.
- 2. Positioning:**
Turn the collet until it starts to resist a little and insert the bit. Using your thumb as a marker, check that the bit is properly in the collet (at least the height x of the collet-ring). The bit should only stick out as much as is required by the particular milling job. Long bits tend to vibrate more (resulting in slower cutting speed).
- 3. Tightening:**
Using the special tools, tighten the collet firmly - do not overdo it as you will have to release the bit at a later stage.

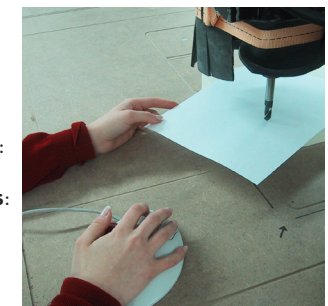
5. Setting Z-Zero & Material Thickness

NOTE THAT THESE ARE THE MOST IMPORTANT SETTINGS ON THE MILL. IF DONE CORRECTLY, THEY PREVENT THE MILL FROM MILLING ITSELF. IF DONE WRONG, THE MACHINE CAN DESTROY ITSELF AND INFLICT INJURY ON THE OPERATOR (YOU!). BEFORE YOU START ANY MILLING JOB, DOUBLE-CHECK THAT THESE SETTINGS ARE CORRECT.



XY-Arrows Z0 Z-Arrows

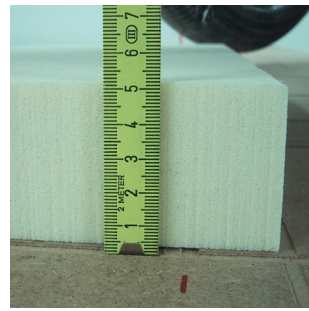
Large Arrows:
0.25mm
Small Arrows:
0.025mm



- 1. Adjust Z-Height:**
Using the XY-Arrows, move the mill over your piece of material. Using the Z-Arrows, move the mill down so it just touches the top surface (Use a sheet of paper on top of your material, move it gently, you will feel when the bit touches it).
- 2. Set Z-Height:**
Click "Z0" - your zero Z-Height is now set to the top of the material block.

!!! REPEAT THIS PROCESS EVERY TIME YOU CHANGE THE MILLING BIT !!!

6. Setting the Material Thickness



Z-M: enter thickness here
Material Thickness

1. Measure:

Measure the thickness of your material block.

2. Set Z-M:

Click the numeric field next to "Z-M" and enter the thickness value (mm) using the mouse. It is good practice to subtract a few millimeters from the measured thickness to make sure the mill will not go into the table.

Example: measured material thickness: 50mm
Z-M: 48 (mm)

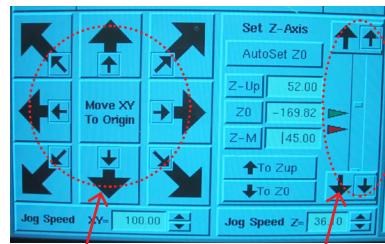
3. Check:

Make sure the value next to the "Z-M"-button is correct.

Using the XY-Arrows, move the mill away from the material. Using the Z-Arrows, slowly move the mill down. If set correctly (Example), the tip of the milling bit should be 2mm above the table when the mill stops.

IF SET CORRECTLY, THE "Z-M" PARAMETER PREVENTS THE MILL FROM GOING INTO THE TABLE.

7. Setting Z-Up & more



XY-Arrows

Z-Arrows

1. Z-Up:

Z-Up is the height above the material at which the mill moves around the table. Make sure the setting is GREATER than the RAPID PLANE Setting in SurfCam. If it is smaller, the mill might not process your job.

2. Setting Z-Up:

Following procedure described in 6., set the value for Z-Up.

3. more Buttons:

The "To Z-Up" and "To Z0" Buttons move the mill to the Z-Up or Z0 (top of the material) position. Be aware that pressing "To Z0" after changing a milling bit can destroy the material and the table as the mill will move to the "old Z0" setting which might not correspond to the new bit.

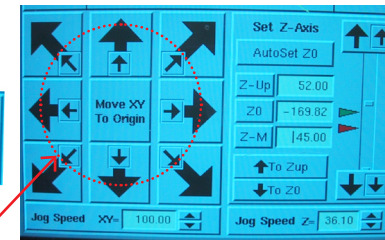
7. Setting XY-Origin

"Absolute", "Relative" changes coordinate display relative to the mill's origin (home) or to your origin.

	X	Y	Z
Absolute	0.00	0.00	0.00
Origin	379.91	256.90	-169.82

"Origin"

XY-Arrows



1. Move up:

Firstly, move the bit up 0.5mm (2 Clicks on the big Z-arrow).

2. XY:

Using the XY-arrows, position the mill at the corner of your material block.

3. Set Origin:

In the top left corner of the computer screen, click "Origin".

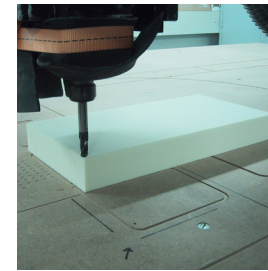
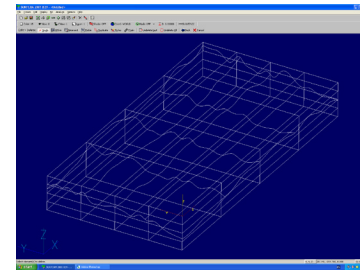
"Absolute": displays the position of the mill relative to the machine's origin (homed).

"Relative": displays the position of the mill relative to the Origin which you set.

4. Check:

At this stage, your piece on the table should correspond to your model in SurfCam.

Clicking "Move XY To Origin" and "To Z0" should position the bit as below:



8. Loading a Job & Preview

1. Load a job:

Clicking "Add Job To Queue", manage through the file system and select your job. Repeat as needed.

2. Select:

Highlight the job you wish to process and click "Select As Current Job", the file name appears in the small window "Current Job:".

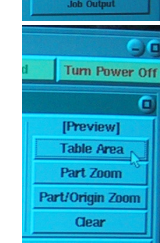
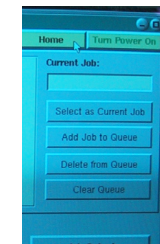
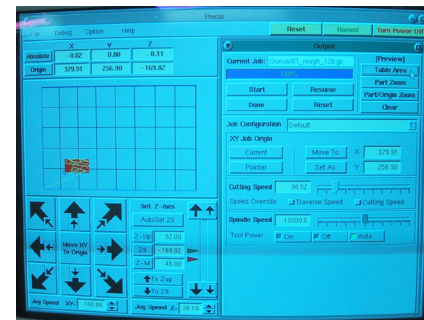
3. Output:

Click "Job Output".

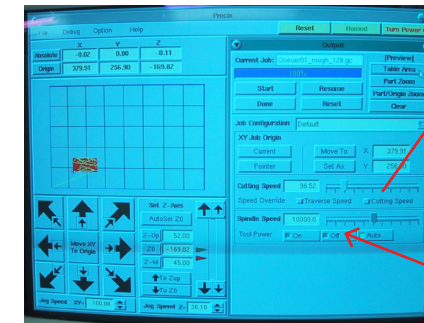
4. Preview:

There are different views which display your file as the Mill sees it. Check "Table Area" to see the piece in relation to the whole milling area (table). Make sure it corresponds to the block of material which is on the table. "Part / Origin Zoom" is a close-up view.

Hint: the mill will not start until you've done a preview.



9. Cutting Speed & Spindle Speed



Spindle Speed

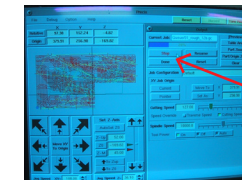
1. Cutting Speed:

Defines how fast the mill moves in X, Y and Z - to set it, you HAVE to select the check box "Cutting Speed" and adjust the value using the slider. To start, set it to 25-35. When the job has started and is processing smoothly, increase the value gradually. Note: a change will only take effect once the computer processes the next line of the G-Code, eg. when a new "line" or toolpath-part starts.

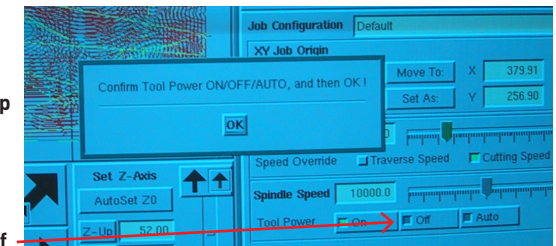
2. Spindle Speed:

The rotational speed of the milling bit (rotations per minute). Per default, "Auto" is selected, the value defined in SurfCam is taken.

10. Stopping & Resuming, Errors



Stop



Turn Tool Power Off

1. Stop:

If you are unhappy with how the mill works, click "Stop". In most cases, the mill will now just move to the "Z-Up" position and turn off the spindle.

2. Really Stop:

You are now prompted to "Confirm Tool Power ON/OFF/AUTO, and then OK!"

IMPORTANT: DO NOT CLICK "OK"!

First, you click "Tool Power: Off" - the spindle stops. Now click "OK".

3. Fixing:

Fix / adjust whatever was causing trouble.

4. Re-Start:

1. Click "Tool Power: On" - the spindle starts to rotate

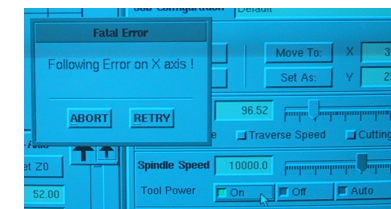
2. Click "Resume" (next to the "Stop" button)

3. You will get the "Confirm Tool Power ON/OFF/AUTO, and then OK!" - message as before, click "OK".

The mill continues the job.

5. Error:

The most common error is a "Following Error" (can happen for example when the G-Code tells the mill to go somewhere which is outside the table area, or when the Cutting Speed is set to high). Usually, it will stop the spindle and show the following dialog:



To resume, **FIRST** click the Tool Power to "ON", then click "RETRY".

BEFORE STARTING WORK, CAREFULLY READ THE SAFETY GUIDELINES POSTED IN THE LAB. DO NOT PROCEED UNTIL YOU HAVE PROPERLY UNDERSTOOD THEM.