FABLAB BRUSSELS WWW.FABLAB-BRUSSELS.BE

Tutorial CNC-milling part1

Making a cutting board with Inventor HSM/ Fusion360

Download pdf at http://www.fablab-brussels.be/fablab/tutorials-2/

Table of Contents

*The steps in **bold** must be performed at the machine: the remainder can be studied at home, before coming to the fablab or whilst waiting for your turn*

Edit log:

-2017-12-06 Lieven - added screenshots for inventor
I also added a number of picture (on machines, notes on drilling)
Part on actually using USBCNC (5.4 and following) still needs some work

-2017-11-20 Lieven - started adding images
-2017-11-20 Lieven - changed margins to wider margins & font size to 10
-2017-11-20 Lieven - I changed the GCODE appendix to a reference intermizzo in the text

small corrections still to make (then we can format for printout & someone can test Milling):
p15 - new screenshots with **Stepover** carefully chosen & "**Axial to Leave = 0**";
p20 - does Inventor now have the "**drill tip through bottom**" feature (instead of fudging bottom height)?
p24 - correct screenshot to show a **toolpath that mills the ENDS** (and not the sides);

Feeds & Speeds:

- **short explanation in tutorial so students will be able to mill different woods with a variety of endmills;**
- **a full explanation, either to insert in Tutorial 2 or as a separate .pdf booklet students can use as a reference (together with g-codes etc.).**

SHOW SOMEWHERE YOU CAN GET AN EXPLANATION ON EACH PARAMETER BY HOVERING OVER IT? *OK I'll paste that into a natural gap in the text AFTER page layout is complete*

About this tutorial

This tutorial is designed so that you can work independently. Look out for the 3 icons below:

	Here you should stop and ask a responsible person to look at your work or check it.
	This is an important point about safety.

It takes about 1,5 hours to complete this tutorial.

The tutorial assumes that you have either **Inventor HSM** or **Fusion360** installed on your own laptop and are familiar with its basic operation. The tutorial shows information for both pieces of software.

If you are unfamiliar with those programs, this tutorial will be of limited use to you.

If you do not have a laptop, you can work on one of the PCs in the FabLab or in Mechatronica.

1. What is CNC Milling?

CNC (computer numerical control) milling is a form of subtractive manufacturing that predates 3d-printing (computer controlled additive manufacturing) with which it has much in common.

The computer moves a rotating cutter (like a drill bit) to progressively remove material from the work piece until the desired result is achieved.
Just like in a 3d printer, movements are specified with gcodes.

There are many kinds of CNC machine, to deal with everything from large panels of (soft) wood to blocks of (very hard) metal.

Today you will be using a “CNC portal mill” in which an X axis gantry carrying the spindle moves over a large table platform. This mill has only modest rigidity but is ideal for wooden panels. As such it is equipped with a vacuum system for removing wood chips rather than a coolant/lubricant system better suite for cutting metal.

2. Process and Toolchain

	Step 1 - CAD	***Step 2 - CAM***	***Step 3 - Prepare the Machine***	***Step 4 - Run the Job***
	Design on your laptop using software like: *Fusion360 (makers' licence)* *Inventor (VUB student licence)* *(CAD = "Computer-Aided Design")*	*Design a milling strategy & generate a "CAM toolpath" - a file containing commands telling the cutter how & where to move.* *Save the tool path as "my_toolpath.cnc"* *(CAM = "Computer-Aided Machining")*	*Clamp the workpiece, home the machine and set the origin for the toolpaths.*	*Transfer mytoolpath.cnc to PC that will control the CNC mill or router (network or USB key) & import into control software .* *Our mills controlled by the software Eding USBCNC.* *The PC controls the motor controllers (stepper drivers) which in turn pilot the stepper motors.*

3. Configure Toolpaths

In this chapter you will configure the milling operations on your computer.

You do not need a CNC-machine yet.

You could prepare this chapter at home if you'ld like.

3.1 Download 3D-model: Cutting Board

In this tutorial you will mill a cutting board, like the one shown below:

Let's begin by downloading the model:

Go to **http://www.fablab-brussels.be/tutorialsfiles.html**

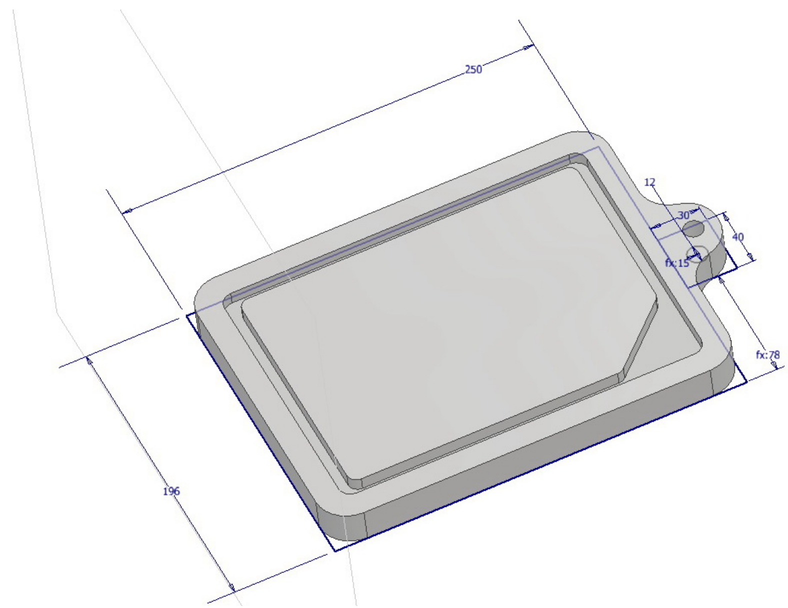
Download the file in your preferred format:

cutting_board.ipt (if you use Inventor)
cutting_board.fusion (if you are using Fusion360)

Inventor-users:: open the file in Inventor:

Everyone else: upload to the Fusion360 cloud and then open.

Have a look at the model, stepping through the timeline to understand its features, dimensions (the board, its holes, channels etc.) and how it was created.

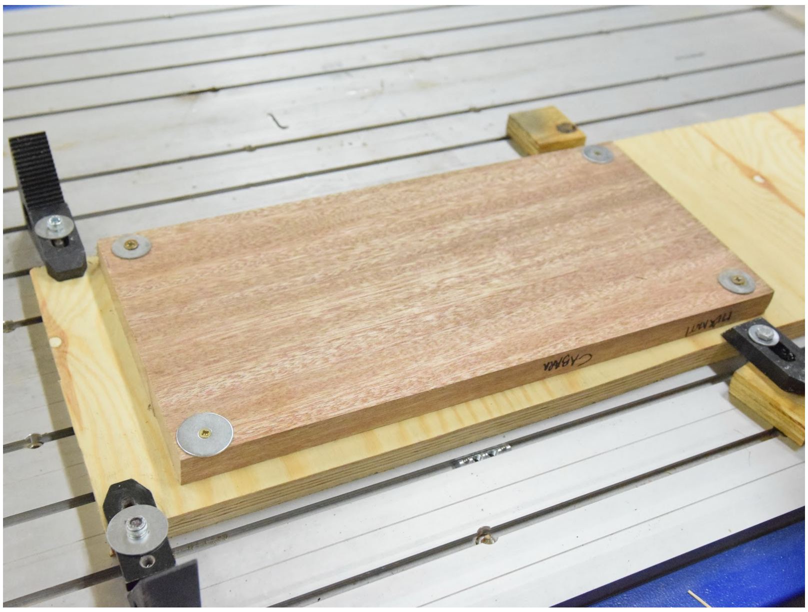


3.2 CAM - create a “setup”

Note:

Inventor and Fusion360 are both by Autodesk and share the same CAM module, though the same features may display differently.

To understand this chapter better, you see below a picture of the plank you will cut the board from in a later chapter.



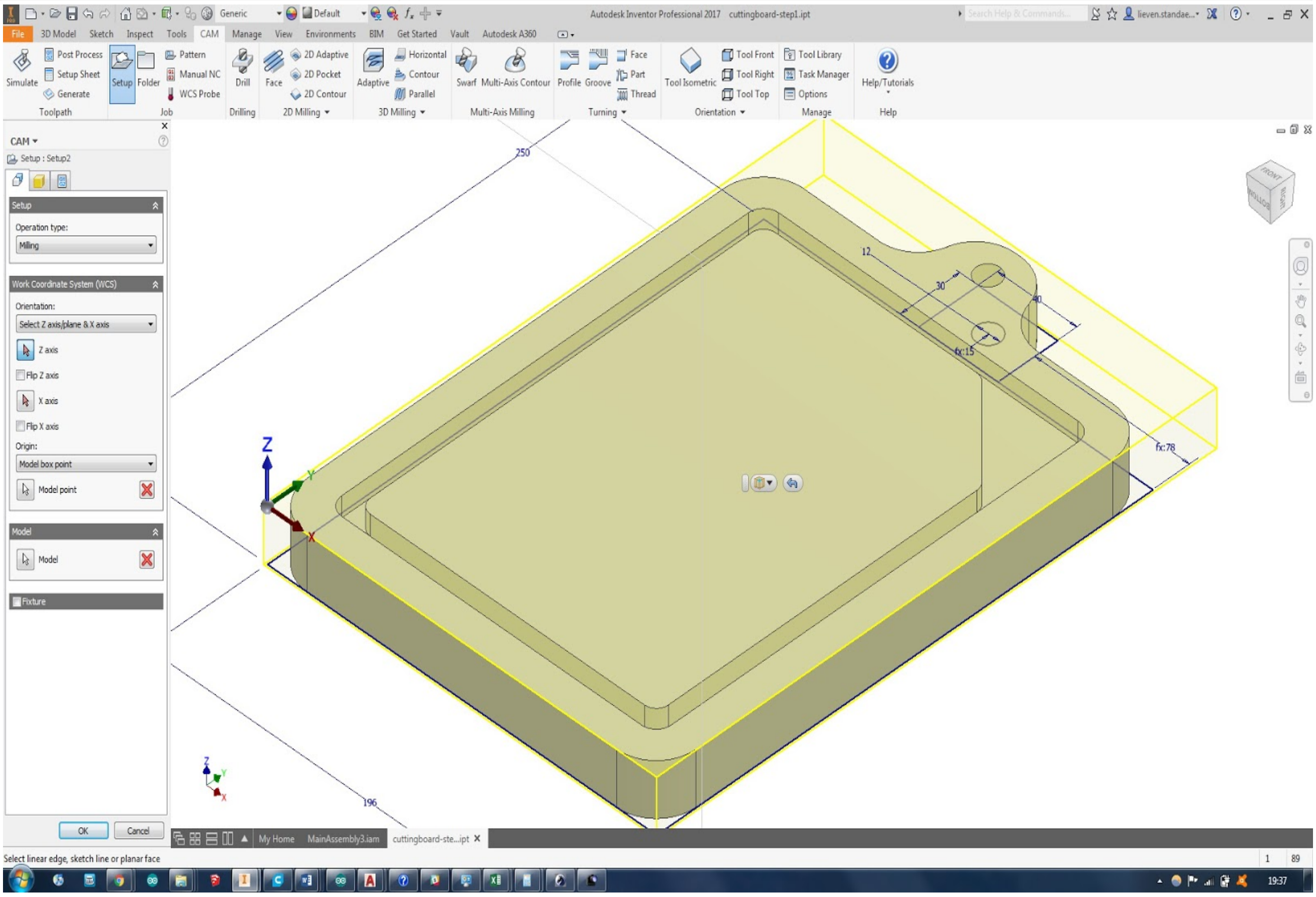
Note:

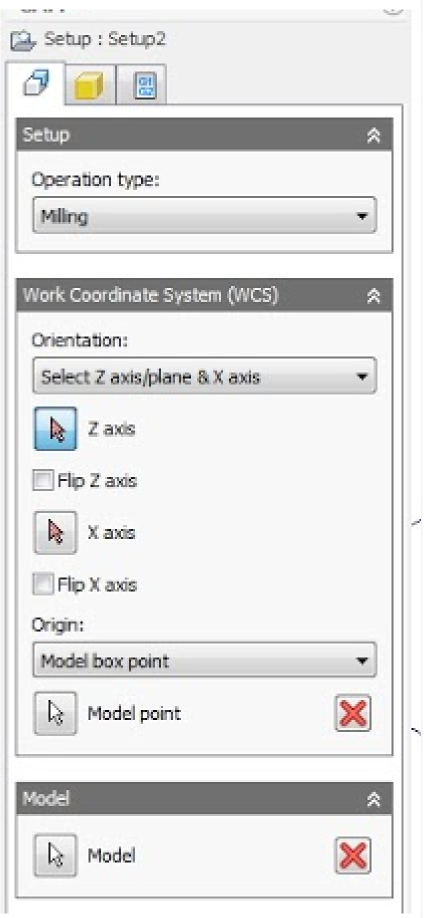
- the plank is fixed on a spoiler board (we don’t want to cut into the alu table)
- A part needs to be fixed in place: the board is held by 4 screws at the corners.
- The plank is about 100mm longer than the 3D-model.

You need to prepare the cutting toolpaths using the CAM module of your software package and the first step is to define the size and orientation of the stock material (the plank of wood).

Switch from the Modelling environment to the **CAM** module,
then click on “**Setup**” to create a new Setup.

You should see a panel similar to the one shown below.





First define the Origin (0,0,0) that will be used by the toolpaths and that we will set in the Mill.

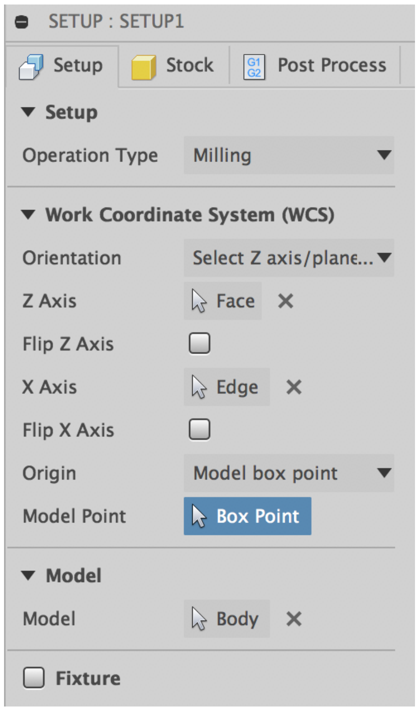
1. **Model**: ” select the bodies which are to be milled (this design contains only one);

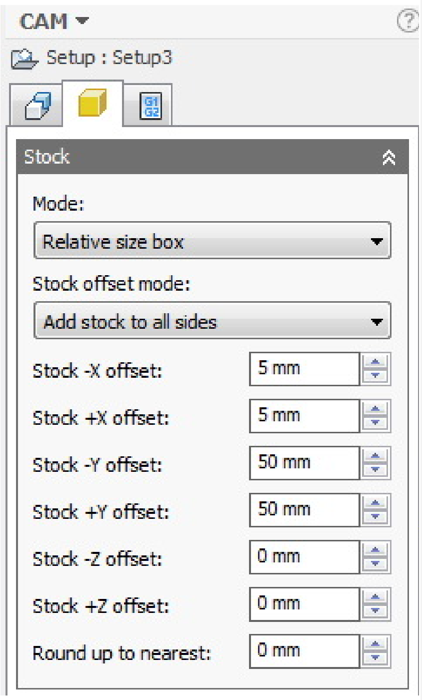
2. **Orientation**: select “Z axis plane and X axis” (this enables you to define the axis directions);

   Then click the top of the Part(to define the Z-axis plane) and one of the edges of the part parallel at the end of the plank (to define the X-axis).

3. **Origin**: select “Model box point” to enable the selection of a point on the model. Now identify one of the points on the top of the model that correspond to where the corner would have been if the cutting board had not been filleted. (See image right).

4. **Check against the screenshot** at the start of this section and, if necessary, go back and make changes.



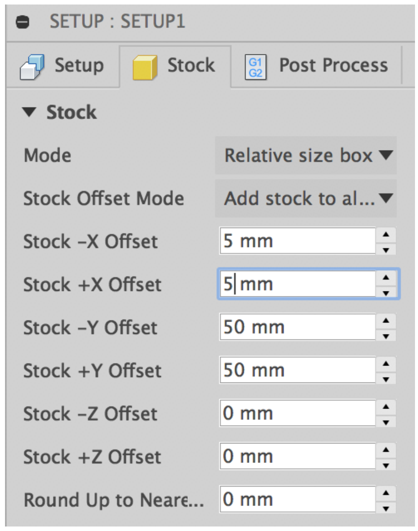


Now go to the **"Stock" tab:**

The 'Stock' is the piece of material from which you will cut the part.

1. Select "Relative Size Box" (the stock will be sized relative to the model);
2. **Set Y** to extend 50mm in each direction;
3. **Set X** to extend 5mm in each direction;
4. (Leave Z as 0mm;)
5. **Check the result** against the screenshot at the start of this section;

Now close the **Setup** window, select the "Setup" that you have created and examine the final result. You have created a plank of wood from which the cutting board will be created and the Origin has been defined from which the toolpaths will begin.



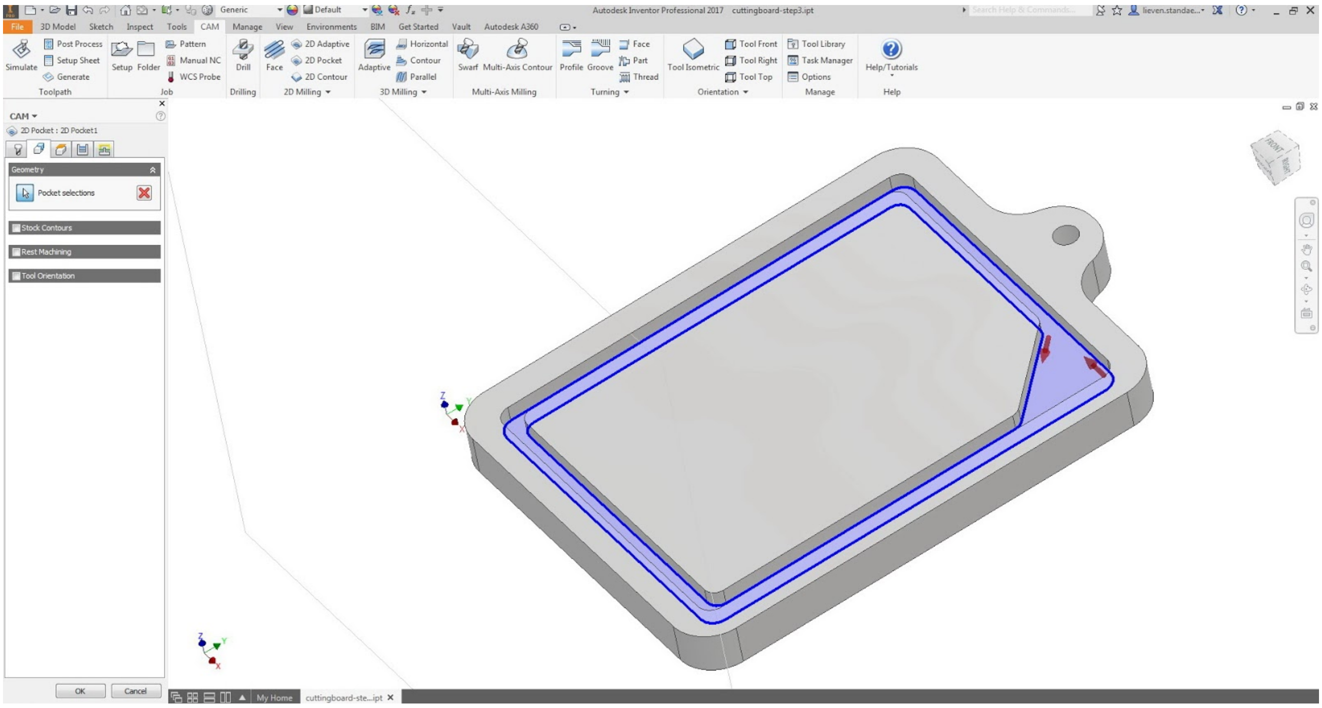
3.3 CAM “Pocket” toolpath - cut the gutter.

A pocket toolpath is used for removing pockets of material.

It is not the only way of doing this but it is the one we will use here, because the in this toolpath the cutter moves in a simple linear fashion and that is ideal for cutting out the long channel that runs around the cutting board.

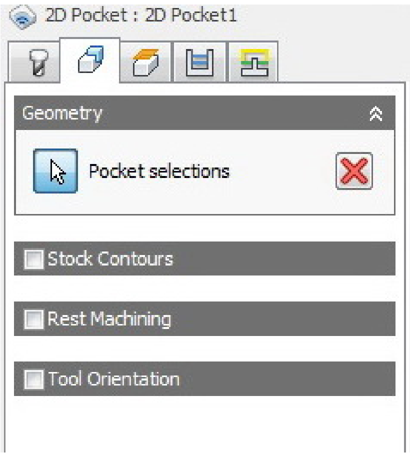
__“Pocket”__ toolpath is not well suited for harder materials.

When cutting into eg. aluminium, better use __“Adaptive”__



Click on “**2D Pocket**” toolpath to create: note that the toolpath is organised under the Setup that you created earlier.

Now we will configure the essential parameters. Many will be left as default. Try to develop an understanding of what you are doing as you go along.

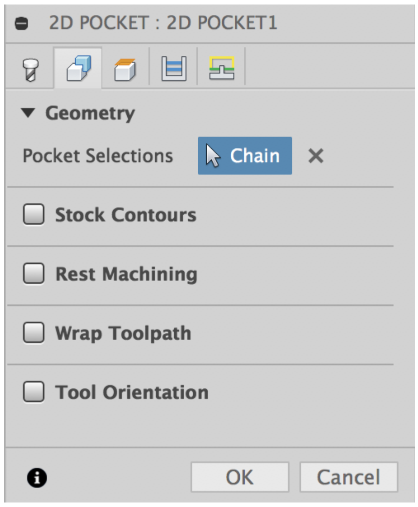


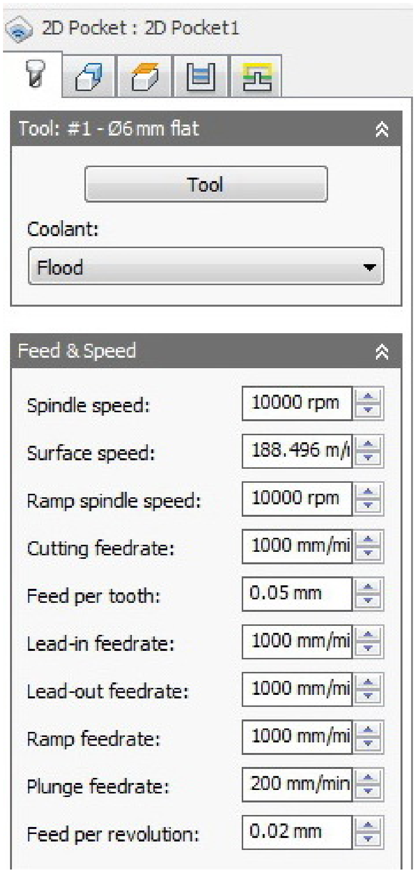
First we will select the Geometry that we want the cutter to reveal.

On the second tab **GEOMETRY**:

1. Select the bottom of the “swimming pool and channel” cavity on the top of the cutting board;

2. Ensure the selection looks like the photo above.





Now, on the first tab **TOOL** we will configure the “Feeds and Speeds” (see box).

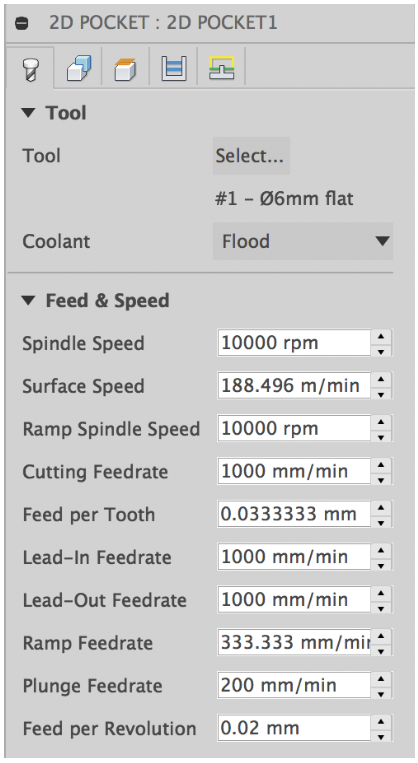
1. Tool: select a 6mm flat endmill;

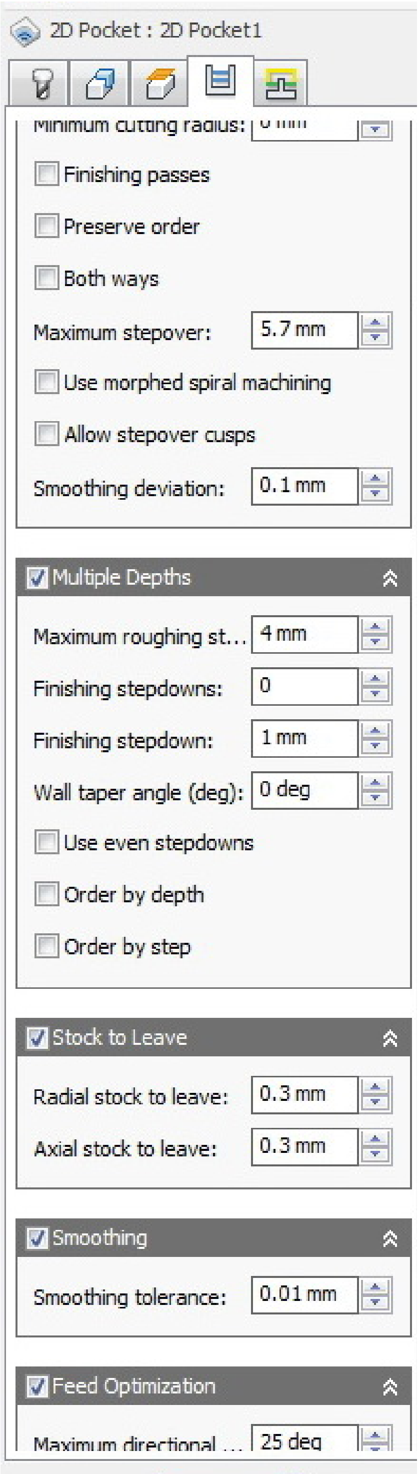
2. Set the spindle speed to 10,000rpm

3. Cutting Feedrate: set to 1.000mm/s;

4. Lead-in AND Lead-out: same as Cutting Feedrate (this ensures a smooth entry /exit)

5. Set Plunge speed to 200





Next we will specify that we want the cutter to cut down in layers (so as not to overload the machine).

On the fourth tab "**PASSES**":

1. Set **Maximum Stepover** to XXXXXXXXX
2. Select "**Multiple Depths**" checkbox;
3. Set "**Maximum Roughing Stepdown**" to 4mm;
4. Set (number of...) "**Finishing Stepdowns**" to 1;
5. Set "**Finishing Stepdown**" to 1mm;

Next we will configure "**Stock to Leave**" for this toolpath:

1. Set the the "Radial" to 0.3mm;
2. Set the "Axial" to 0mm;

"Stock to leave" means in this operation you will take off a little bit less than you should.

This is called a roughing operation.

You will take off the last bit in the next operation, where you put low loads on the cutter. This is called a finishing operation.

Select **"Smoothing"** with a tolerance of 0.01mm.. This makes the size of your final file smaller.

3.4 CAM “Contour” toolpath- do a finishing cut around the gutter.

Typically you will use 2 kinds of operations when machining a part:

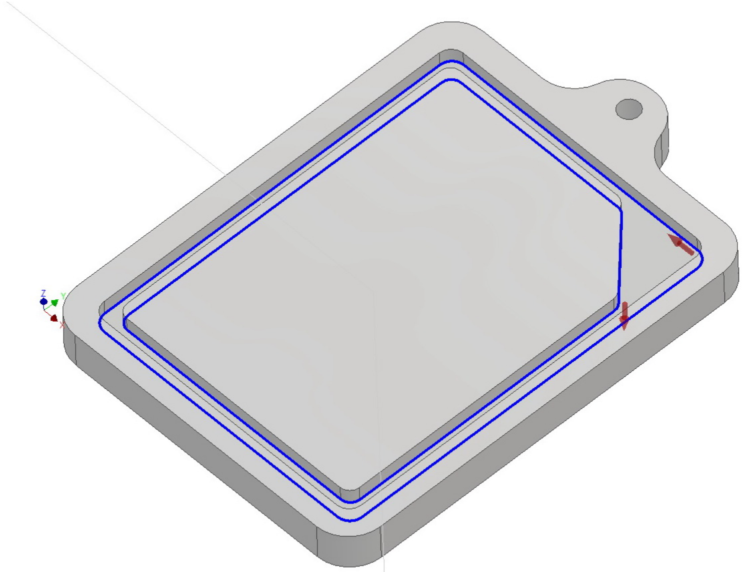
- A first **’roughing’** operation that does the hard work of removing most of the material.
    - You may even use an “old” less sharp cutter for this since the finish does not matter and you will deliberately leave a thin layer of stock on the part.

- A second **‘finishing’** operation in which you remove the remaining stock with very little load on the cutter.
    - The cutter can move quickly; there will be no deflection (of the bit) nor backlash (of the motion system) so your result will be more precise.
    - Using a cutter in good condition will achieve the best possible finish.

The previous toolpath was a roughing toolpath: now we need to finish this part of the cutting board, before moving onto the next feature. The pocket toolpath stepped down in layers so as not to put too much load on the cutter at any one time. If you look closely you’ll see that it left horizontal lines on the vertical surfaces.

Now we will make a single pass over to remove the material previously specified in the (radial) **“Stock to Leave”** dialogue.

Our toolpath will follow the contours below. The cutter has little material to remove and no resistance so it can - and should - move quickly.

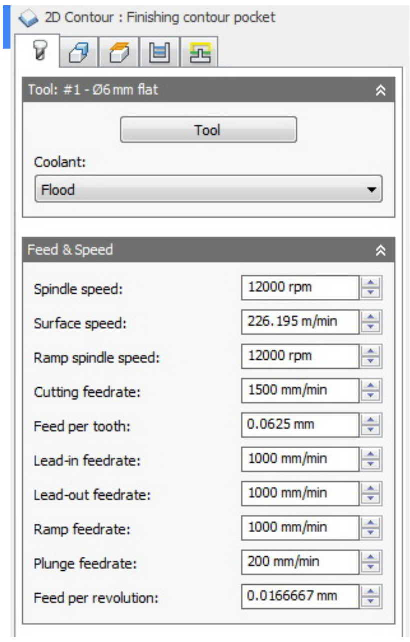


Click the icon for 2D toolpaths and select **“2D contour”** from the drop-down menu.

Now follow the detailed instructions, tab-by-tab, to configure Autodesk’s 2D-contour.

Configuring 2D-Contour (basics)

INVENTOR



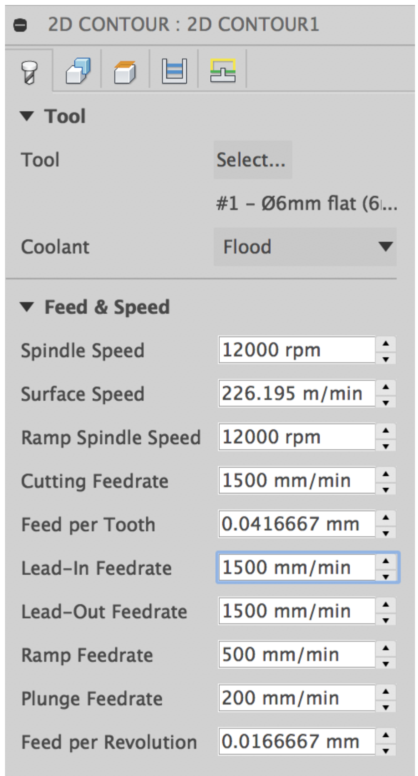
NOTE: We will leave many values as default.

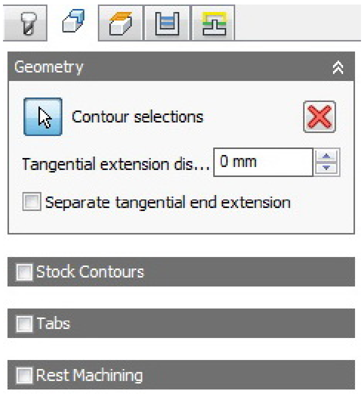
TOOL TAB (first - leftmost - tab)

Tool: 6mm flat
Cutting Feedrate 1.500mm/min
Lead-in AND Lead-out: same as Cutting Feedrate (this ensures a smooth entry /exit)

Spindle Speed 12.000 RPM
Plunge feedrate: 200mm/min

FUSION360





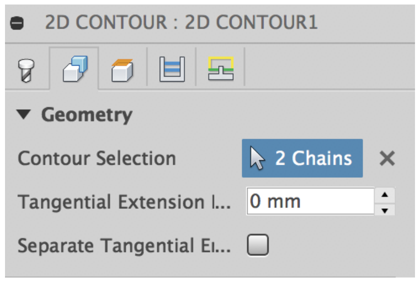
GEOMETRY TAB

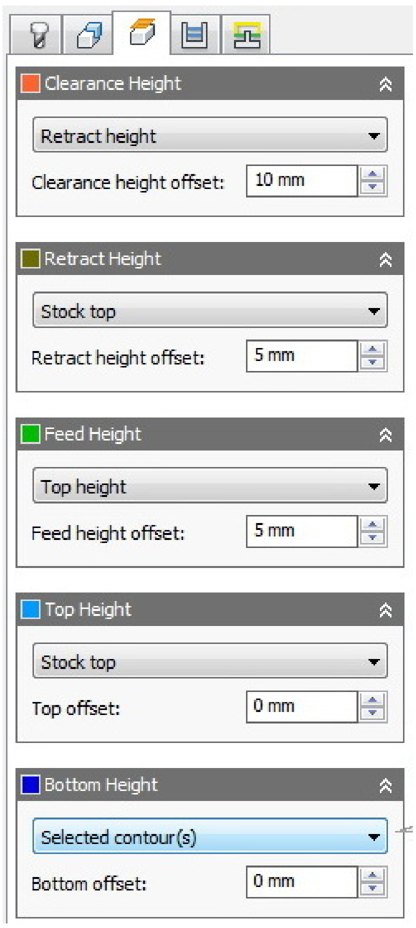
Select the Geometry that we want to mill:

1. Select both BOTTOM contours of the channel. (By selecting the contours at the bottom of the channel, we automatically tell the software what DEPTH to mill to.)

2. Verify both are selected as in the photo at the beginning of this section above.

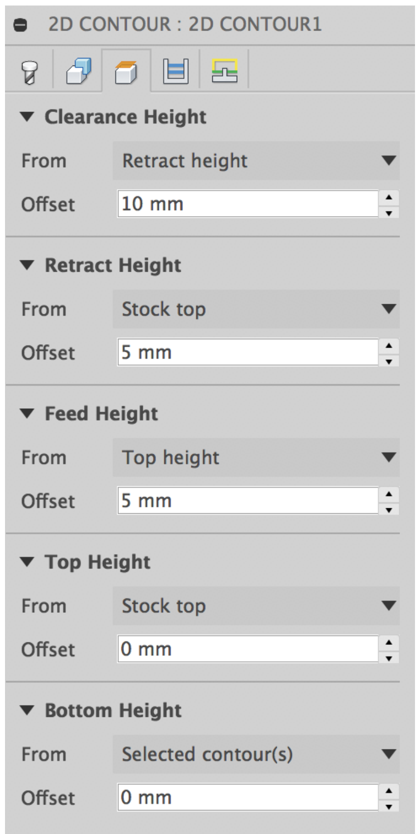
Do not check other boxes

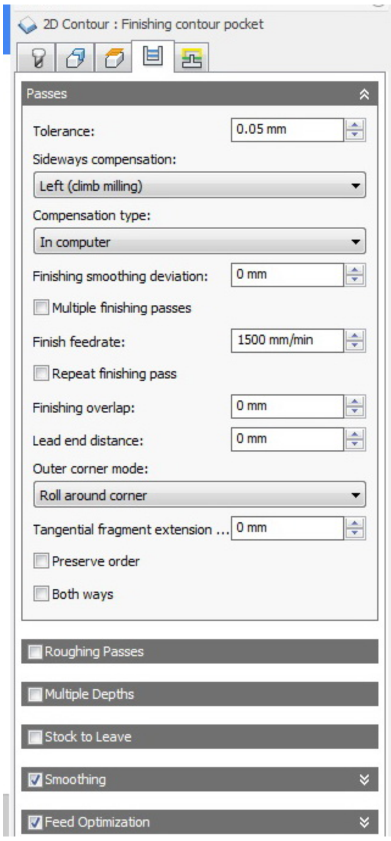




HEIGHTS TAB

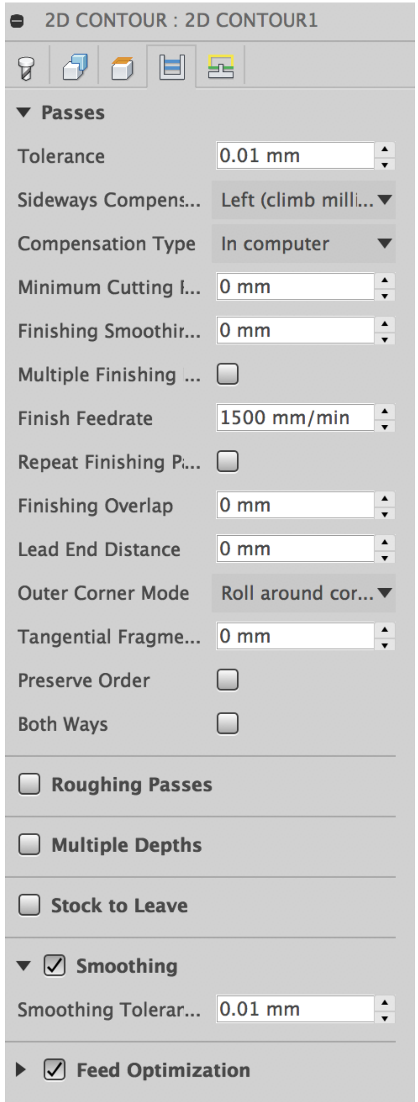
Clearance height: 10mm
Top Height: Stock top
Bottom Height: Selected contour(s)
Offset: 0mm





PASSES TAB

Uncheck 'Multiple Depths'
Uncheck 'Stock to leave'
Check 'Smoothing'
Check 'Feed optimisation'



LINKING TAB (the 5th tab)

Leave as-is

Click 'OK' at the bottom.
This exits the tab menu.

Doubleclick on the name of the operation and rename to:
"**Finishing Contour Pocket**"

3.5 CAM “Drill” - cut the hole in the board

You will now make the toolpath to drill the hole in the handle of the board.
There are 2 ways to go about drilling in CNC.

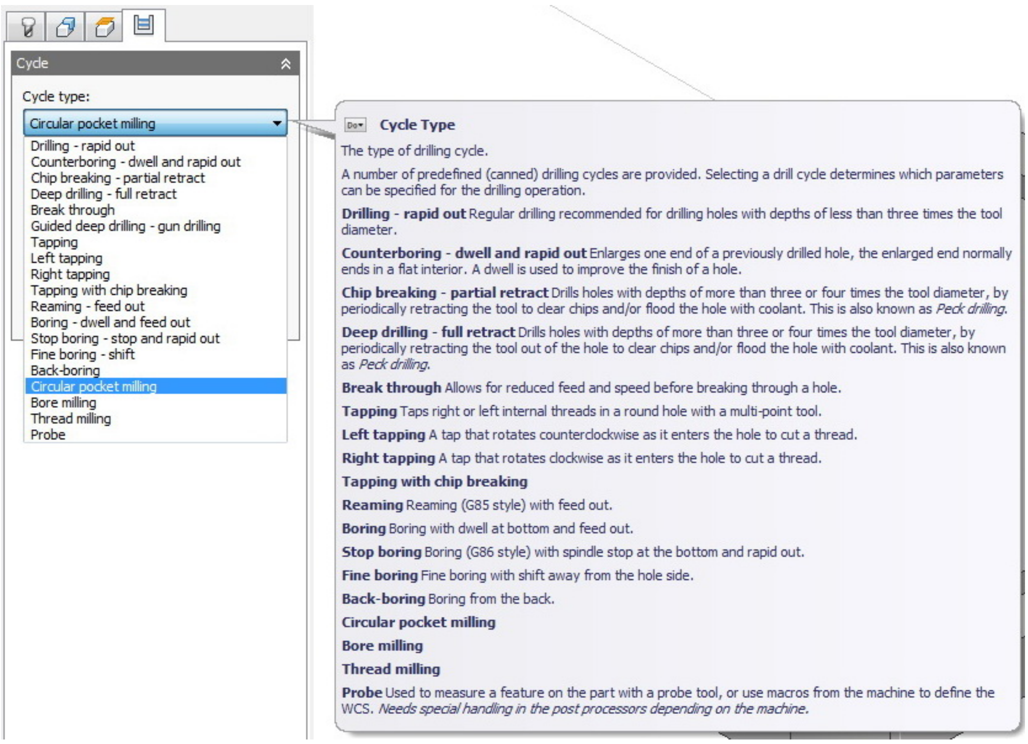
1-You can insert an actual drill bit and use the mill at a lower RPM.
2-You can try to drill using the regular flat endmill.

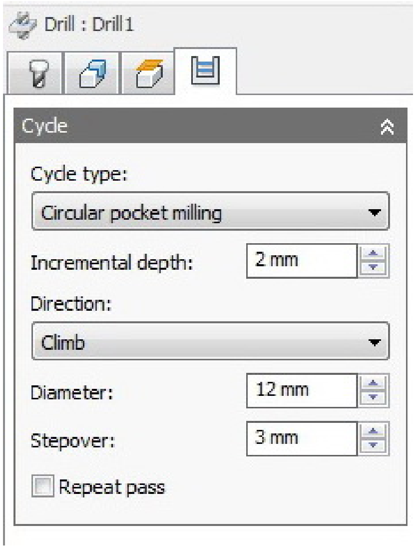
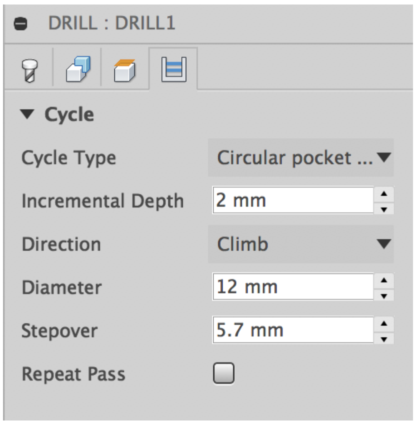
Both have their limitations, which are explained in the ‘Notes on drilling’ page.
We will here use the milling bit to drill.

- Find, select and create a **“Drill”** toolpath.
-
- In the **“Geometry” tab** select the hole in the board
- In the **“Tool” tab** select the same **6mm bit** you used for the previous toolpath
- In the **“Heights” tab** leave all values as is, except the last one:
  change **“Bottom height**” to **“Hole bottom**” with **“Offset”** set to **-2mm.** This makes sure we drill all the way through. (In Fusion there’s a new checkbox next to “hole bottom” called “**Drill Tip through Bottom**” followed by a box to fill “**Break-Through distance**” 2mm. Does Inventor have it????)
- In the last tab **“Cycle”** hover over the**”cycle type’”** dropdown. A popup explaining the different drilling modes appears.

When using a mill to drill holes,
either go for
“Deep drilling -full retract” *and do peck drilling*
(this means drilling a bit at a time, go up, drill a bit more))

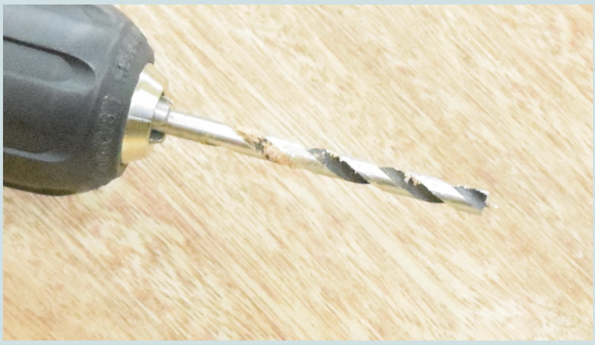
or choose ***“Circular pocket milling”***



	Choose “**Circular pocket milling**” , **direction “Climb”, diameter 12mm, Stepover 3mm** Choose “**Ok**”, exiting the tabs and rename the toolpath **“Drilling hole”**	

NOTES ON DRILLING :

DRILL BIT VS MILLING CUTTER



Drills only cut at the tip, the spirals are for chip evacuation. Trying to move sideways with a drill WILL break it.

The tip is typically a 60° point.

A drill is typically used at much lower RPM's than a milling bit.
Eg. an 8mm drill in a drill press would be run at about 500RPM.

Because the motors on our CNC-machines are light-weight, high-speed motors we cannot run at those speeds.

A dril bit will also not self-center, they might deflect when you try to drill a hole without using a center punch or center drill first.

You can get away with using a drill in our CNCs if:

-the drill bit is 6mm or smaller.

-the spindle speed is around 4000 RPM (motor will not have torque at lower RPM - smaller drill bits can be run faster)

-do peck drilling and move in & out agressively not to burn the drill bit.
Staying in one place too long WILL overheat the drill bit.

A **flat endmill** can drill, however:

-on the light CNC machines we cannot use the same pressure a drill press uses.
-we also are running at much higher RPMs.
-cutters with a straight edge clear chips less well than spiral cutters.

Trying to drill down in one go, as you might with a drill will result in the motor stalling, or the mill bit overheating.

You will typically use **peck drilling**, drilling eg 1mm at a time, go up, dril a bit more, then repeat.

A different approach, for larger holes is to mill small circles. This is called **circular pocket milling.**

It can be noted that using these techniques the resulting hole is more accurate than when executed with a drill.

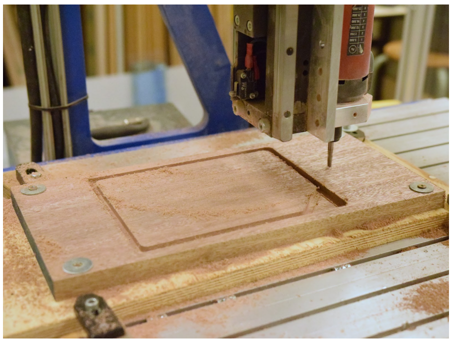
Ballnose cutters are terrible at drilling. Do not use these for this purpose

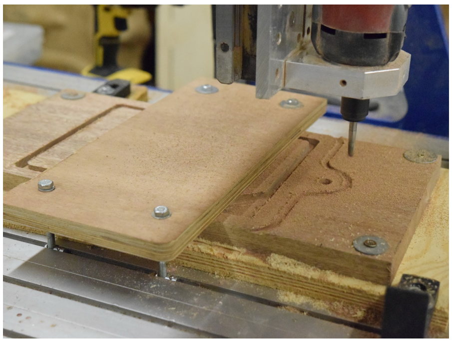
3.6 Problem: cutting the outside of the board

We now wish to cut the outside of the cutting board.

This introduces a complication:

if you cut out the board you will separate it from the screws holding it in place. (see image on the left)





This will result in the board moving, getting damaged by the cutter or simply be thrown at your head by the milling cutter.
ALL PARTS NEED TO BE FIXED IN PLACE AT ALL TIME.

Figuring out how to do this will depend on your part & can get complicated.

Here we will solve the problem by:

- Milling the sides of the board
- Adding additional clamping across the board (see image on the right)
- Then milling the top and the bottom of the board, separating the board from the screws while the clamp keeps it in place.

So the contour will be cut in 2 separate steps.

3.6 CAM "Contour" - cutting ends (rough)

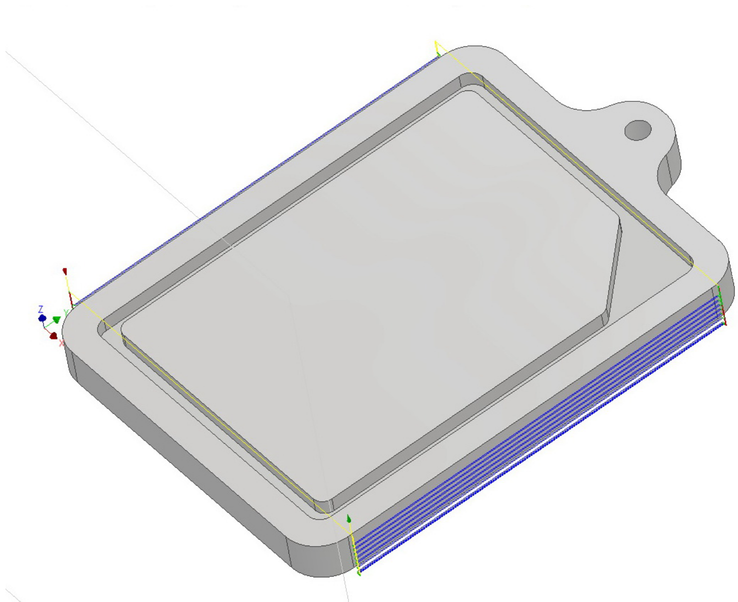
Click on the 2D menu and select "2D contour"

The toolpath will appear pre-populated with the last cutter you configured - the 6mm flat endmill.

If you need help setting the parameters, please refer to the guide with screenshots for the first contour toolpath that we did.

TOOL TAB
Tool: 6mm flat
Cutting Feedrate 1.500mm/min
Lead-in AND Lead-out: same as Cutting Feedrate (this ensures a smooth entry /exit)

Spindle Speed 12.000 RPM
Plunge feedrate: 200mm/min



GEOMETRY TAB

Select the Geometry that we want to mill:

- select the bottom contour of the part that runs around each end. NOTE you will need to select this in three parts, the fillet (arc) at each corner and the straight section in between.
- verify selection corresponds to the screenshot above. **You should only be milling the sides and be using multiple depths.**

	Do not check other boxes **HEIGHTS TAB** Clearance height: 10mm Top Height: Stock top Bottom Height: Selected contour(s) Offset: 0mm **PASSES TAB** Check ‘Multiple Depths’ - 4mm Check ‘Stock to leave’ - 0.3mm Check ‘Smoothing’ Check ‘Feed optimisation’ **LINKING TAB** Leave as-is Click ‘OK’ at the bottom. This exits the tab menu. Label the toolpath “Roughing Contour Ends”

3.7 CAM “Contour” finishing the ends

This toolpath will cut the board free from the screws holding it in place. It will be run after a clamp is added as shown in 3.6.

Click on the 2D menu and select “2D contour”

The toolpath will appear pre-populated with the last cutter you configured - the 6mm flat endmill.

If you need help setting the parameters, please refer to the guide with screenshots for the first contour toolpath that we did.

TOOL TAB
Tool: 6mm flat
Cutting Feedrate 1.500mm/min
Lead-in AND Lead-out: same as Cutting Feedrate (this ensures a smooth entry /exit)

Spindle Speed 12.000 RPM
Plunge feedrate: 200mm/min

GEOMETRY TAB

Select the Geometry that we want to mill:

- select the bottom contour of the part that runs around each end. NOTE you will need to select this in three parts, the fillet (arc) at each corner and the strange section in between.
- verify selection corresponds to the screenshot above.

Do not check other boxes

HEIGHTS TAB

Clearance height: 10mm
Top Height: Stock top
Bottom Height: Selected contour(s)
Offset: 0mm

PASSES TAB

Uncheck ‘Multiple Depths’
Uncheck ‘Stock to leave’
Check ‘Smoothing’
Check ‘Feed optimisation’

LINKING TAB

Leave as-is

Click ‘OK’ at the bottom.
This exits the tab menu.

Label the toolpath “Finishing Contour Ends”

3.8 CAM - Exporting the Toolpaths as .cnc files

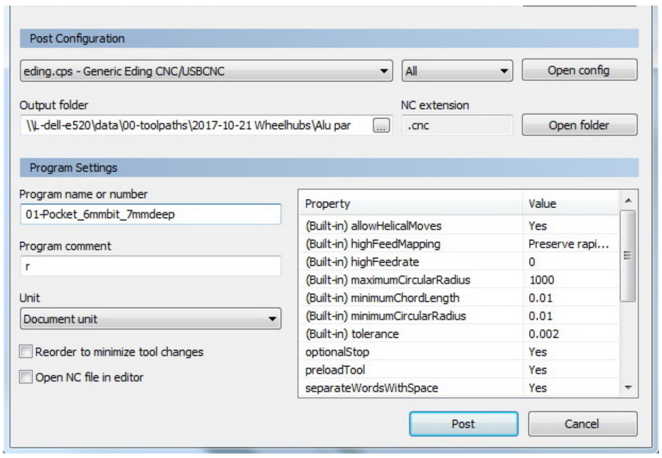
In order to transfer the toolpaths to the computer that controls the CNC router we must save them as .cnc files.

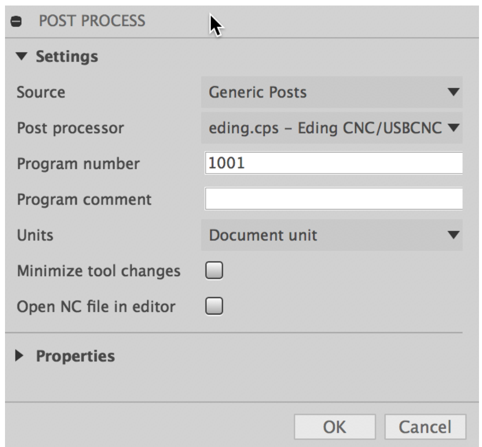
Whilst it's possible to save multiple toolpaths in a single file, it's safer and clearer to keep them separate so as to be able to execute one at a time (or to make modifications to one as necessary).

Before exporting it's necessary to select the appropriate **Post-Processor.**

A Post-Processor is analogous to a Print Driver and it ensures that the file created is in an appropriate Gcode dialect to be read without errors by the control software of the CNC machine you have selected.

All our CNC machines are piloted by software called "**Eding USBCNC**" and so we will select the **"Eding"** Post Processor.





1. **Select the first toolpath "Pocket";**
2. Click on the **Post Process icon** in the main toolbar**;**
3. **Select the Post-Processor drop down** and find the **"Eding CNC"** post processor in the list (see screenshot above - once you have made this selection it will remain as a default on your computer);
4. **Click "Save";**
5. Name the file that you are creating: **"1-Pocket-6mmcutter-7mmdeep.cnc"** (a descriptive file name will help you identify it later when you are operating the machine).
6. **Save to a folder** with a view to transferring via USB stick, network drive or other.

Now repeat for the other toolpaths:
"2-Finishing_Contour_6mm bit",
"3-Hole_6mmbit_22deep".
4 -
5 -

You want to:

	• use number prefixes so you will know in which order to execute the files; • make the names descriptive; it is good to include the cutter (many jobs require different endmills for different toolpaths) and adding the maximal depth cut to serves as an additional check.

4.0 Now pick a machine….

If you prepared the first chapters at home, or haven't said hello to a lab supervisor yet, this is a good time to do so.

Find a lab supervisor.

Ask which machine you can use to do the next part of the tutorial.

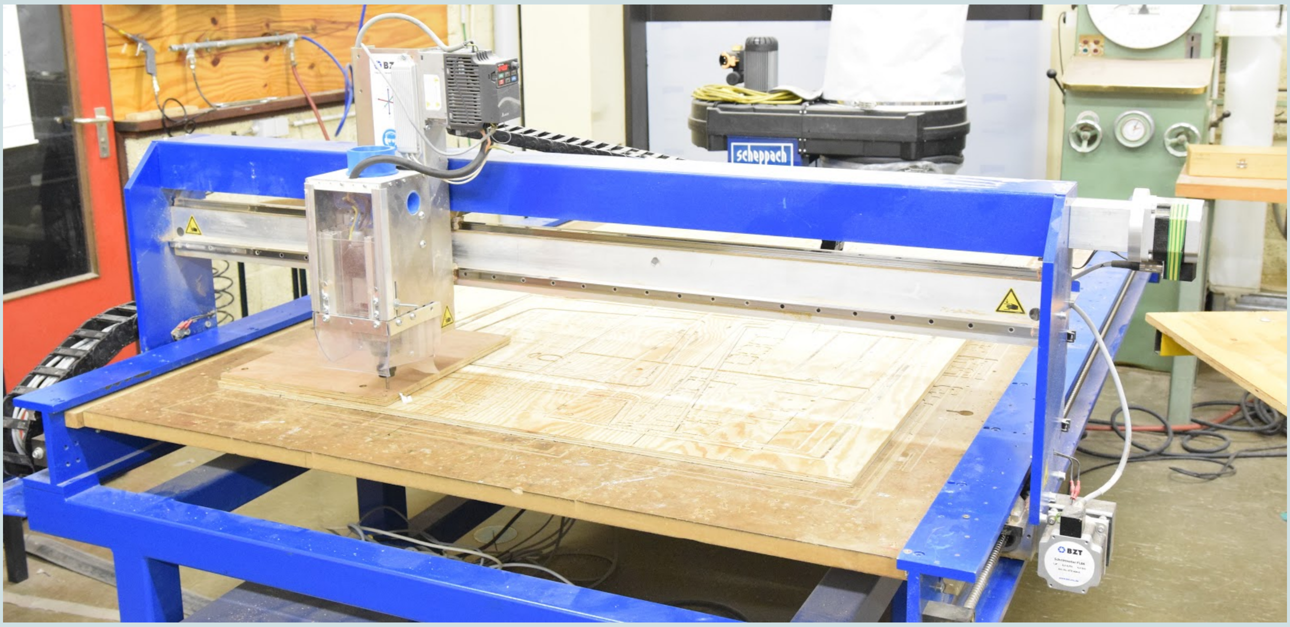
CHECKLIST LAB SUPERVISOR

-Assign a machine to the lab user.

-Explain the differences between the different machines, especially the differences between the spindle motor control and how to clamp work. See the following chapter for reference.

-Provide a 6mm milling bit and a piece of the correct plank to make the tutorial board. (20mm thick, 200mm wide Meranti)

-Show the lab user where he can find the safety equipment & advise to use hearing protection.



Large BZT panel mill

This panel mill has a work area of 1,500x2,500mm and a Z-height of 120mm.

We use is a lot for milling plywood sheets (18mm) and polyurethane foam blocks (pink) for molds.

It is not equipped to do aluminium, as it does not have a mist-coolant system.

Work area: 1.500x2.500x120mm

Accuracy: 0,1mm to 0,2mm depending how you work.

Spindle: starts automatically, speed set in software. The spindle controller has a bug in it which makes it start when the machine is turned on. Hit 'F1' in USBCNC to stop it.

Spindle speeds: 5,000-20,000 RPM
Milling bit sizes: up to 12mm, ER20 collets on table

Weight: 600kg

Collet wrenches nr: 20 and 30

PICTURE OF CLAMPING

PICTURE OF VACUUM CLEANER AND SYSTEM

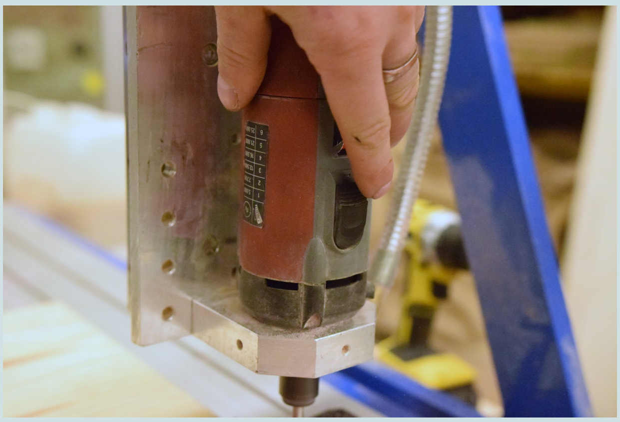
The bed is a thick sheet of MDF. You mount material on the bed using wood screws.

You MUST use a sacrificial board as shown in the picture: NEVER mill into the MDF bed!

The small BZT

This machine has a high Z-axis, so can do taller object. This also means it is the least rigid machine in our machine park.

Is equipped with a spray-mist coolant system.

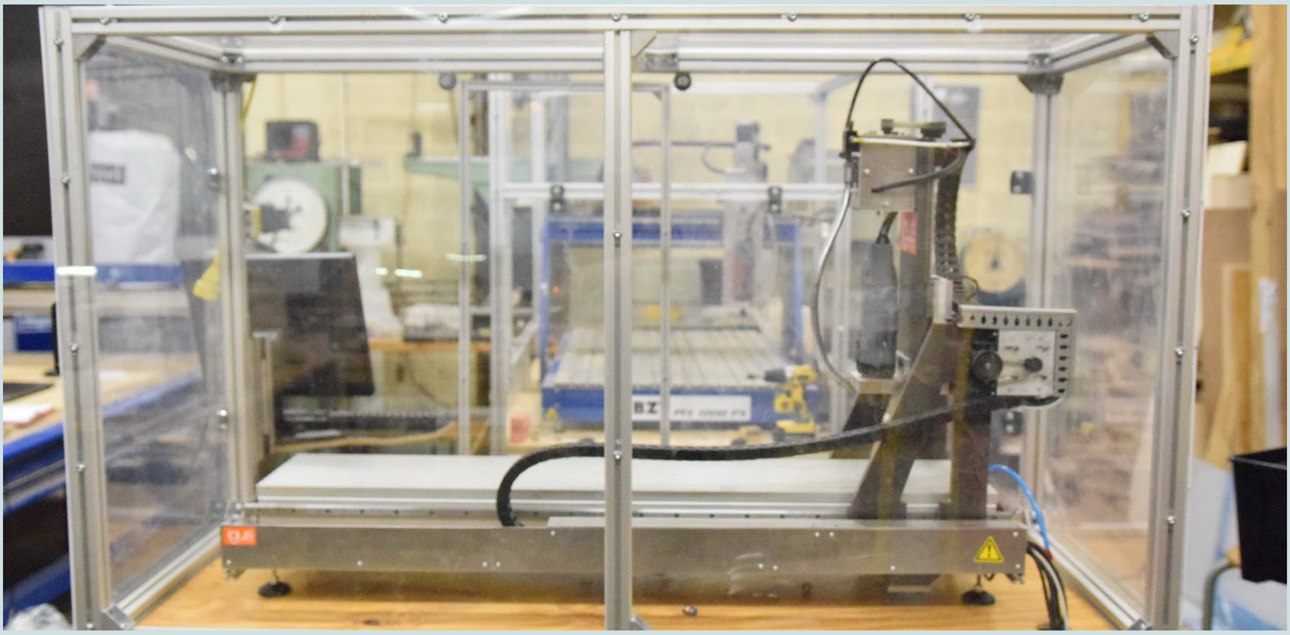


Work area: 1000x700x250mm
Axis: 3
Accuracy: 0,1mm to 0,3mm depending how aggressive you mill.

Spindle: needs to be started by hand (see picture), speed set by hand on Kress tool.

Spindle speeds: 5.000-20.000 RPM
Milling bit sizes: up to 8mm
Weight: 160kg

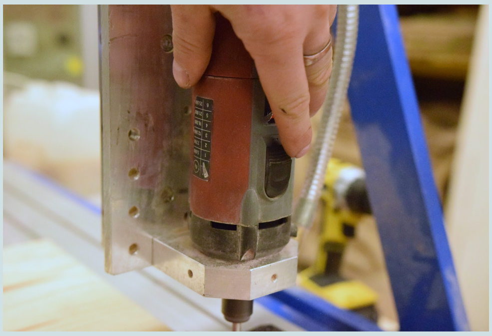
Clamping system:
smaller nuts in slot table, not compatible with ISEL or CAM2
Collet wrenches nr: 14 and 17



CAM2 MILLING MACHINE

This machine was built by engineering students Nick & Julien.

Is it more rigid than the small, blue BZT.
It is equipped with a coolant system and can mill aluminium.



Picture above: switch on tool should be locked in bottom position

add picture clamping system

Work area: 154x1000x120mm
Axis: 3

Accuracy: 0,05mm to 0,2mm depending how aggressive you mill.

Spindle: starts automatically, if the tool is locked in the ‘ON’ setting (see picture), speed set by hand on Kress tool.

Spindle speeds: 5.000-20.000 RPM
Milling bit sizes: up to 8mm
Weight: 130kg

Clamping system:
M6 MOTEDIS nuts in slot table, same as the ISEL

Collet wrenches nr: 14 and 17

The ISEL MILL

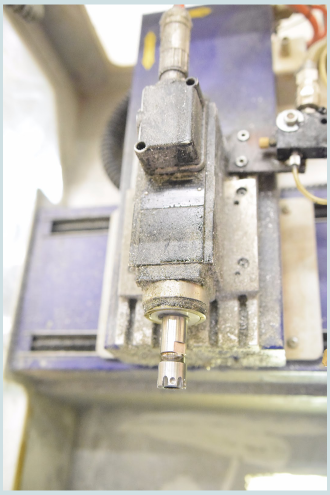
This is the most rigid of our light-weight CNC machines. If you need a part to be accurate to within 0,05mm then this is your best choice.

This machine also has an optional (small) rotary 4th axis.

Work area: 300x400x120mm
Axis: 3, 4th one available if needed

Accuracy: 0,05mm to 0,2mm depending how aggressive you mill.

Spindle: starts automatically, speed set in software.

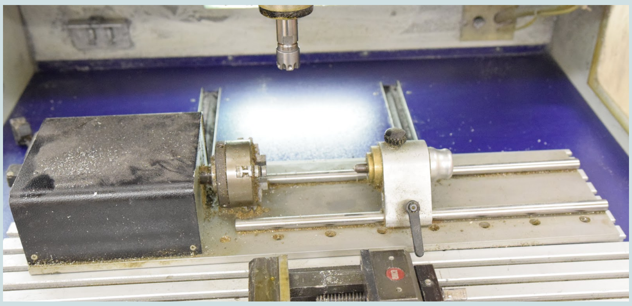


Spindle speeds: 5.000-20.000 RPM
Milling bit sizes: up to 8mm, ER16 collets next to machine

Weight: 140kg

Clamping system:
M6 MOTEDIS nuts in slot table,

Collet wrenches nr: 17 and custom ‘hook’ wrench on toolboard



Optional 4th axis (rotational) installed.

HARTFORD VMC

This machine is only available to experienced users. If the other machines are small Toyota's, this is a DAF truck...

It will mill you a part up to 0,05mm tolerances at 10 times the speed of the light machines.

It has flood coolant and can mill steel.

It has a 5kW low-speed spindle.
It has an automatic toolchanger, that has not been configured yet.

Work area: approx 500x250x300mm

Accuracy: 0,05mm depending how meticulous you work.

Spindle speeds: 50-5.000 RPM
Milling bit sizes: up to 20mm, toolholders on table next to machine.

Weight:3.500kg

MACHINE-SPECIFIC NOTES:

ISEL:

On the ISEL the controller is built-in. The machine is normally on. The green 'POWER' button on the front should be lit up.

5. USBCNC - machine controller

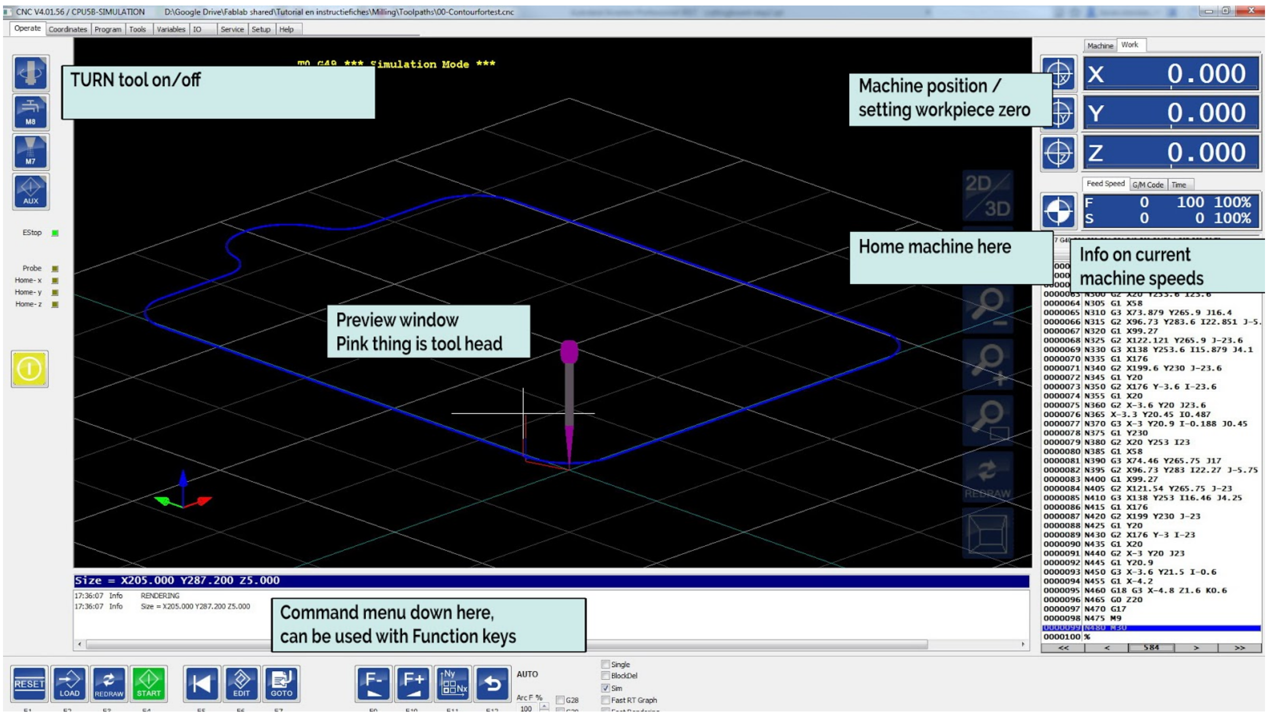
- Locate **'USBCNC'** on the desktop of the pc at the machine and doubleclick.

The software should open and you should be able to get to the window shown below.

5.1 Overview

Below is a quick overview of the USBCNC interface.
All our CNC-machines use this same controller.

All functionality can be accessed with the menu buttons at the bottom of the screen.
You can use the 12 function keys to navigate through this menu.



After the software has started up:

- Ensure that the bed is free of objects **-** there should be nothing that the spindle might collide with. If there are tool on the bed of the machine, remove these first.
- Locate the machine controller box. It is a large blue or grey box under the machine.
- Find the ON switch and turn it on.
- In the software, press **'F1'** twice. This powers the motors.
(on the large BZT this resets the spindle)
- Check if you are connected. Pressing one of the arrow keys should move the toolhead. If this is not the case, find a lab supervisor.

5.2 Home the Machine

Have a look at the previous page 'Overview interface' to locate the Home button

When you power off the machine, it does not remember the position of its toolhead.

Therefor, when you power it up, you should first **'Home'** the machine.

Find and press the **'Home sequence'** button.

The machine will move to one corner (and raise the head) slowly, until it hits 3 switches, one for each axis X, Y, Z.

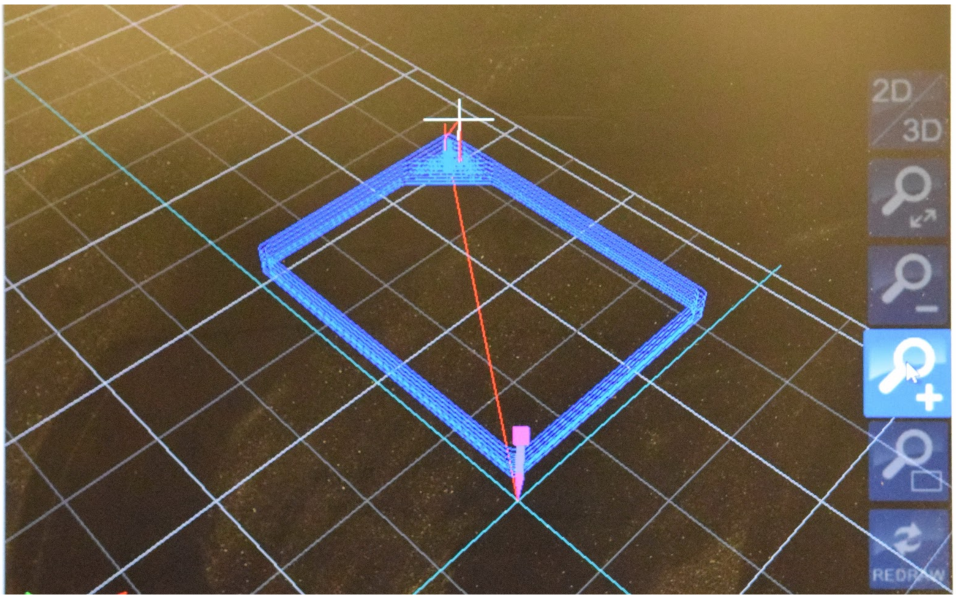
When the machine has finished homing you will see the **Machine Coordinates display (top right of the interface)** as 0,0,0.

Typically a CNC machine is interested in two coordinate systems (in fact there can be many more). The first are the coordinates that describe the physical volume of the machine. These are the Machine Coordinates. The second coordinate system of importance are the WorkSpace Coordinates (WCS). The WorkSpace Coordinates are centred at the origin that you placed on the stock material when you created

5.3 Open your first file

- Copy the toolpath files you made in chapter 3 to a folder on the machine computer

- Now go to “F4 AUTO” , select “F2 load” and open **“1-Pocket-6mmcutter-7mmdeep.cnc”,** your first toolpath

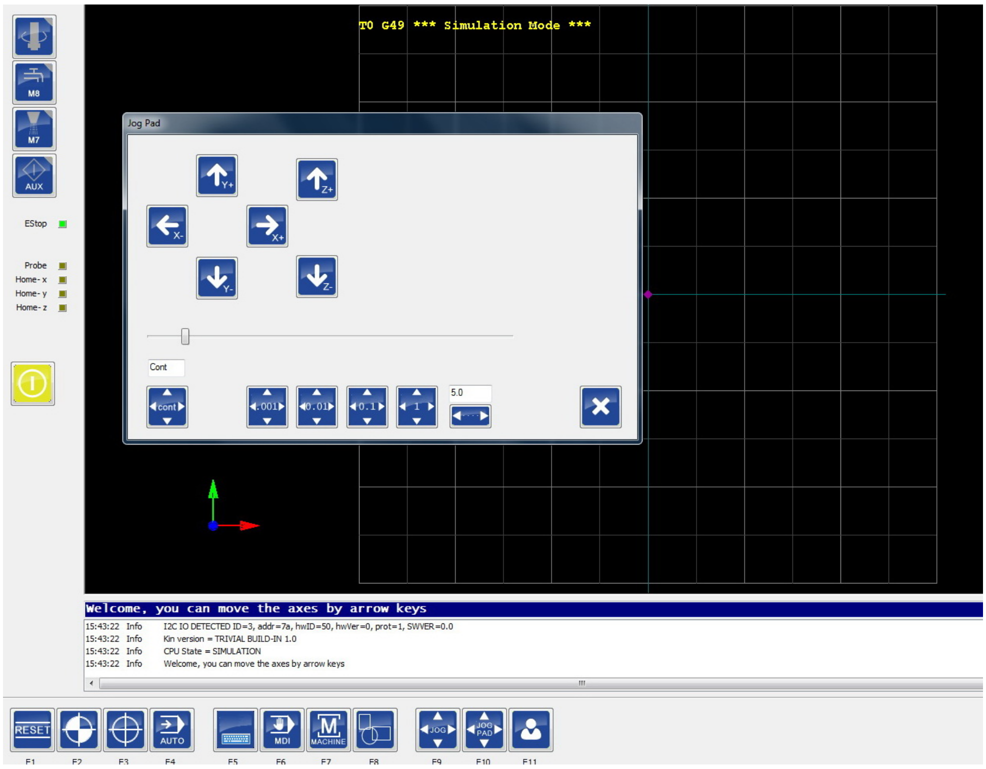


The main window should look something like the image above.

Note:

- You can zoom/pan with the buttons on the right
- You can also zoom/pan with different combinations of CTR or SHIFT or ALT with left or right mouse button.
- You can see the position of the toolhead as a pink pencil. It should be somewhere on the machine bed (not necessarily in the correct position yet)

5.4 Move the toolhead



You can move the toolhead using:

- the arrow keys of pageup/pagedown
- shift + the arrow keys of pageup/pagedown to move fast.
- F10 Jog pad to move accurately (the window looks like the image above

	5.4 Move the testset origin? mount tool

REFERENCE: G-CODE COMMANDS

A tool path is a long series of g-codes that tell the mill everything it needs to do in order to make the part. G-codes were once typed individually by operators of milling machines to perform single operations.

You'll still find it useful to know a few g-codes. Below is a short reference of some useful commands.

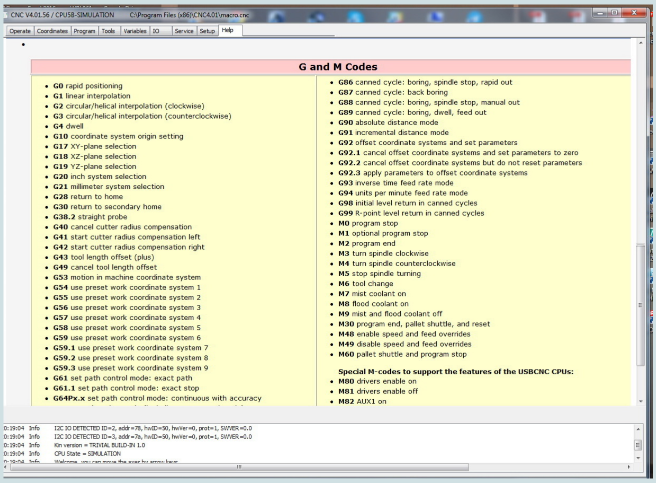
movement
G00 - rapid move (to move the cutter quickly from one place to another at a predefined maximum speed);

G01 - interpolated move (to move the cutter at a feedrate (speed) you specify);
F100 - set feedrate to 100mm/s;

spindle
S 2000 **-** set spindle speed to 2000 rpm;
M3 - spindle ON clockwise (the normal direction for cutting);
M4 - spindle ON anti-clockwise (you won't use this!);
M5 - spindle OFF;

coordinate systems
G90 - use absolute coordinates for commands;
G91 - use relative coordinates for commands;
G92 - assign the coordinates specified to the current physical position of the mill;
e.g. G92 Z0 will define the current vertical position as Z = 0;

examples:
S12000 M3 **-** set the spindle speed and turn it on (clockwise);
G90 work in absolute coordinates
G01 X200 Y30 F400 move to (200, 30) at 400mm/s
G91 now switch to relative coordinates
G01 X10 move +10 in the X direction from wherever you are now
Z-5 F60 drill down 5mm from where you are now, at 60mm/s
Note not necessary to type G01 again for the Z move **-** the current command was remembered.



5.5 Try some G-codes

Toolpaths are comprised of a long series of g-code commands. However you can also type G-codes manually and it is important to learn how to do this. There is a reference of some common useful G-codes in Appendix 2 at the end of this tutorial.

Click the button “**Machine Control”**.

Then click “**MDI**”.

At the command line prompt type:

G1 X300 Y400 F200

Double check your typing and then hit “enter”.

The spindle will move to (X=300, Y=400) at a speed of 200mm/s. “F” is short for “Feed Speed”: that’s the speed at which the part is fed into the cutter (cutter into the part - it’s all relative).

Now type:

M3 S10000

Once again double check and then hit “enter”. The spindle will start to turn at 1000rpm in a clockwise direction. “S” is short for “Speed”. Thus with F and S we talk about “Feeds and Speeds” as the basic parameters of CNC operation.

Now type G1 X400 Y100 (enter)

With the spindle turning it will move to the new coordinates at a speed of 200mm/s (the F200 typed earlier remains valid until a new feed speed is specified). If there was a workpiece present you can imagine that the cutter would already be shaping it. This is exactly how a CNC operates, according to the toolpath Gcodes.

Now let us stop the spindle using a button on the interface.

Press Spindle Stop.

6 - Prepare the Setup

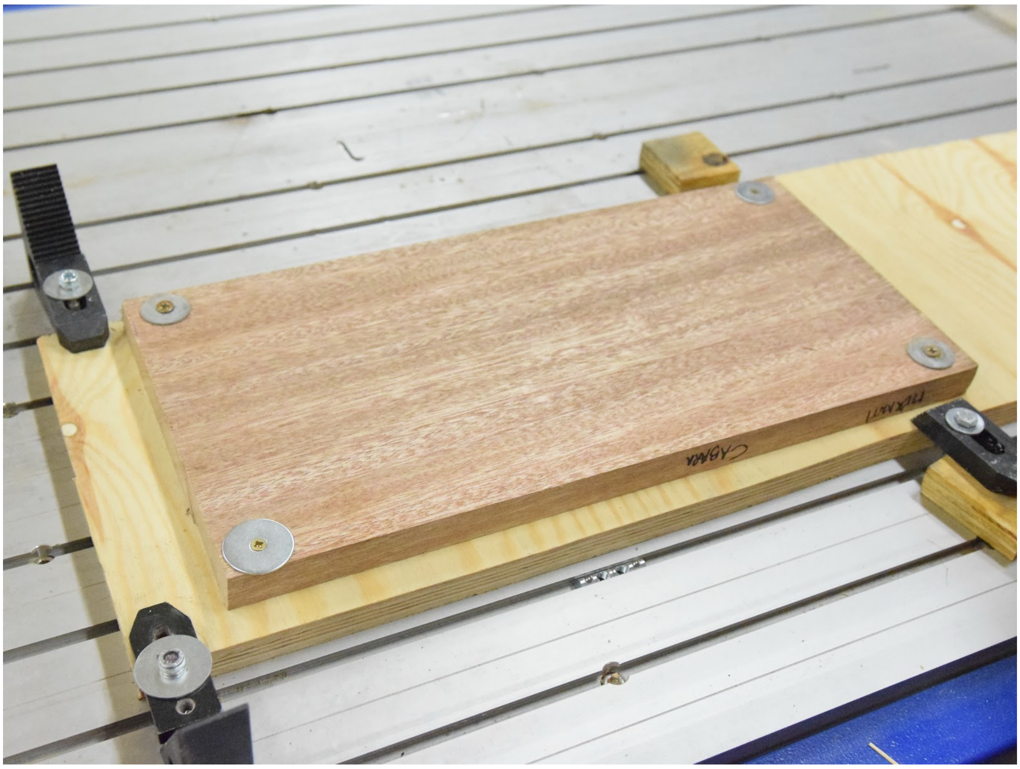
Be smart and use ear protection and safety goggles when using the saws.

Cut a 500mm length of 220 * 22mm board.

With a pencil, create a soft line 60mm from one end: this will represent the start of the tray.

JUST PASTED THE IMAGES, Haven't adjusted the text yet

6.1 Mounting the board



Picture shows the stock mounted on the smaller BZT mill.

On the large panel mill:

Take a sheet of sacrificial wood circa 18mm thick and larger than the workpiece; place it on the bed of the router.

Fix the sacrificial board to the bed with a screw at each corner. It works best if first you drill a hole in the board before placing the screw.

On the small panel mill:

If you are working on a machine with an aluminium bed, fix the sacrificial board using T-nuts….

Now align the workpiece so that it is parallel to the Y-axis (the "lengthwise axis") of the router. Because of the approach we are about to take it is important to align the workpiece as well as possible.

Now drill a hole in each corner of the workpiece (about 20mm in from each side) and place a screw through into the sacrificial board.

7.1 Mount the correct endmill

As you programmed in the toolpaths, for this tutorial you will be using a 6mm flat endmill.

So for the next steps you will need:

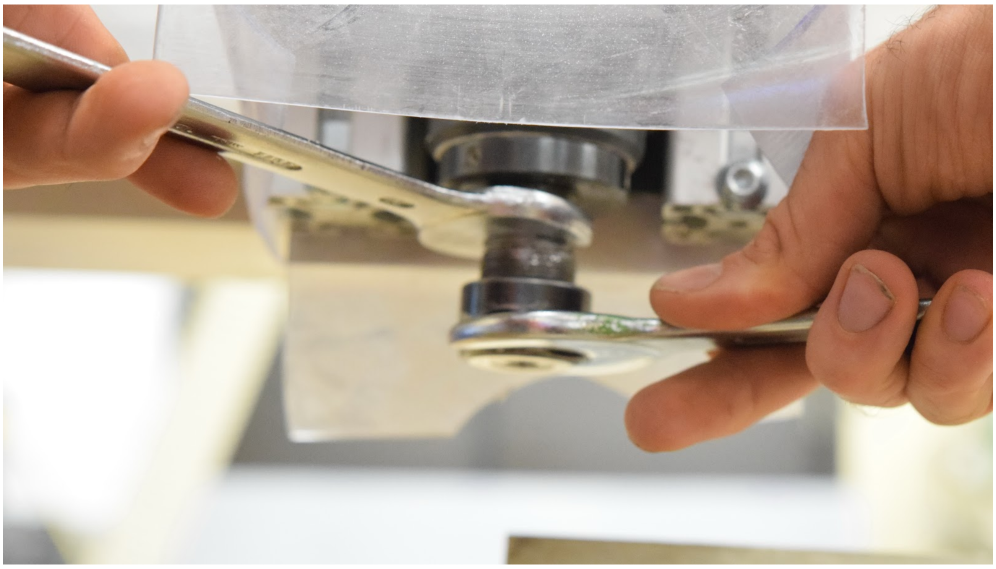
CNC machines use collet chucks. Which is to say you must first locate the right collet for your endmill, then fit it into the chuck, then insert the endmill and tighten.

First, using the Jogpad (see below) manoeuvre the spindle until it is in a place where you will have convenient access.

NOTE: movement with the Jogpad may seem a bit slow. There are also keyboard shortcuts you can use.
Arrow keys = movements in X-Y plane
PgUp-PgDn = movements in Z direction
Ctrl key - when held before pressing one of the above keys will result in a Rapid motion

Now disassemble the chuck

REVIEW THIS TEXT



(on the large BZT you will need two spanners to turn in opposing directions).

Examine the collet and, if necessary, find a 6mm collet and reassemble the chuck.

Insert endmill.

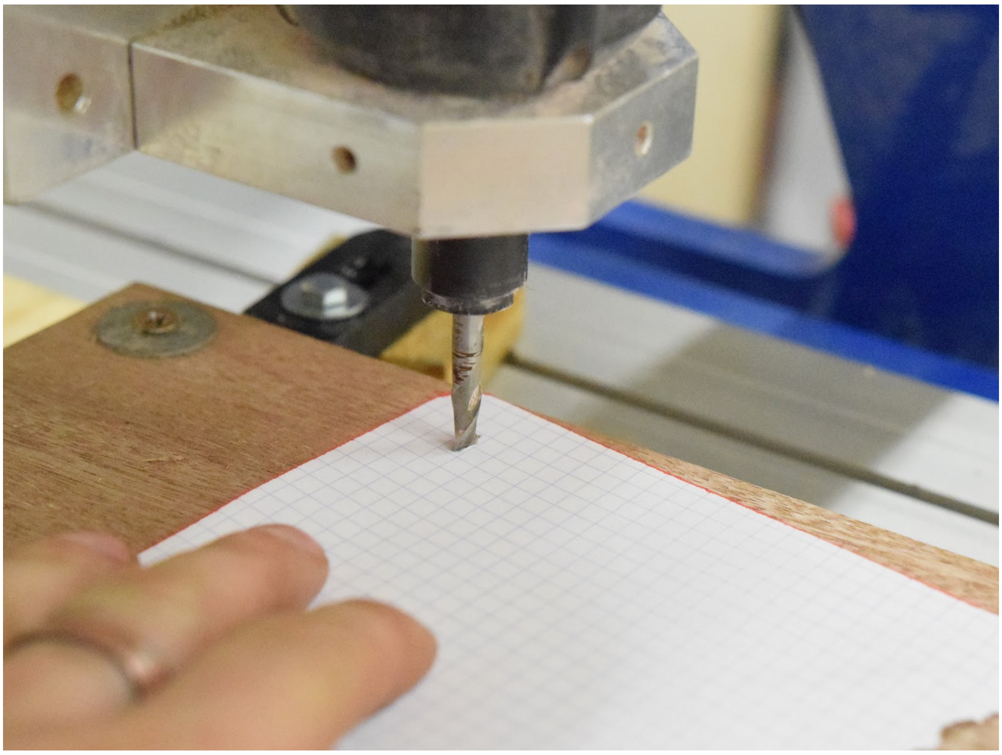
	Note: for reasons of vibration and displacement, when milling the less the “stick out” the better. So insert the endmill as deeply as possible - with the proviso that there must be sufficient stickout to mill to the depth required plus a margin of, say 5-10mm). In this case the part is 22m thick, we will mill to a depth of 1mm below that, so the ideal stickout would be 22mm + 1mm + 5-10mm = 28-33mm When you are satisfied with the Stickout, tighten the endmill in place.

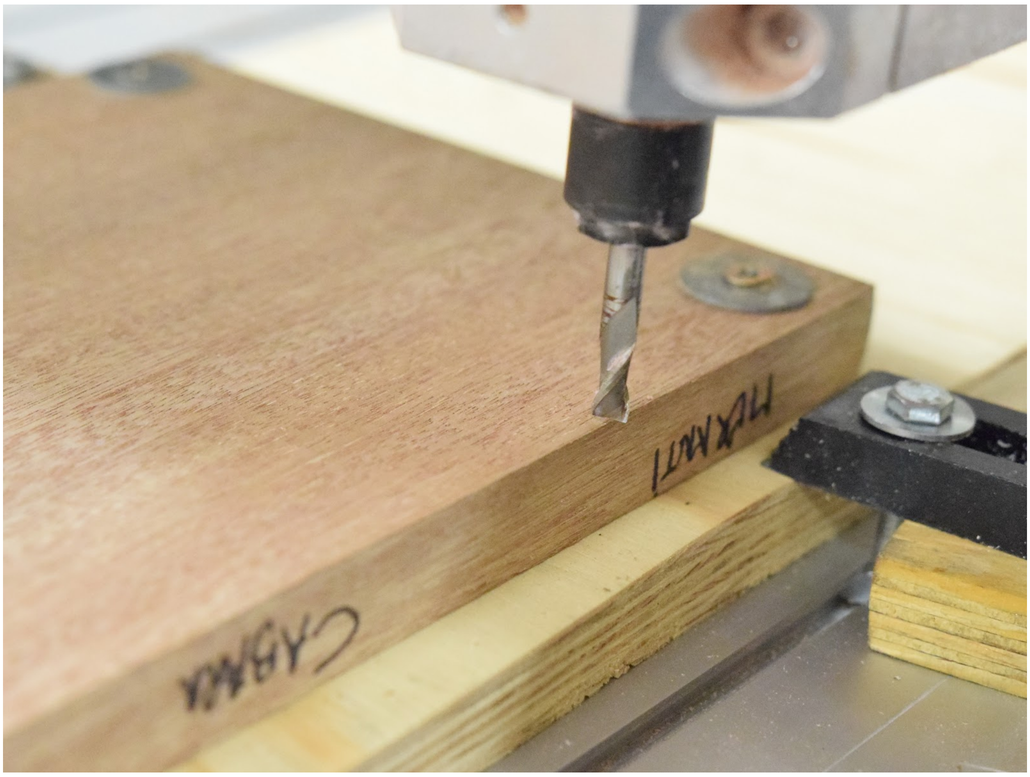
7.2 Set Workpiece Origin

Next let's set the Workpiece Origin.

Remember that when we programmed the toolpaths there was in each case an Origin placed on the stock material? THAT is the Workpiece Origin! Now that the endmill has been tightened in place we must tell the CNC where that origin is with respect to the tip of the endmill.

Mark with a pencil a small line on the stock 50mm from one end as shown below.





Now using keyboard shortcuts (see above) manoeuvre the spindle until it is a few centimetres from the mark.

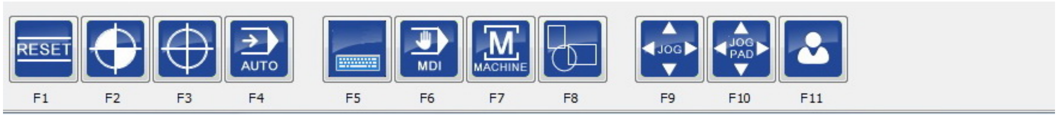
Then, use the JogPad to move in with precision. Notice that on the Jogpad controls you can choose the amount you would like the spindle to move with each button press. As the

	endmill approaches its destination, reduce the movement to 1mm intervals, then to 0.1mm. In this way there is no danger of a crash. We will measure off 3 different positions corresponding to X, Y and Z axes. SPLIT THIS UP IN SETTING ZERO Z WITH PIECE OF PAPER, THEN SET X AND Y (see pictures)

8 - Run First Toolpaths

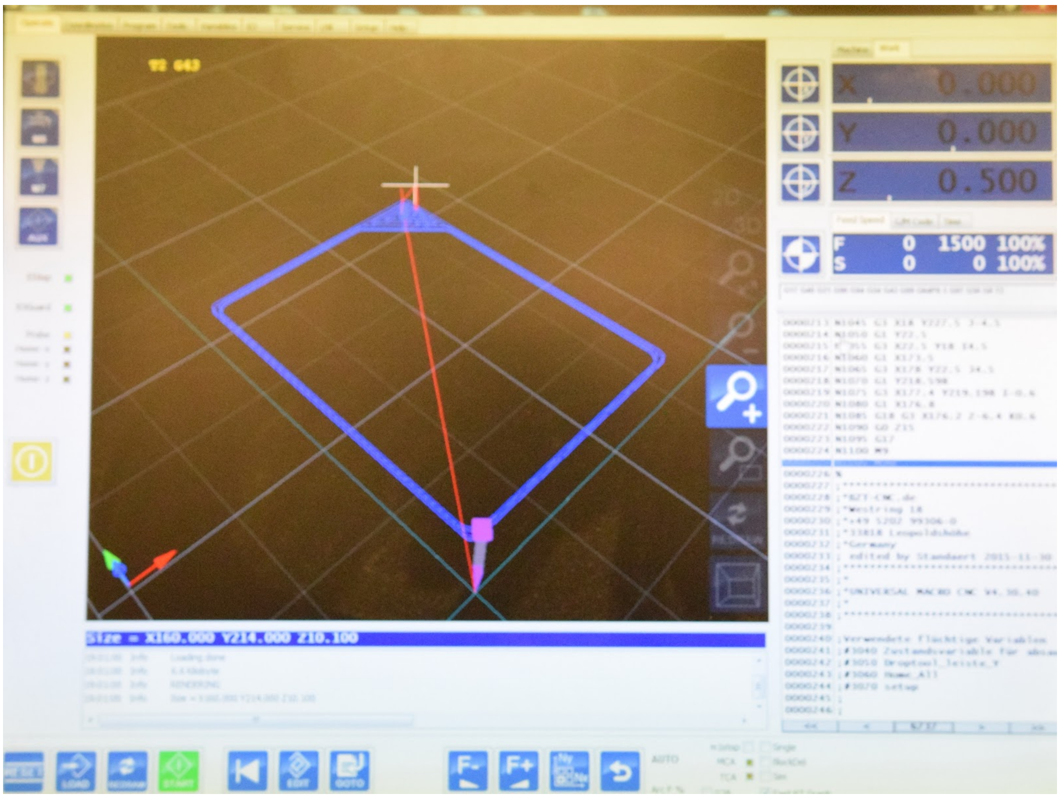
Now you've done all the hard work and it's time for the fun to begin.

Press the AUTO button (F4)in the bottom menu to pass into the Operational Mode.



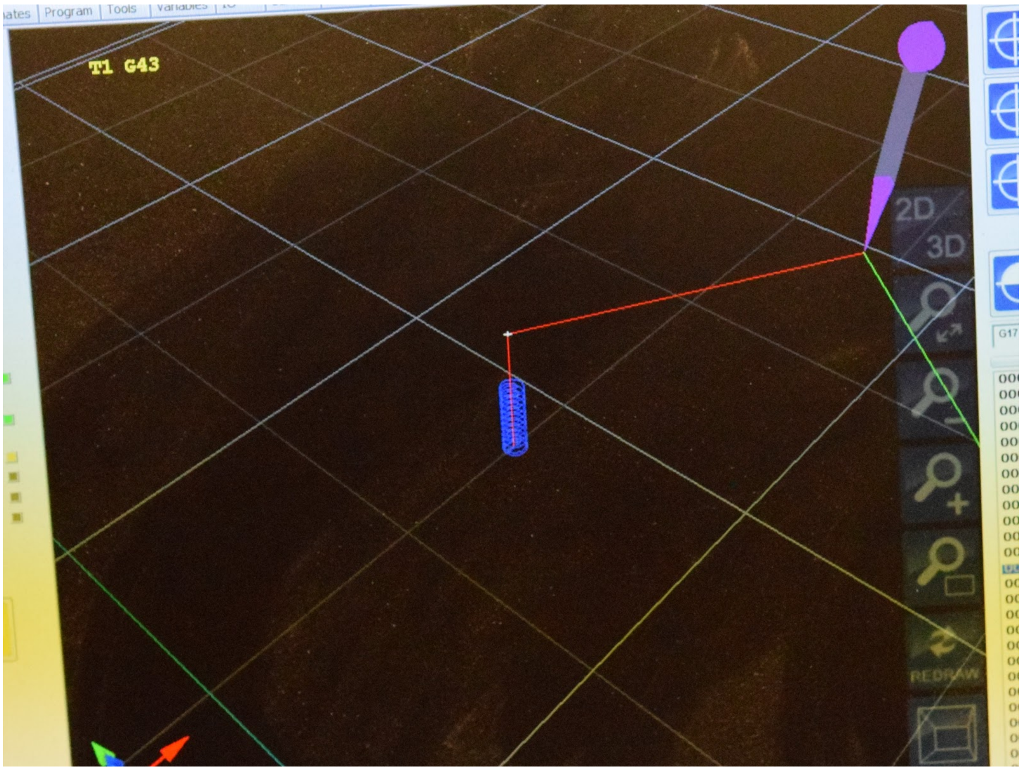
Then the LOAD key (F2) to bring up a file browser. Select your first toolpath. (Most transfer their Toolpaths from Laptop with a USB Key. You may be able to use the Network or a cloud service.)

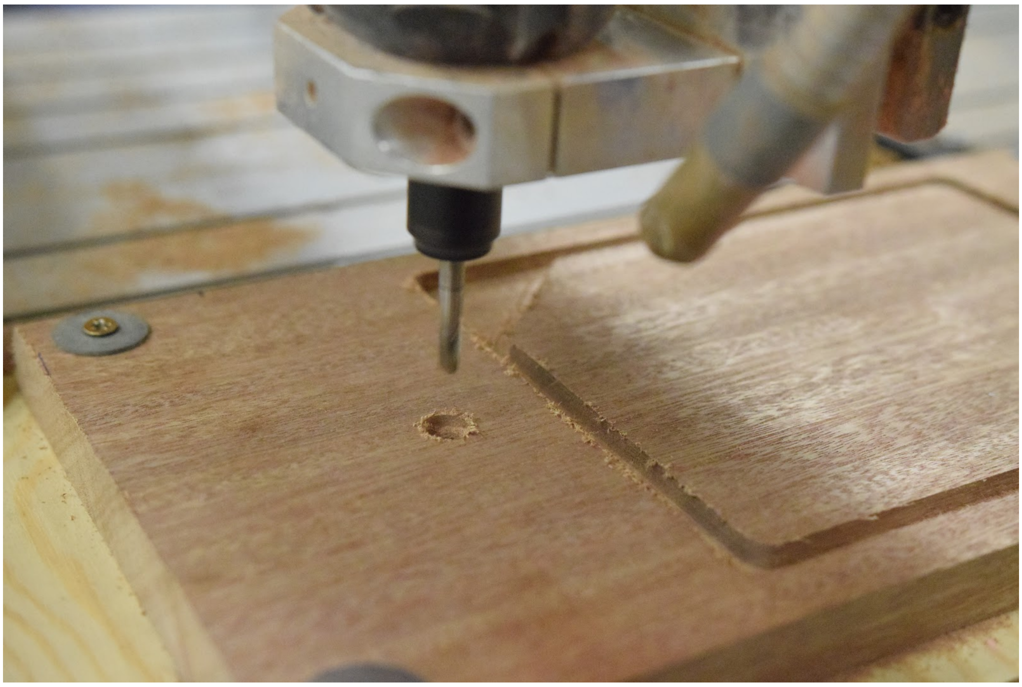
The toolpath should appear in the simulation window as below.



	Now STOP and call a supervisor to verify your setup.





COPY PASTED PICTURES, NOT IN RIGHT PLACE YET

(He / she will check the mechanical setup and that the workpiece origin and toolpath moves correspond to the stock you have clamped on the bed..)

All OK?

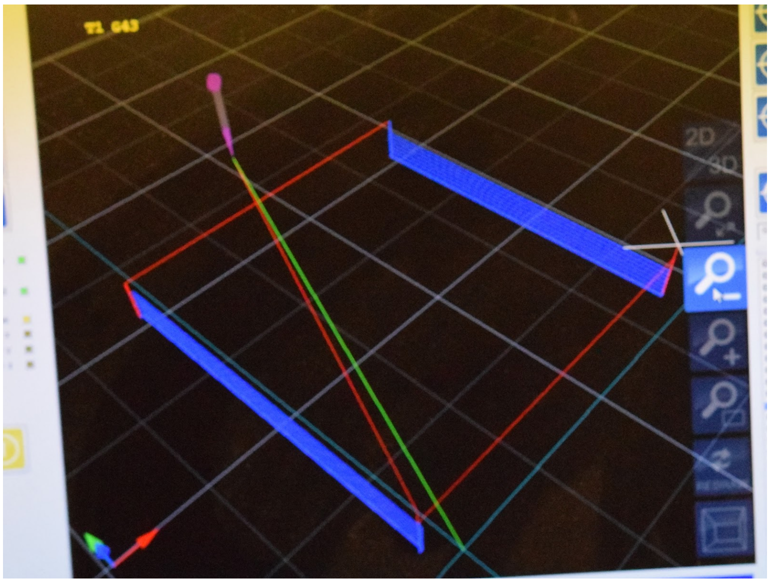
Ensure that the Z axis origin has been set to its original level.

When CNC milling you should wear Protective Glasses and Ear Protection is recommended too.

If using the Large BZT mill, turn on the chip evacuation. If using the small BZT mill ensure that you have the vacuum cleaner plugged in and to hand (to remove chips when they build up).

> If you are using the small BZT mill the spindle is not controlled by the PC. Turn on the spindle manually and increase the speed to the range that most closely corresponds to the toolpath. There is a speed range guide written on the side of the spindle.

Check nobody is close to the machine and then press START.



The CNC will move the spindle to the start position for the toolpath (see the simulation window of the control software), then start the spindle (Large BZT only), then move to the part and start milling.

At the end of the toolpath the spindle will return to the start position for the toolpath and turn off the spindle. (Small BZT: turn off spindle manually).

Note down if anything happened during the toolpath (if this were your toolpath you might wish to modify it.)

If all OK, now load the second toolpath.

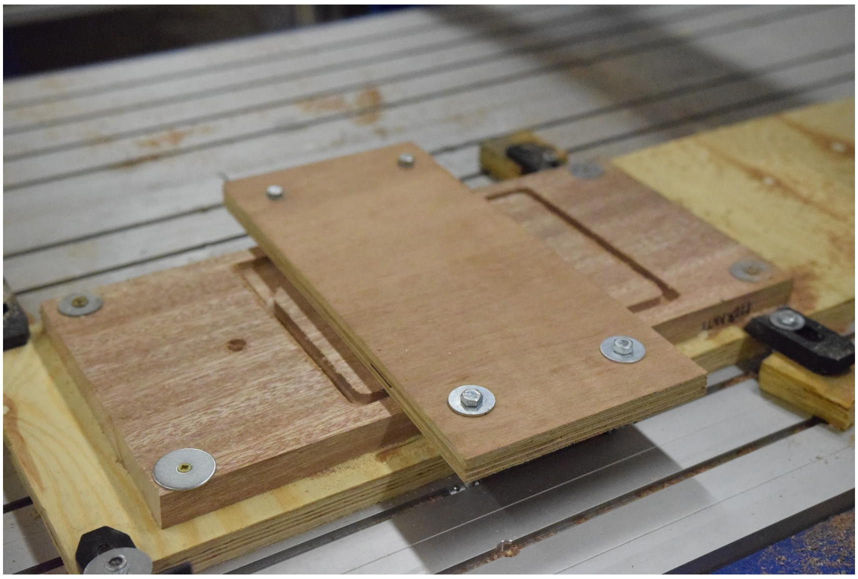
> If using the Small BZT start the spindle (note the second toolpath may require a different spindle speed).

Press START.

Repeat for the

	NOW (on the Large BZT) turn off the chip evacuation).

8 - Change Clamping



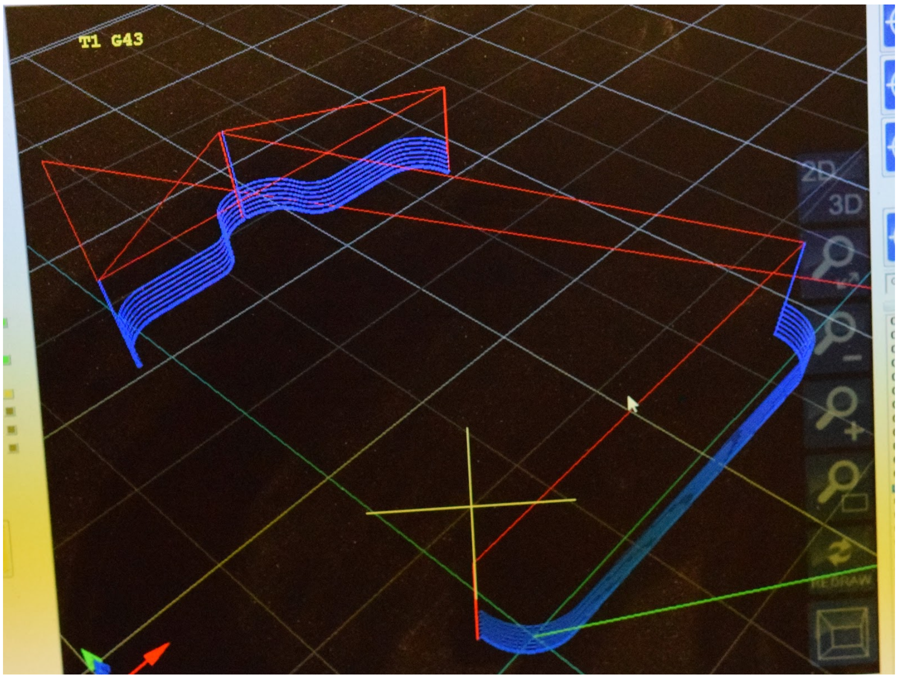
First move the spindle / gantry well clear of the part using keyboard shortcuts.

Cut

9 - Run Last Two Toolpaths

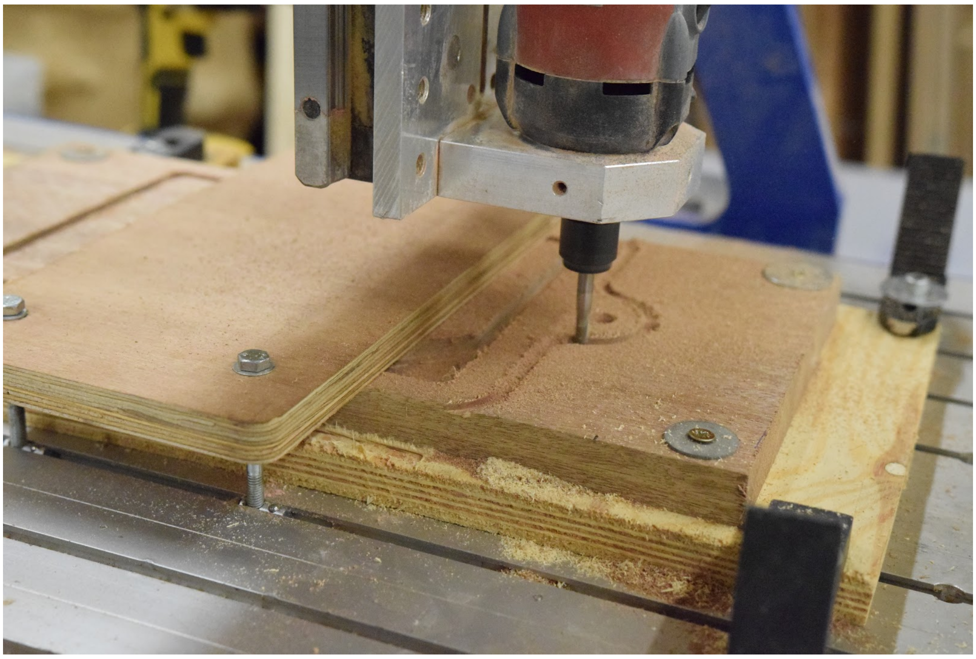
FIrst call the supervisor to check your clamping work.

Load the last toolpaths. It should look like the image below:



(If using small BZT start the spindle at the appropriate speed.

Press START.

	
	To be added: part removal, Cleanup, putting parts back

11 Optional: Coat your board with a protective hard wax.

So there you go…

Our intention was to compose a tutorial that leaves you with a useful piece in the end.

We have a foodsafe hardwax varnish in the FabLab you can use to finish the board.

We hope it lasts you a long time.

ADD CLEANUP AND PUTTING STUFF BACK IN PLACE

10 - What Next?

Get some practice and make something out of wood. Milling is complex and, before trying to take on board any more theory, best to practice what you've learnt

Youtube channels - perfect for absorbing a little more at the end of the day

NYCCNC - self-taught CNC enthusiast goes from bedroom to running fully equipped workshop, making many mistakes along the way.

https://www.youtube.com/user/saunixcomp

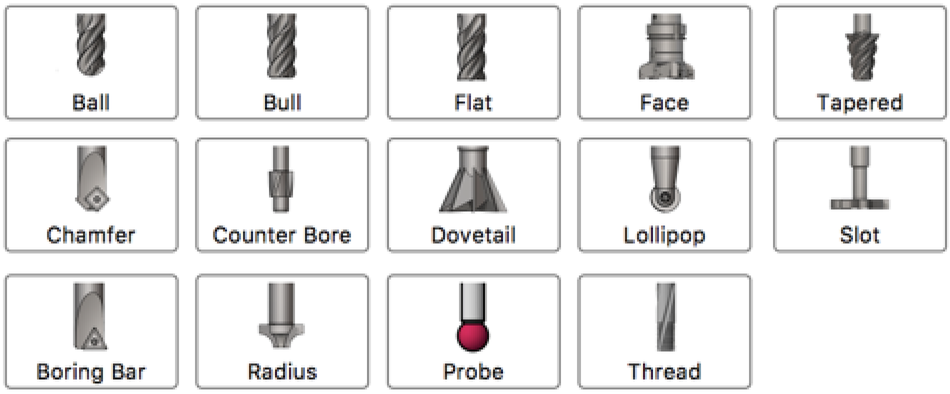
TITANS OF CNC: former convict turns boxer turns machinist and, bring aggression and competitiveness to a world of engineers with 100,000USD machines, discovers an opportunity to make America great again!

https://www.youtube.com/channel/UCc2lUKVOTXKlQR7Fm7h1JfQ

	# 11 - Appendix 1 - Types of CNC mill
A simple manual Mill (note handles)	
Modern CNC controlled mill for metal	

“Vertical Machining Centre” (VMC)	
CNC Router (with vacuum bed)	

12 - Appendix 2 - Types of Endmill



12 - Appendix 3 - Feeds & Speeds

Principles

Some might say that milling is "all about feeds and speeds".

Remember "Speed" = spindle speed and "Feed rate" = rate at which the cutter moves through the workpiece = workpiece moves against the cutter (depending whether the bed or the spindle is moving in the CNC machine in question).

It is a complex subject linking the different components of the machine with the properties of the stock material and the nature of the cutter. Solutions are not always "black and white" but rather, you will find yourself hunting for the "sweet spots" in order to arrive at the desired result in an appropriate cost (level of investment in materials and time). In a sense you are looking for a "recipe" that works.

Start by reflecting upon the following statements:

- if a 6mm endmill cuts perfectly at 300mm/s and 10,000rpm, then a 12mm cutter of the same design would cut at the same feed rate at only 5,000 rpm.
  - The speed of the surface of the cutter is the product of the angular speed (proportional to rpm) and the radius

        (proportional to the diameter). If the angular speed / rpm is the same and the radius doubles, the speed of the cutter surface is TWICE as much.

- if a 10mm 2-flute endmill cuts perfectly at 400mm/s and 8,000 rpm, then an endmill of the same design with 4-flutes (instead of 2), needs only 4,000 rpm to cut at the same feedrate.
    - on the 4-flute endmill, the cutting edges contact the workpiece twice as frequently at the same rpm. In order to remove the same amount of material each time, either double the feed rate or halve the spindle speed.
- When an endmill pressures stock material it bends slightly and so the tip deflects. Whether or not this is an issue depends upon a) the hardness of th ematerial; b) the rigidity of the endmill; c) the stick-out of the endmill; d) the amount of pressure; d) the tolerance desired for the final part;
    - The more flutes, the lower the cutter rigidity. The longer the endmill / the more stickout - the more it will bend. Some endmill materials are especially rigid, for example, Carbide.
    - The pressure the endmill exerts on the stock increases with the feed rate but decreases with the spindle speed.
-

Conventional & Climb milling

These refer to the direction of motion of the cutter relative to the stock. In the case of “climb milling” the cutter turns in a way that complements the direction of feed, as if it were trying to roll along the part. In the case of “conventional milling” the cutter turns such that each cutting edge confronts the moving stock directly.

[diagrams]

Climb milling

- cuts a chip that starts a full width and then tapers towards the end of the cut. This minimises rubbing (increasing cutter life) as well as improving cooling by allowing a maximum of heat to conduct into the chip;
-
- because it is literally “pushing off” against the stock, this can create tool deflection in critical situations;
- pushes down on parts (when face milling) which can be useful if clamping is weak;
- tends to create a better surface finish;

In the early days, CNC machines did not have anti-backlash protection on each axis and so climb milling was generally avoided. Hence the alternative was historically preferred and is called:

Conventional milling

- cuts a chip that starts very thin, the advantage of which is that there are no push-off forces, but which also have for consequence rubbing, heat generation and the creation of a wafer thin chip that removes very little heat (until the end where it is thick but by which time it is rapidly severed from the stock).

How to choose between the two?

- when routing wood, choose the variety that gives the best results (edge finish): this will depend upon the type of wood, the nature of the grain and its direction;
- when cutting full-width cuts (eg slot milling) use conventional milling;
- if you want to mill a hole of very precise dimensions, after removing the majority of material with an adaptive clear, make the first contour pass conventional (low push-off forces ensure you will get the specified dimension without tool deflection or backlash); then climb-mill the second (finishing) contour pass (this will give a better finish and, since the stock to remove is very small, push-off forces will be negligible;
- when milling a part where clamping is an issue (perhaps it is glued?) consider using conventional milling to reduce forces;
- when cutting a hardened material, conventional milling is a better choice;
- if a material risks work-hardening (as a result of the heat generated during milling) climb milling will cause less heat - and dissipate it better;
- in the majority of cases on a modern mill with good backlash protection, use climb milling for better heat removal, better cutter life and a better finish;

Calculating Feeds and Speeds

How much stock should the cutter bite off?

Some endmill manufacturers will provide a table suggesting a recommended chipload for their cutters for different types of material. Chipload has units of mm and indicates the amount of material removed by each cutting edge as the cutter advances.

chipload = (distance moved forward in a unit of time) / (number of cuts in a unit of time)

= (feed rate) / (spindle rpm * number of flutes)

Typically, the harder the stock material, the lower the recommended chip load.
If you are faced with a new material / a new endmill chipload information is a good starting place to calibrate your first cut.

Rearranging the formula above, feed rate can be calculated from chipload as follows:

feed rate = spindle rpm * number of flutes * chip load

(pay attention to the units - if feed rate is mm/min, chip load should be mm and not inches!)

There is, however, another important consideration when planning a toolpath:

How big an area of stock can the cutter advance into before problems occur?

The area that the cutter bites into is proportional to the length as well as the width of cut. Sometimes referred to the depth of cut and the stepover.

The three most obvious problems that may occur?

1) if the cut is too deep (in particular) the tip of the cutter may deflect (because it bends under strain);
2) if the cut is too wide or too deep the spindle motor may not have enough power (torque) to maintain the selected rpm and, as it slows, chipload increases resulting in a very different situation;
3) if the cut is too aggressive, even if the spindle can handle it, depending on the clamping method that has been used, it might displace the stock;

Calculators

An alternative to consulting chipload calculators is to use a feeds & speeds app.. The most complete - and with the best reputation - is called Gwizard and is available with a one month free trial linked to your e-mail address. With this tool not only can you select the material (a particular grade of aluminium, phenolic resin board…) with some precision but also you start by telling it about the characteristics of your spindle so that it can take that into account as well.

https://www.cnccookbook.com/feed-speeds-calculator-gwizard/

Freeware calculators include - ….

Observe the results and tune your Feeds & Speeds

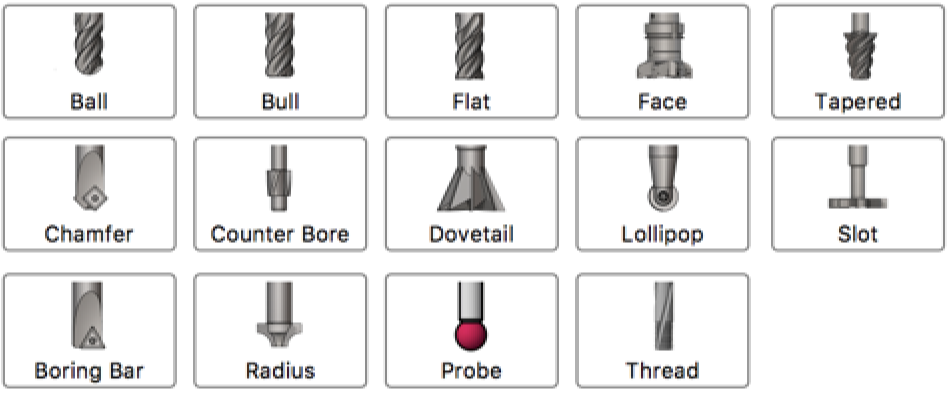
Any feed and speed calculation is just theory. You'll get the best results by paying attention to what happens in reality and refining the parameters accordingly. Here are some basic questions to ask yourself each time that you mill:

What do the chips look like? Steel / Aluminium / Wood

What does the motor spindle sound like? Does it slow appreciably when a cut begins?

Are their vibrations?(“chatter”)

Does the endmill look “burnt”? Are there signs of excess heat?



Drilling Feeds and Speeds

There are many types of drill cycle. A good all-rounder would be “peck drilling” which has the following key properties.

- spindle speed;
- feed speed;
- partial retraction distance;
- full retraction frequency;
- pause time;

The spindle speed should be matched to the material and to the drill bit diameter just as it would for manual drilling. Smaller bits require a higher rpm. Harder materials a lower rpm.

The feed speed - in this case - is the speed at which the drill penetrates the material.

The purpose of “partial retraction” is to reverse the drilling just enough to “break the chip”.

The purpose of “full retraction” is to clear chips from the hole and the flutes of the drill, before continuing with the drilling cycle.

The “pause time” determines how long the drill waits before performing any kind of retraction.
