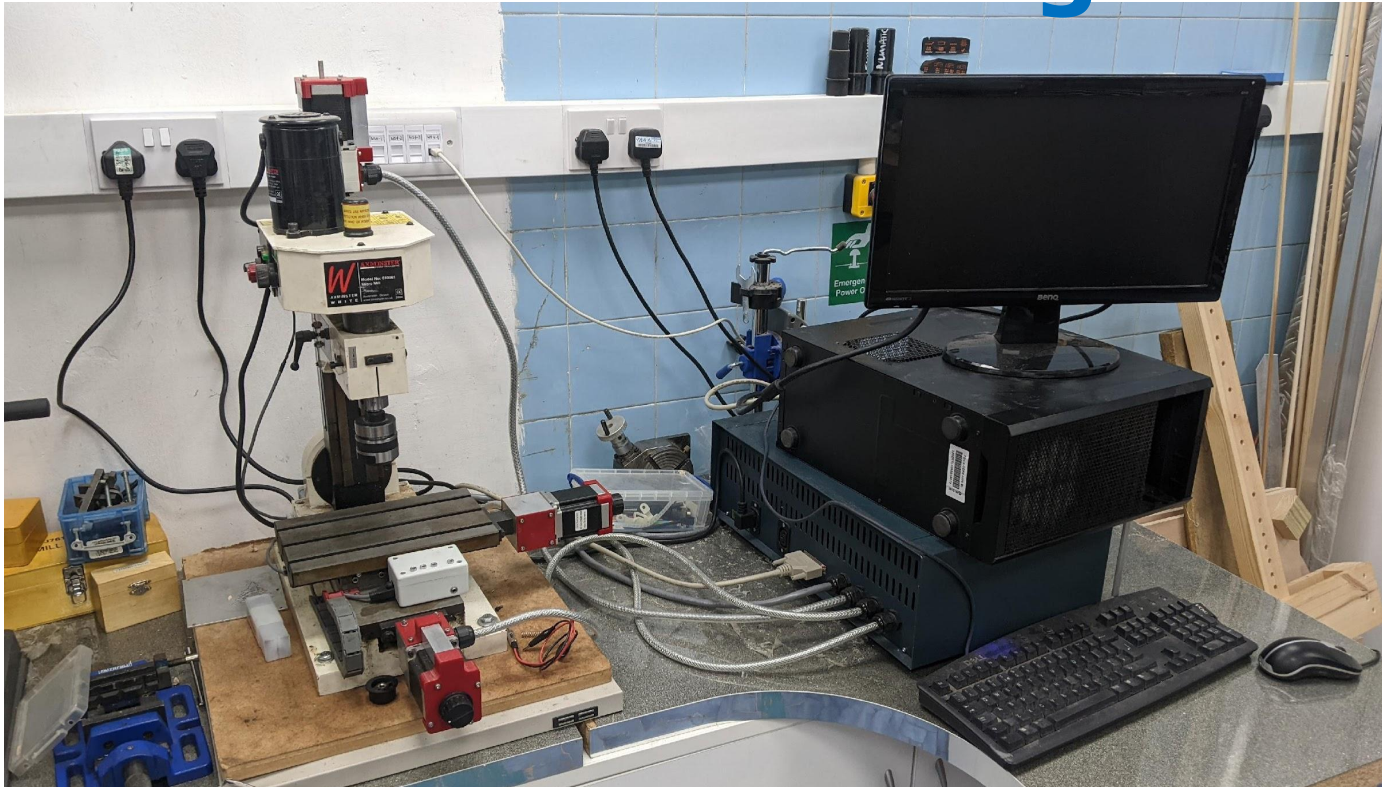
CNC Mill Training



https://wiki.hackhitchin.org.uk/index.php?title=CNC_Micro_Mill

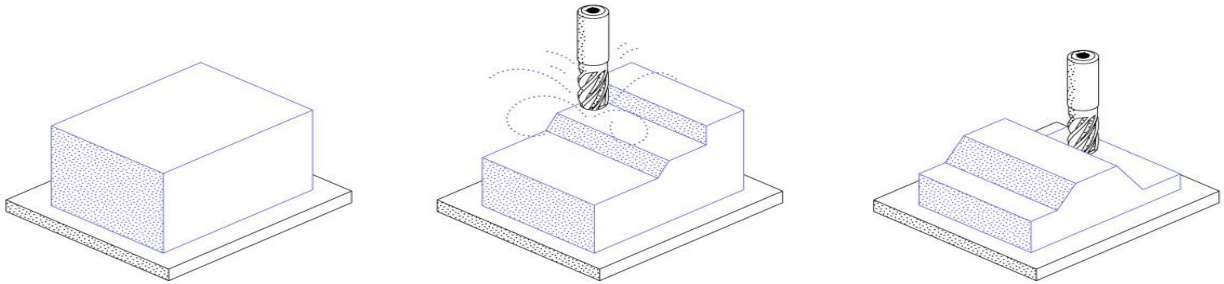
= needs checking or modifying

Todo:
Add resources (links) for fusion 360
Cutter selection
Feeds and speeds

Introduction to the CNC Mill

“CNC” Stands for “Computerised Numerical Control” or “Computer Numerical Control”.
A Mill is a machine tool that can move a specialised cutting tool in 3 axes to remove material.

A CNC mill is related to a manual mill but, unlike a manual mill, the tool movement can be controlled with a computer. It is a machine for cutting materials. For example metals, wood, composites, plastics, and foams. It is one of many kinds of tools that have CNC variants. A CNC mill is very similar in concept to a CNC router. The instructions to control the mill are uploaded into the controlling PC or machine controller from a separate CAD program and then the controller moves the milling tool to remove material from the workpiece.



Introduction 2

CNC milling defines a computer controlled process in which you model your object in specialised software and send it to a milling machine that cuts material by following automated toolpaths.

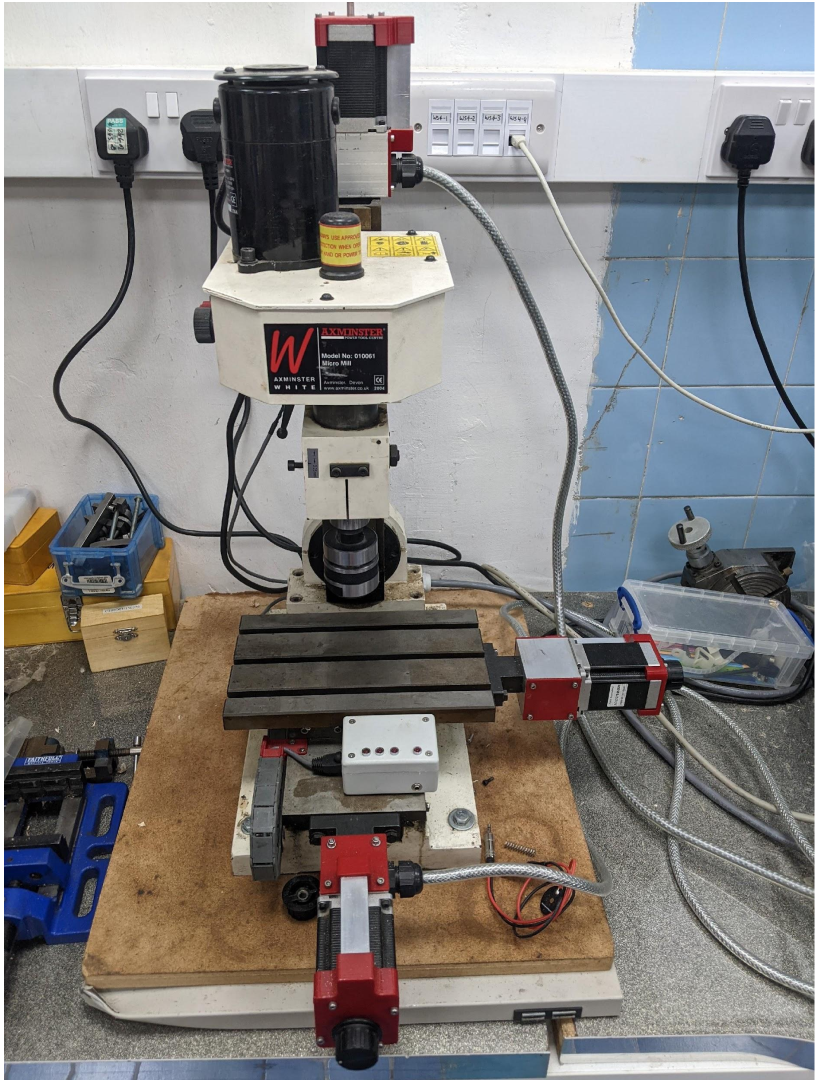
Basic production stages are:

1. Design a 3D CAD model of the part you want to make. Using CAD design software program. e.g. Fusion 360, OnShape, TinkerCAD, FreeCAD
2. Convert the CAD model into a compatible CNC program. Using a tool path generation, or post-processing, tool. e.g. Fusion 360, Vectric V-Carve Pro
3. Prepare the CNC milling machine and workpiece to start machining the part. Mount the work piece using appropriate work holding techniques, install the cutting tool in the correct chuck, define X, Y, Z system coordinates.
4. Executing the CNC Program to start the milling operation. The Computer interprets the toolpath instructions and send steps to the Mill to move the workpiece and cutter to remove material.
5. When the program has finished and movement has stopped remove the workpiece and clean machine thoroughly.

Specifications of the CNC mill

- Bed travel: 200 x 180mm (X and Y)
- Z (vertical) travel: 170mm
- Travel speed: 1500mm/min
- Spindle Speed: 100 to 2000rpm
- Mechanical Precision: 0.01mm
- Spindle Power: 150W Variable Speed Motor
- Max Tool Diameter: 20mm

NOTE: The full Z travel isn’t usually available for the material/workpiece height due to the tool length offset and any jigs, vices or other work holding, or support structure, required to hold the workpiece.



What can you cut ?

It is safe to cut metal, wood, plastics and foam with this machine.

The way the CNC mill is set up means wood and foam need careful cleaning during and after cutting to avoid clogging and corroding the machine. There is also no coolant feed which may limit which metals can be machined.

For any other materials please contact the owners before use.

Scope and Objectives of the training

Scope:

This training covers basic safe use of the CNC mill. Because of the nature of milling machines, the power of the motor, and the health and safety risks of operating a milling machine the hackspace has identified this as equipment which requires members to undertake an induction course. Members need to be able to demonstrate that they understand the risks and practice safe operation of the mill before they are allowed to operate the machine.

Objectives:

1. Review the tool path creation software.
2. How to **use the CNC mill safely.**
3. How to **set up the CNC mill .**
    1. Selecting Milling Bits and fitting them to the Spindle.
    2. Clamping the workpiece and setting X, Y and Z coordinates.
4. How to **perform simple a Cut program.**
5. How to **clean the CNC mill** after use.

Training: What is expected of you ?

1. Have a project in mind BEFORE you start the training.
2. READ this document, including ALL of the linked videos and documents.
3. BOOK a live Induction session at which you will:
    1. Demonstrate basic knowledge learnt above – IF THE INSTRUCTOR FEELS YOU HAVEN'T , YOU WILL BE ASKED TO RETURN TO STAGE (1) ABOVE
    2. Complete a practical induction to the CNC Mill, including safety, machine set-up and simple 2D milling.
4. PRACTICE on the machine as soon as possible after you have finished the training – ideally within a week - expect to commit ~ 8 hours to this if you are new to CNC machining.
5. UNDERSTAND that this is only the first step on the long road of learning about CNC tools and Milling. We cannot teach you everything there is to know about CNC or Mill use in only a few hours.

Safety

CNC mill

CNC mill Hazards

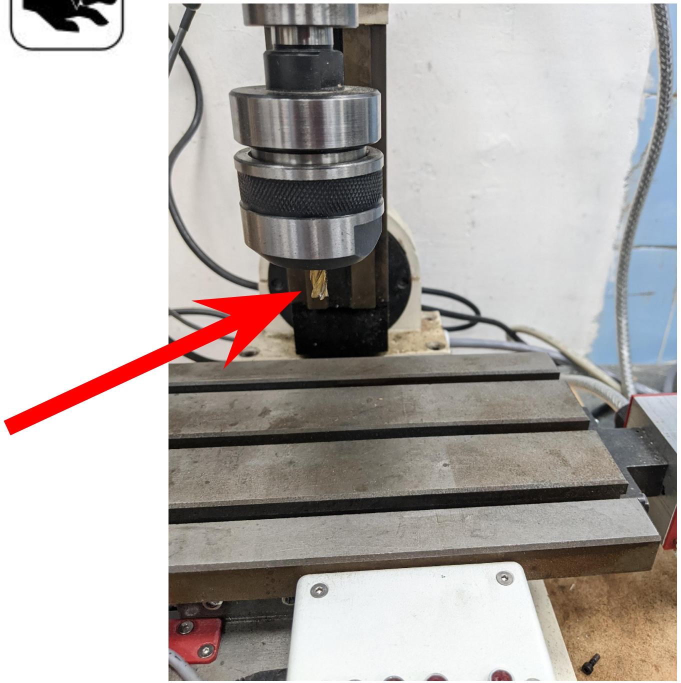
As a trained user, you should be very aware of the following risks and how to deal with them:

- **Fire** - Cutting with the wrong settings could result in the material igniting. Hot swarf could ignite materials in close proximity to the mill.
- **Moving Parts** - Stand clear of the machine while in motion.
- **Sharp Edges** - Take care installing and removing cutting tools. Also freshly machined parts and swarf can have very sharp edges.
- **Dust** – If dust is being created during cutting, run the workshop air cleaner throughout the job and regularly pause the mill and use a vacuum cleaner to remove dust.
- **Noise** - Ensure that you wear the correct PPE when working with the machine.
- **Projectiles** - Poorly clamped work or snapped cutters can be ejected from the mill at very high speed.

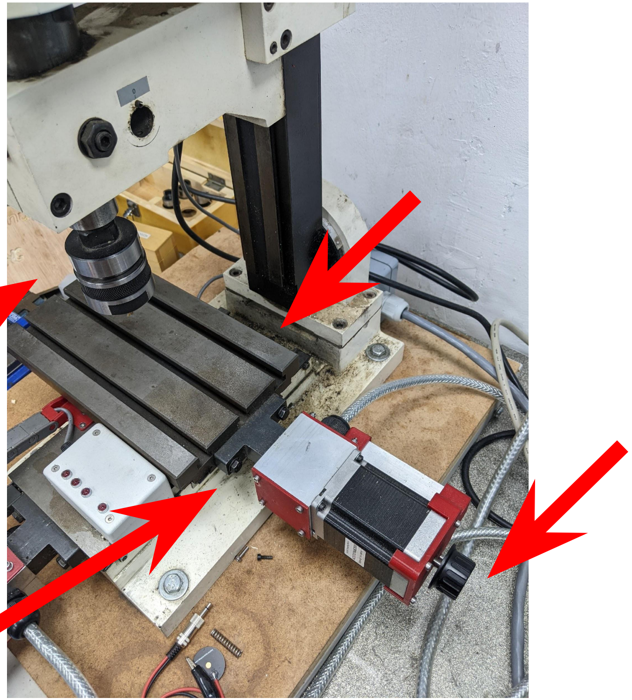
The risk assessment can be found here:

Hazard Points

Caution: Sharp tool
contact may cause injury



Caution: Hands or fingers may
become pinched, resulting in injury.



CNC mill Safe Preparation

- Personal Protective Equipment
  - Eye protection and hearing protection are mandatory. A dust mask is recommended for irritating dust e.g. MDF
- Clothing, Hair and Jewellery
  - Remove loose clothing and jewellery. e.g. rings, watches and necklaces. Tie back long hair or wear a hair net.
- Be careful fitting and removing milling bits, handling machined parts and swarf.
  - The edges of milling bits are very sharp (cut hazard)
  - After milling operations, the milling bit may be very hot (burn hazard)
  - Freshly machined parts may have sharp edges and swarf can be very sharp.
- Ensure the spindle is running before commencing cutting
- Ensure the spindle is stopped before attempting to change a tool or adjust a workpiece
- Ensure the dust extraction is on before cutting?

CNC mill Safe Operations

- NEVER leave the CNC mill running unattended
  - You must ALWAYS be within reach of the emergency cut-off button while it is running
  - If you need a bathroom break, you can pause the operation. Pause the program, retract the cutter and power down the spindle motor. Place a notice on the equipment or have an assistant stay with the machine.
- STAY CLEAR of the machine when it is running automatically
  - Manually-controlled spindle head movements are required for alignment
  - For manual spindle head movement, ENSURE no part of your body is inside the milling machine envelope – PINCH & CUT HAZARD
- In an emergency, hit the big red Emergency Stop Switch
  - This immediately shuts down the machine.
  - No estop on the machine! Workshop EPO? Need to add an E-STOP

Safe Materials and Tools

With the correct tooling, feeds and speeds, you can mill all types of metal, wood, plastics and foams.

- **DO NOT** cut glass or ceramics (MBR: Macor machinable ceramic?)

Take care to match cutters with correct collets

- The CNC mill has an ER16 collet holder and a Posilok chuck
  - We currently only have a single ¼” ER16 collet
  - We have ¼”, 3/8”, ½”, 5/8”, 6, 10, 12 and 16mm collets for the Posilok chuck and 2mm, 3mm, 4mm, 5mm, 6mm Posilok-compatible cutters
- Ensure the shank of the milling bit matches the collet – measure to confirm
  - Failure to do this will break the collet
  - Cutters with a thread on the end are for the Posilok collets
- Only get out one collet at a time – it is easy to mix them up although they are individually marked with their sizes.

Take care to match Feeds and Speeds to Material

- Selecting the wrong feed rate and spindle speeds for the milling cutter and material being cut can cause poor cutting performance, excessive tool wear or in the worst case catastrophic tool failure.

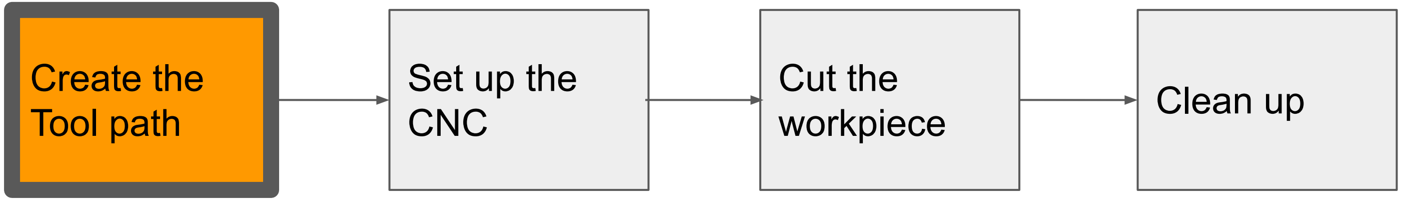
Further essential material

You will need to review this material to be able to safely use the CNC mill:

- These websites discuss basic CNC Mill safety:
  - https://www.multicam.com/6-unbreakable-safety-rules-for-cnc-machinery-safety/
  - http://www.technologystudent.com/cam/cncman3.htm
- The CNC Mill Risk Assessment provides more details

CNC mill Operation

Creating the Tool Path (CAM)



https://wiki.hackhitchin.org.uk/index.php?title=CNC_Micro_Mill

CNC Specification

- Capacity: 200mm (X) x 180mm (Y) x 170mm (Z)
- Rate (X, Y, Z): 1500mm/min
- Spindle Speed: 100 to 2000rpm
- Mechanical Precision: 0.01mm
- Spindle Power: 150W Variable Speed Motor (manual control, not automatic)
- Max Tool Diameter: 20mm
- Control Data Format: G code
- Control System Software: LinuxCNC
- Control System Hardware: MESA card interface

Recommended feed rates and tool speeds for some materials can be found on Slide XX.

Designing For CNC

This is a bit of a which came first? The Chicken or the Egg? Problem. When you are designing your part you need to consider the machining operations you would need to make to create each aspect of the part you are designing. If you have only designed for 3D printing you won’t have needed to think this way as the 3D printer can produce items that would be impossible to machine. However to be able to think about the order of operations you need to have some idea about how you would machine the part and the way different physical features can be achieved with a milling machine. This will be trial and error to start with. Try to think about how you would clamp your part down and which order you might need to drill holes or machine features so that you don’t have to reposition the part or move clamps. Also make sure that you think about how you would get the cutter in to make a cut. It is easy to design something you can’t actually make with a mill!

Tool path generation

Hopefully you considered the order of operations, how you were going to hold the material and the cutting tools needed to make your geometry, whilst you were designing the part?
Toolpath generation is the process of referencing the final geometry to produce the recipe to drive the machine appropriately for those tools and cut strategies.

We are supporting two software packages:

1) Vectric's range of tools:
    - Cut 2D
    - V-Carve Pro
    - Cut 3D
2) Autodesk Fusion 360.

Other programs are available but their output will require validation and are not recommended for beginners.

Comparison of CAM approaches

Package	Input file format	Suitable for	Tool paths styles available
Vectric Cut 2D	DXF, DWG, SVG	Beginners	Pocket, Profile
Vectric V-Carve Pro	DXF, DWG, SVG, STL, SKP	Beginners	Pocket, Profile, V-Carve, drilling, X/Y Raster, thread milling
Vectric Cut 3D	DXF, STL	Beginners	X/Y Raster, profile
Fusion 360	Almost everything 3D	Intermediates to Advanced	Almost everything 2D & 3D

Vectric has been bought by the Hackspace with a Hackspace Licence and is available at the hackspace.
Fusion 360 is Free for Personal Use but you will need to register to be able to use the product

Preparing the Toolpath

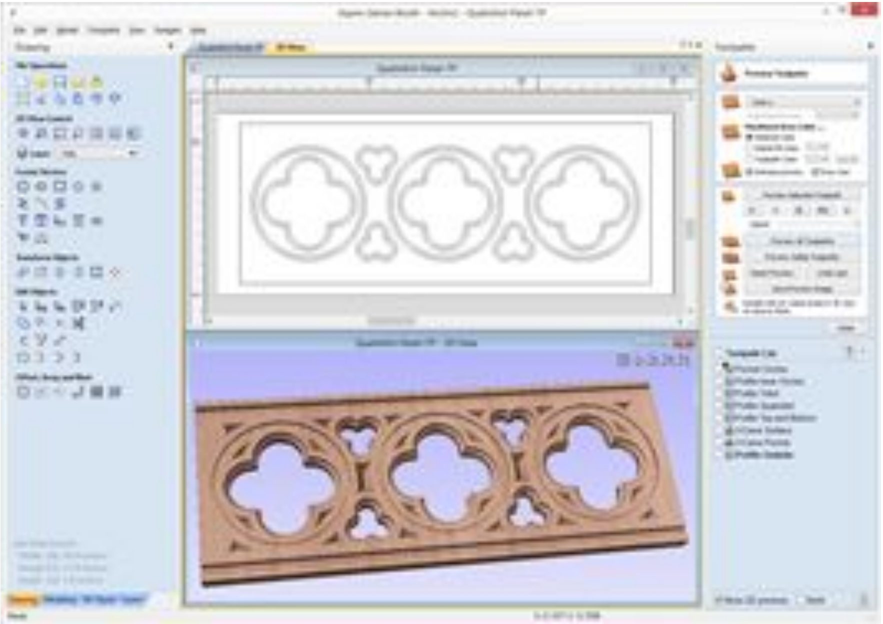
Links to the V-Carve manual and tutorial are below

- Vectric Manual for V-Carve Pro is available at:
  - https://docs.vectric.com/docs/
- Vectric V-Carve Pro online video tutorials are available at:
  - https://www.vectric.com/support/tutorials/vcarve-pro?category=TutorialCategories&playlist
  - In particular, “Getting Started > Avalon Nameplate” video
  - Watching other Vectric videos is not mandatory (there are many of them), but is highly recommended – select the ones that apply to the job you have in mind

VCarvePro

VCarve Pro provides a powerful but intuitive software solution for 2D design and calculation of 2D and 2.5D toolpaths for cutting parts on a CNC mill.

The software can import 2D designs from other programs but also includes a full set of drawing and editing tools with advanced layout options such as True-Shape Nesting. The toolpath options cover all typical 2D routing operations such as Profiling, Pocketing, Auto-Inlays and Drilling as well as 2.5D strategies such as V-Carving, Prism carving, Fluting and even a decorative Texturing strategy. Each toolpath includes appropriate options to customize the settings and provide a high level of control for different types of operation. In addition all toolpaths can be previewed to show just how the part will look when it is actually cut, this allows instant feedback to allow toolpaths to be further optimized.



Details: http://www.vectric.com/products/vcarve-pro.htm

Training Resources:
http://support.vectric.com/training-material/vcarve-pro

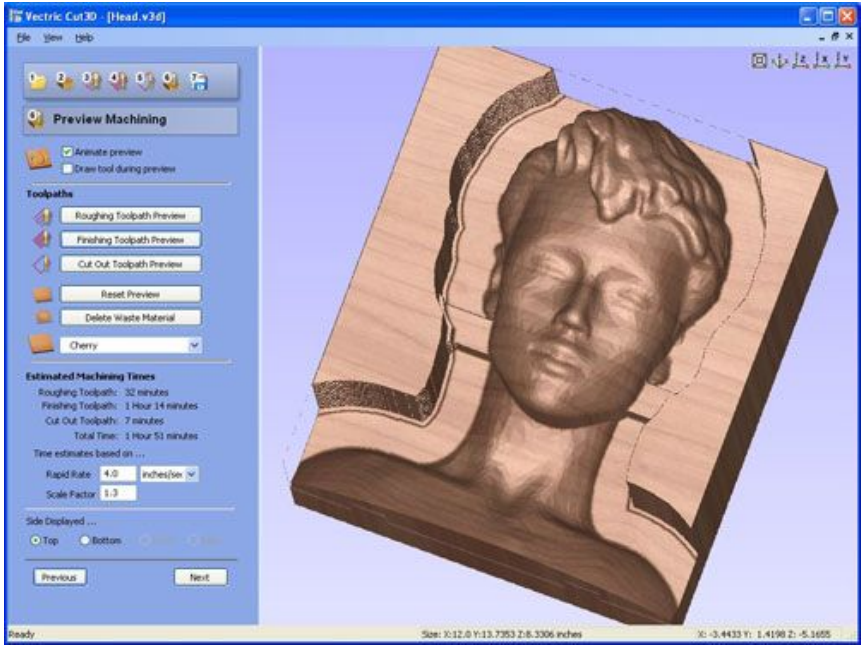
Cut3D

Cut3D is a dedicated toolpath engine for CNC machining 3D models that have been designed using a 3D CAD or Graphics design product such as AutoCAD, Rhino3D, 3D Studio etc. or scanned using a laser or touch probe device. Cut3D's exceptionally easy to use interface leads you step-by-step through the process of loading a model, setting the size, interactively placing tabs to hold the job in place, calculating single, double or four sided roughing and finishing toolpaths, previewing the results and finally saving the CNC code to run on your machine.

3D Models that are too large / thick to machine because of gantry height, cutter length or limited material thickness can be 'Sliced' into separate pieces. Each slice can then be machined and the complete job assembled.

Details: http://www.vectric.com/products/cut3d.html

Training Resources:
http://support.vectric.com/training-material/cut3d



Creating the Tool Path

From the Vectric software, save your tool path file (.tap), then rename it to be a .NGC, and transfer to a USB drive.

If you are using software other than the Vectrics tools or Fusion 360, the tool path must be created with a compatible type of G-code that only uses commands LinuxCNC interprets correctly **(How do we identify this?).**

The next step is to mount your work piece to the mill and mount the tools indicated in your tool path. See slide XX for these instructions.

Feeds and Speeds

Creating the Tool Path

https://wiki.hackhitchin.org.uk/index.php?title=CNC_Micro_Mill

What are feeds and speeds? Why are they important?

[Milling machine tutorial - cutter selection, speeds and feeds]()

[Wikipedia - Speeds and feeds]()

The phrase speeds and feeds (or feeds and speeds) refers to two separate velocities in machine tool practice, Cutting Speed and Feed Rate. They are often considered as a pair because of their combined effect on the cutting process. Each, however, can also be considered and analysed in its own right. They are somewhat analogous to the speed and power settings on the laser cutter; the two can be varied together to get similar results but too much or too little of either can cause problems.

Appropriate feeds and speeds depend on many factors, the more important ones being:

Tool Diameter	Workpiece Material, Grade or Alloy	Depth of Cut	Width of Cut	Tool Stick-out
Surface Finish	Accuracy Required	Tool Material	Workpiece Rigidity	Machine Rigidity

If any of those factors change, the feeds and speeds may need changing to avoid tool breakage or poor cut quality.

Fusion 360 has suggested feeds and speeds for all cutters and materials, but the values are inappropriate for our small mill, generally the suggestions it provides require more spindle speed, more spindle horsepower, more rigidity and higher feed rates than our little machine can manage. Always test new feeds and speeds, starting conservatively.

Cutting speed

Cutting speed (also called *Surface Speed* or simply *Speed*) is the speed difference (relative velocity) between the cutting tool and the surface of the workpiece it is operating on. It is expressed in units of distance along the workpiece surface per unit of time, typically *surface feet per minute* (sfm) or *meters per minute* (m/min).

Feed rate is the relative velocity at which the cutter axis is advancing along the workpiece.

Feed rate

Feed rate units depend on the motion of the tool and workpiece; when the workpiece rotates (e.g., in turning and boring), the units are almost always distance per spindle revolution:

- *inches per revolution* (in/rev or ipr)
- *millimeters per revolution* (mm/rev)

When the workpiece does not rotate (e.g., in milling), the units are typically distance per time:

- *inches per minute* (in/min or ipm)
- *millimeters per minute* (mm/min)

Although distance per revolution or per cutter tooth are also sometimes used.

Sample feed rates

For working out feeds and speeds, we recommend https://zero-divide.net/fswizard.

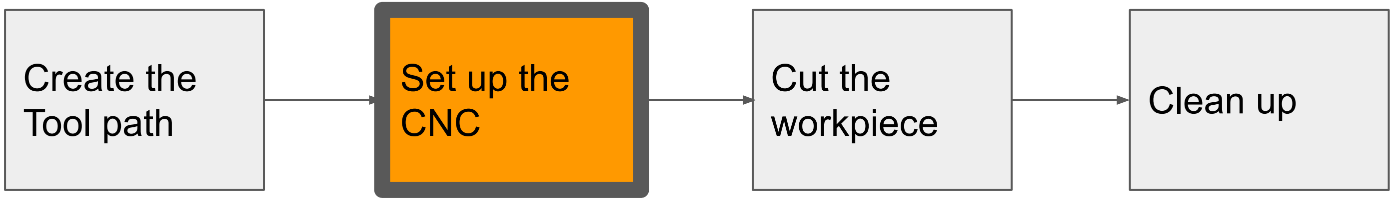
The feeds and speeds below have been tested on our machine. The 3mm cutters were single flute carbide.

Note depth of cut, width of cut, tool stick-out, spindle speed and feed rate all affect the cut quality and chance of braking cutters.

- Foamed PVC (foamex):
  - 6mm diameter, 10mm deep pass depth, spindle 2000rpm, feed 300mm/min.
  - 3mm diameter, 3mm deep pass depth, spindle 2000rpm, feed 300mm/min.
- Polyurethane foam:
  - 6mm diameter, full depth pass (30mm+), spindle 2000rpm,feed 300mm/min.
  - 3mm diameter, 6mm deep pass depth, spindle 2000rpm,feed 300mm/min.
- Polycarbonate & Acetal (Delrin):
  - 6mm Cutter, 2mm deep pass depth, spindle 2000pm, feed 300mm/min.
  - 1/8th or 3mm cutter, 1.5 - 2mm deep pass depth, spindle 2000rpm, feed 300mm/min.

CNC mill Operation

Setting up CNC equipment

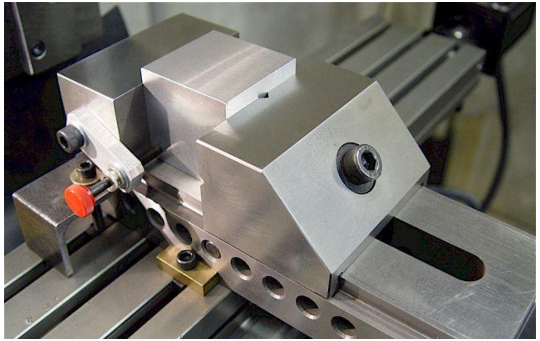


https://wiki.hackhitchin.org.uk/index.php?title=CNC_Micro_Mill

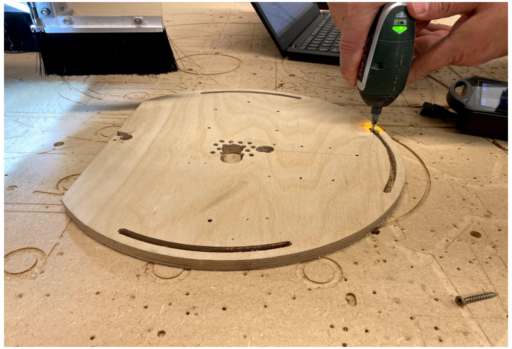
Secure workpiece

There many ways to secure workpieces, the most common are:

- Clamped in a machinist's vice



- Screwed down to a spoil board or sacrificial board



- Toe clamped to the bed

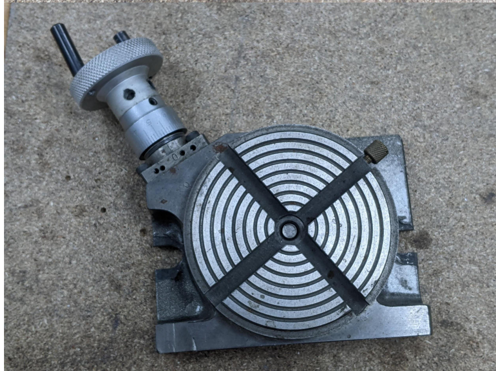
- In a rotary axis/chuck

Secure workpiece

1. Position the work on the bed

2. Measure the thickness and size of the material (again)
   - Confirm it is consistent with your cut depths, practical tool stick out and the size of the work, with a suitable allowance for the clamping method.
   - Cuts should not penetrate into a sacrificial bed by more than **0.5 mm**
   - If necessary, modify the tool path file to be consistent with the actual size of the material

3. Clamp the workpiece using an appropriate method. In general:
   - Billet/block/thick sheet: clamped in a machinist’s vice
   - Very large or unusually shaped: Toe clamped to the bed (with spoil board if cutting through)
   - Thin sheet: glued or screwed down to a spoil board
   - Round, rod-like or working from multiple sides: on a rotary axis or using a V-Block

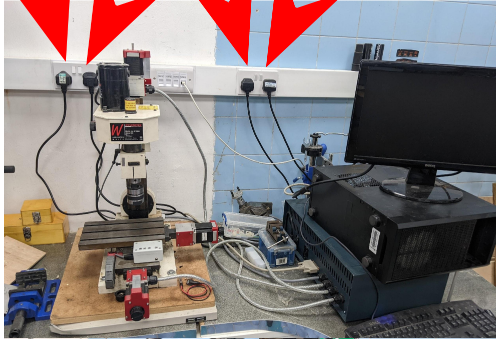


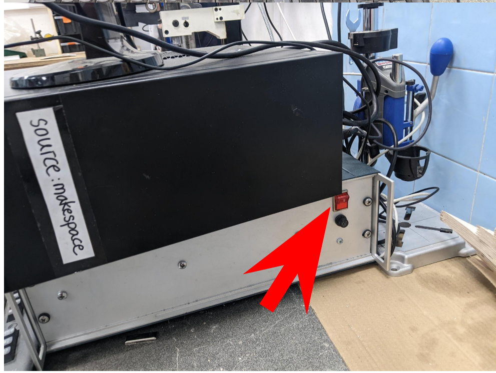
Workholding considerations

- Work Holding is possibly the most important aspect of working on a mill. You must ensure that your workpiece is held very securely so that there is no chance of it moving or being dislodged during any machining operation.
- There are many different ways to clamp your workpiece down:
- List the ways, the many, many ways
- You must also make sure that your clamping or workholding doesn’t stop you from making the machining operations you intend to make. You don’t want the tool or chuck to clash with any of your clamping.

Power

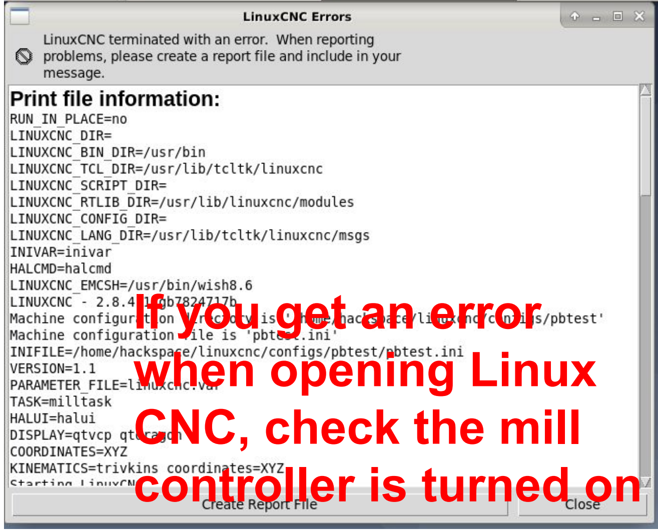
- Machine Setup
  - Turn the Mill, PC , Monitor and machine controllers on at the mains sockets on the wall
    (Note that the Power to the cutting bit spindle is not powered at this point, it will be turned on just prior to cutting.)
  - Press the button on the side of the PC case to boot the PC
  - Turn on the controller with the red rocker switch

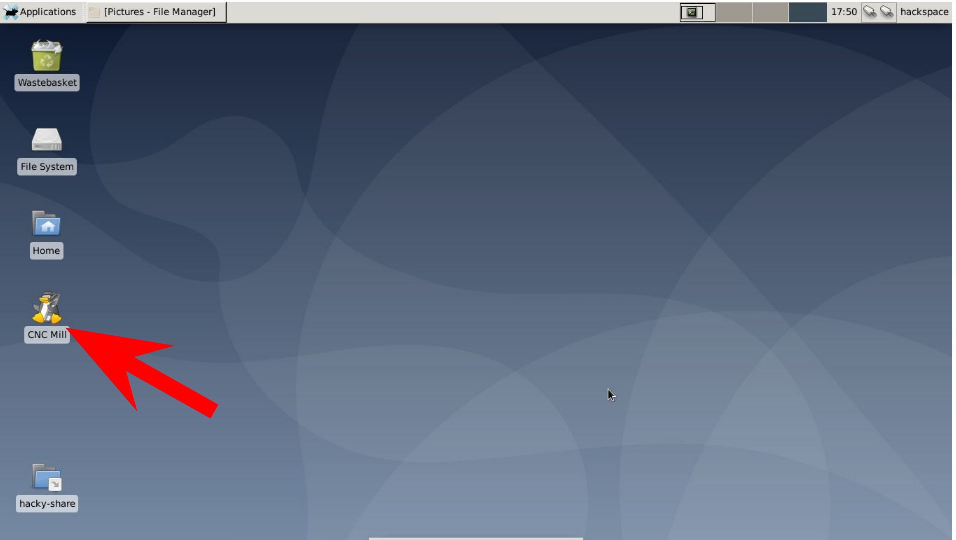


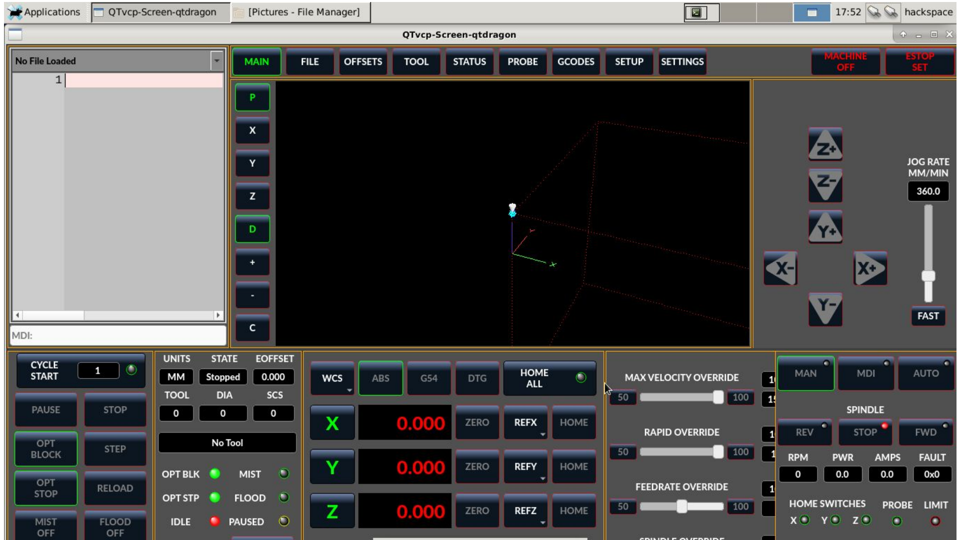


Opening LinuxCNC

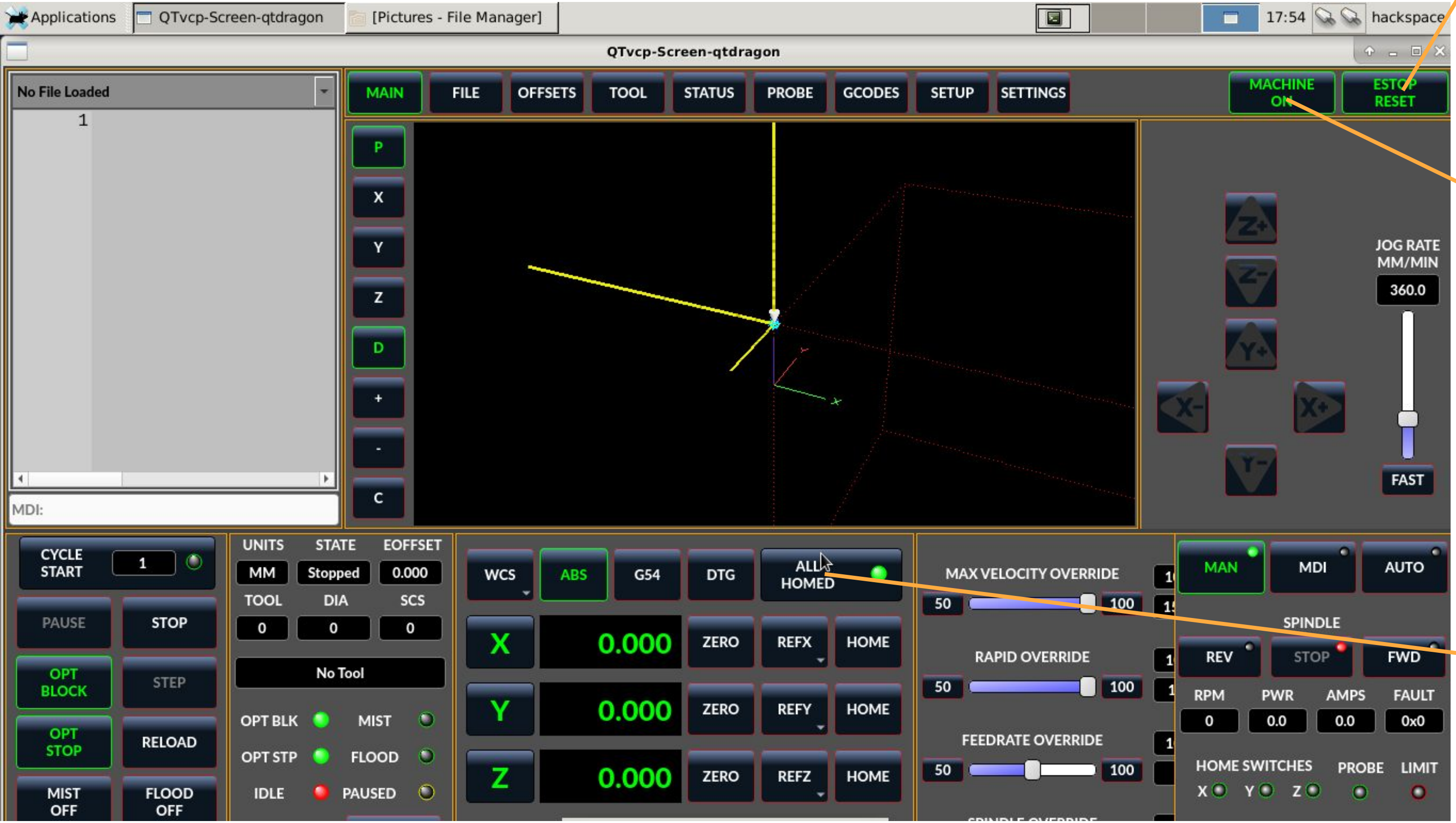
- When the PC has booted, login with the credentials noted besides the machine
- Open LinuxCNC by doubling clicking the CNC Mill icon on the desktop







Connect to and Home the Mill



1) Reset ESTOP

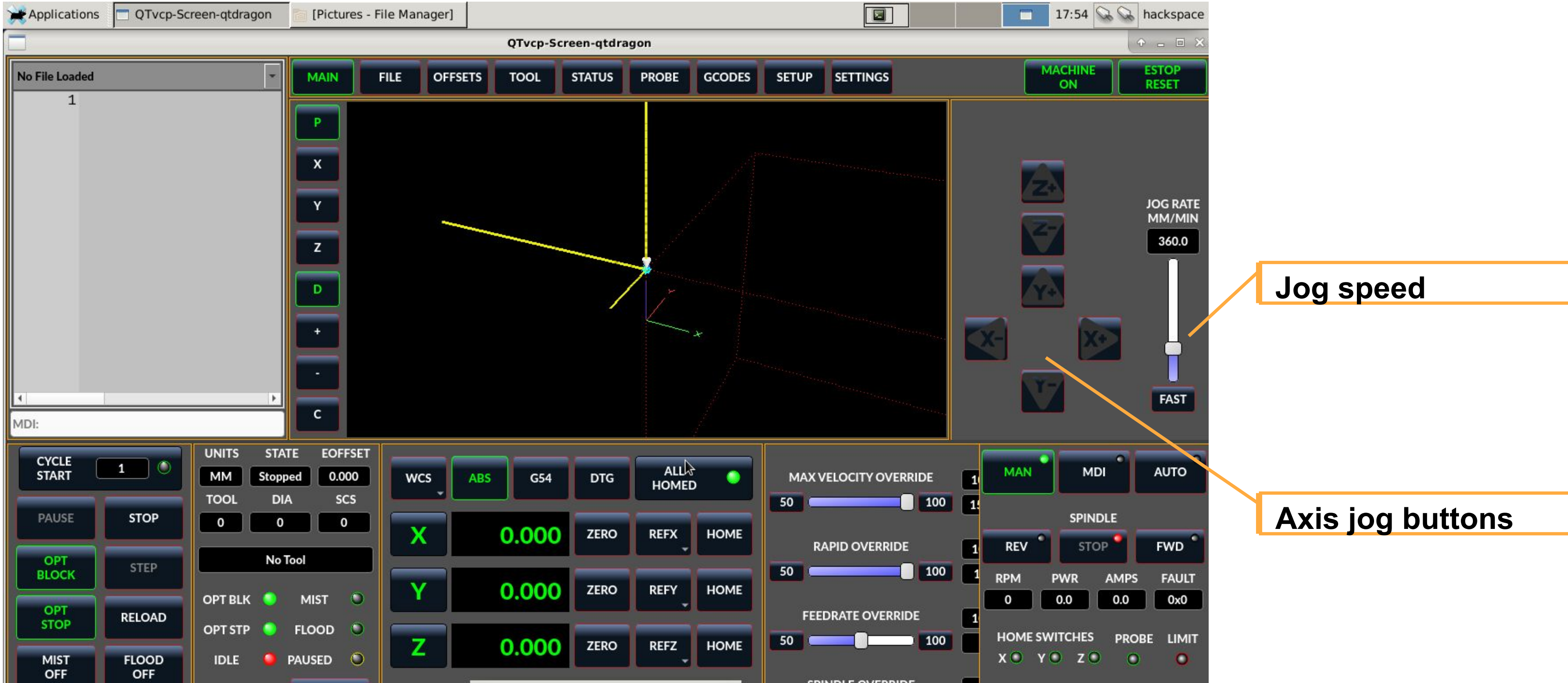
2) Turn Machine ON

3) Check machine has room to move

4) Home All Axis

Or press each home in sequence?

Manually controlling the mill



Mounting the Cutter

- If the spindle is not already near the top of its travel, use the Z+ jog button to move the spindle upwards

- Select the tool (bit) as specified in the tool path file
  - Confirm the tool diameter with callipers
  - Choose an appropriate collet/chuck

Mounting the Cutter

If using an ER Collet, select the correct size collet

- Test the cutter in the selected collet.
- It must be an easy press fit. Do not force fit.
- Remove the cutter from the collet.
- Do NOT attempt to fit collet into the nut with the cutter in the collet!

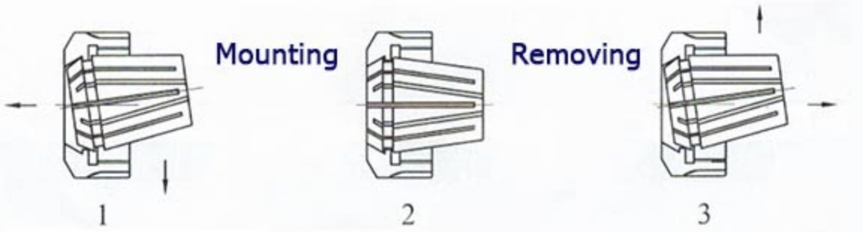
Mounting the Nut, Collet and Cutter

- Remove nut from the spindle (it should be loose and no tools are needed at this stage)
- Clip the collet into the nut from the reverse side of the nut, initially at an angle then rotate to be parallel with the axis
- Insert the tool partially into the collet
- Place the collet, nut and tool into the spindle. Lightly finger tighten the nut.
- Turn spindle to confirm that the tool is centred and not bent.
- Position the cutter in the collet so the “stick out”, the distance from the tip of the cutter to the face of the collet, is more than enough to cut as deep as you need it to (e.g. through your material), but not so long that the cutter will snap, overly deflect or vibrate during the cut. The stick-out should be the same as used in the feeds and speeds calculator
- Use the wrenches to tighten the spindle nut. Tighten using single hand to grip both wrenches

Mounting the Cutting Tool into the mill Spindle

Always clip the collet into the nut before inserting the cutter or attaching the nut to the spindle.

Never attach the collet/nut without first clipping the collet into the nut.



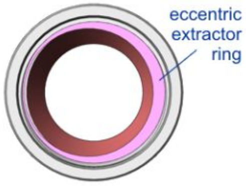
Failing to do this will result in the tool being off-center and may result in the cutter slipping out of the collet while operating.

Collets mounting instructions Keypoint:

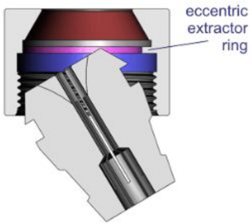
1, Place collet inside nut and rotate the nut to engage with the extractor spiral in collet groove

2, Only then mount nut and collet together into the collet holder or machine spindle

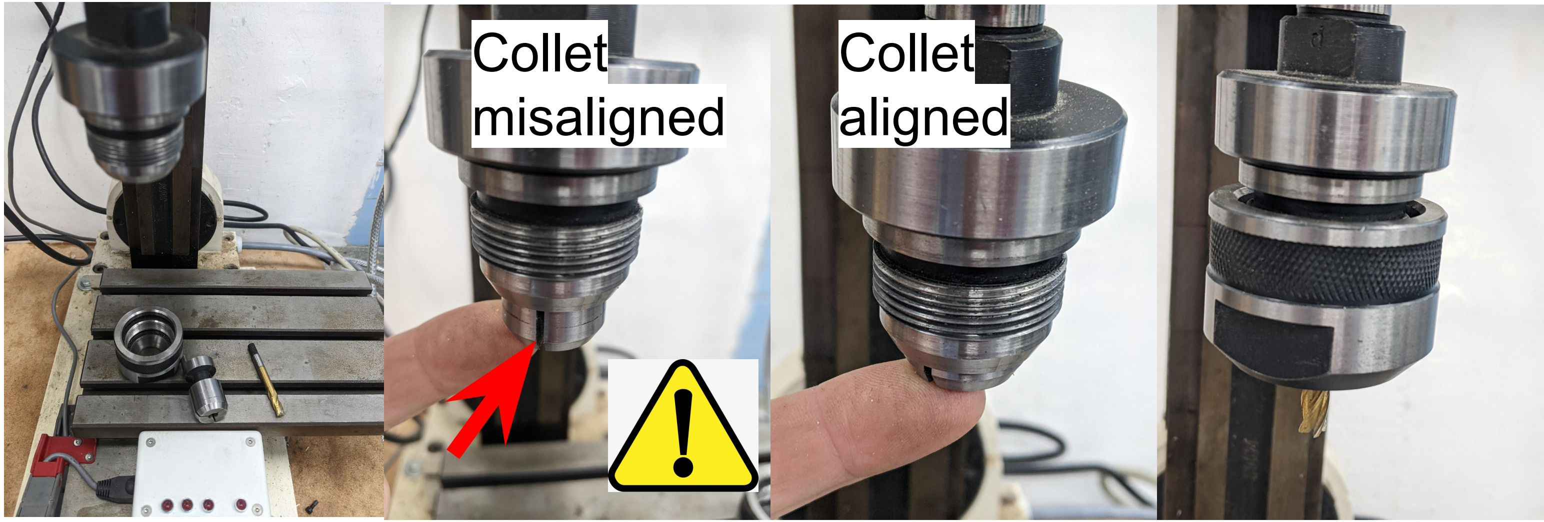
3, Removing the collet from the nut is also quite easy. Loosen the nut, take out the cutter, then remove the collet from the nut.



(fig. 1)



Using the Posilok chuck/collets



1) Select the right collet for the tool shank diameter.
2) Insert the threaded end in to the front of the collet until the thread on the shank engages with the thread in the collet.
3) Screw the tool in to the collet until the thread is flush with the back face of the collet (show with a picture).
4) Unscrew the locking collar from the chuck
5) Insert the assembled tool and collet in to the chuck. Ensure that the step engages with the pin in the base of the chuck. The collet should not be positioned on top of the pin.
6) With the collet in the chuck fit the locking collar screwing it on to the front of the chuck and tighten until the collar stops on the shoulder. Using the correct spanners and chuck tool

Setting the Origin

Setting the Origin

The machine coordinate system datum needs to be established to be equivalent to as used when creating the G-code. This could be in the top-centre of the material, the bottom left corner or somewhere else. It should be somewhere accessible with the cutter, preferably flat or round and easy to measure.

The origin can be established by setting an axis to zero or a particular value whilst the mill is in the appropriate position.

For zeroing, the X/Y/Z axes can all be zero’d simultaneously or individually. You need to be in G54 WCS mode (or other “Work Coordinate Systems” (WCS) such as G55–G59.) The next slide shows the button to press to achieve this.

Set Origin to arbitrary position on work

1. Ensure the bed is clear of all loose tools, clamps and other objects
2. Position tool over the desired origin using the jog buttons
3. Ensure LinuxCNC is in a Work Coordinate System or WCS (as opposed to absolute mode) by clicking G54
4. Set X and Y Origin by clicking appropriate ZERO button. The values next to X and Y on the screen should change to read 0.000

Set Origin to an existing hole, edge or other datum feature

Video: https://www.youtube.com/watch?v=YoN_MSuqzE0

Methods:

1) Gauge Pin
2) Dial Test Indicator
3) Coaxial bore gauge
4) Transfer punch/centre drill
5) Wiggler
6) Edge Finder
    - - kept in the drawer labelled “Draw Bars & Pin Chucks”

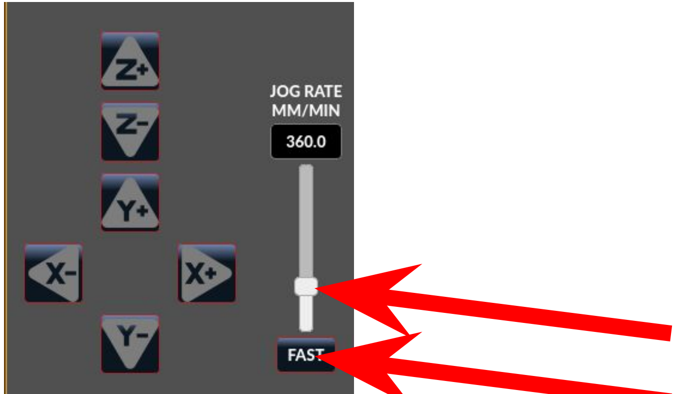
Set Z Origin using “probe”

1. Attach test lead to tool using crocodile clip
2. Ask Paul for details of procedure

Set axis origin to edge or top of work using paper

1. Ensure the bed is clear of all loose tools, clamps and other objects
2. Position the tool slightly outside the edge of the workpiece, or just above the surface
3. Set the jog control to slow mode by toggling the FAST button
4. Set the jog rate to around 10mm/min
5. Jog the machine towards the material, checking the gap with a piece of paper. Decrease the linear increment as you get closer, finishing with the accuracy you require.
6. Set the axis origin by clicking appropriate ZERO button The axis value should change to read 0.000.

Video: https://www.youtube.com/watch?v=V6xGkOWz7RU



Set axis origin to edge or top of work using Edge Finder

1. Ensure the bed is clear of all loose tools, clamps and other objects
2. Fit the Wiggler or Edge Finder in the spindle
3. Position the Edge Finder outside and overlapping the edge of the workpiece
4. Set the jog rate to 100mm/min
5. Turn on the spindle at 100rpm for the Edge Finder or 1000rpm for a Wiggler
6. Jog the machine towards the material, observing the behaviour of the Edge Finder. Decrease the jog rate as you get closer, finishing with the accuracy you require.
7. When the Edge Finder "kicks-out", set the axis origin by clicking the appropriate ZERO button The axis value should change to read 0.000. (if not, you may still be in ABS WCS, if so, click G54)
8. Turn off the spindle
9. For the axis origin to be on the edge of the work, the diameter of the Edge Finder needs compensating for. Either drive the head up, then the bed across the Edge Finder radius and re-zero, or type in the Edge Finder radius into the *refX/refY*-> "Set" popup. The value may need to be negative depending on the direction of the offset.

Videos:

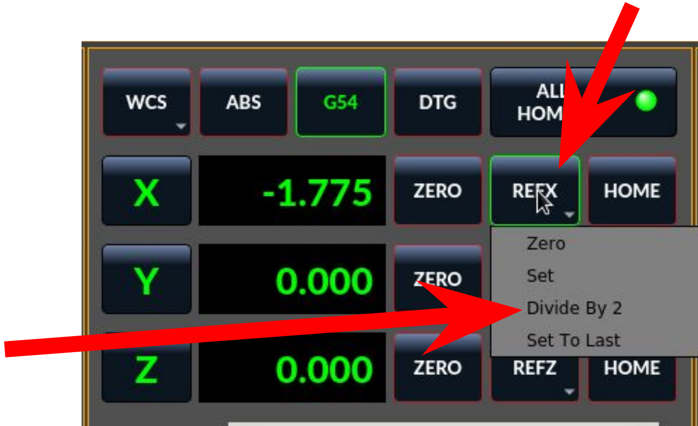
- https://www.youtube.com/watch?v=RhtBdar4iVg
- https://www.youtube.com/watch?v=1TolsGpJZs4

The same method can also be used to find the centre of holes:

- https://www.youtube.com/watch?v=v_tpKP_kJSA

Setting the Origin to the centre of the workpiece

A common machining strategy is for the job to be centred on the material. Rather than eyeballing the centre, a more precise method to establish the centre is to zero the X and/or Y axis on an edge (using an Edge Finder, piece of paper or other method), then move to the opposite edge and align again with the preferred method, but then select "Divide by 2" From the REF menu. This sets the current value to half the length/width, so setting the zero in the centre of the workpiece.



Video: ??????

Loading a G-code file

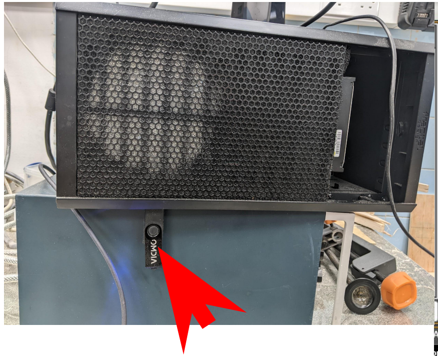
Once the work origin is established, the g-code file for the job can be opened

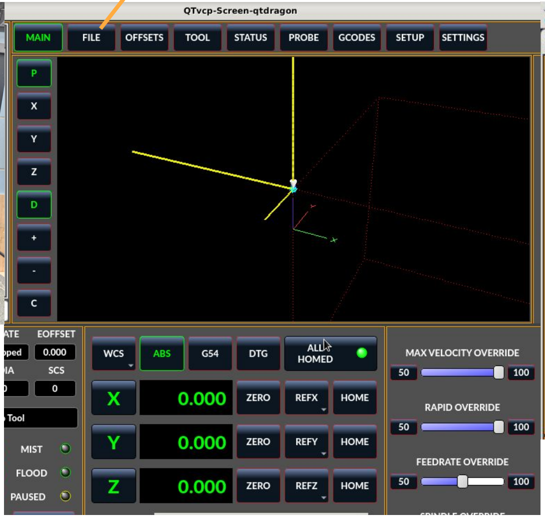
1) Insert USB stick in PC

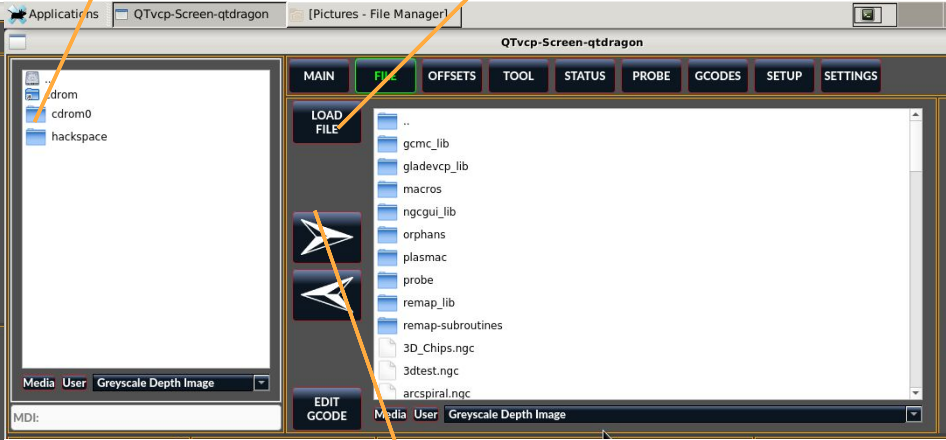
2) Go to File tab

3) Browse to file's folder

5) Load File



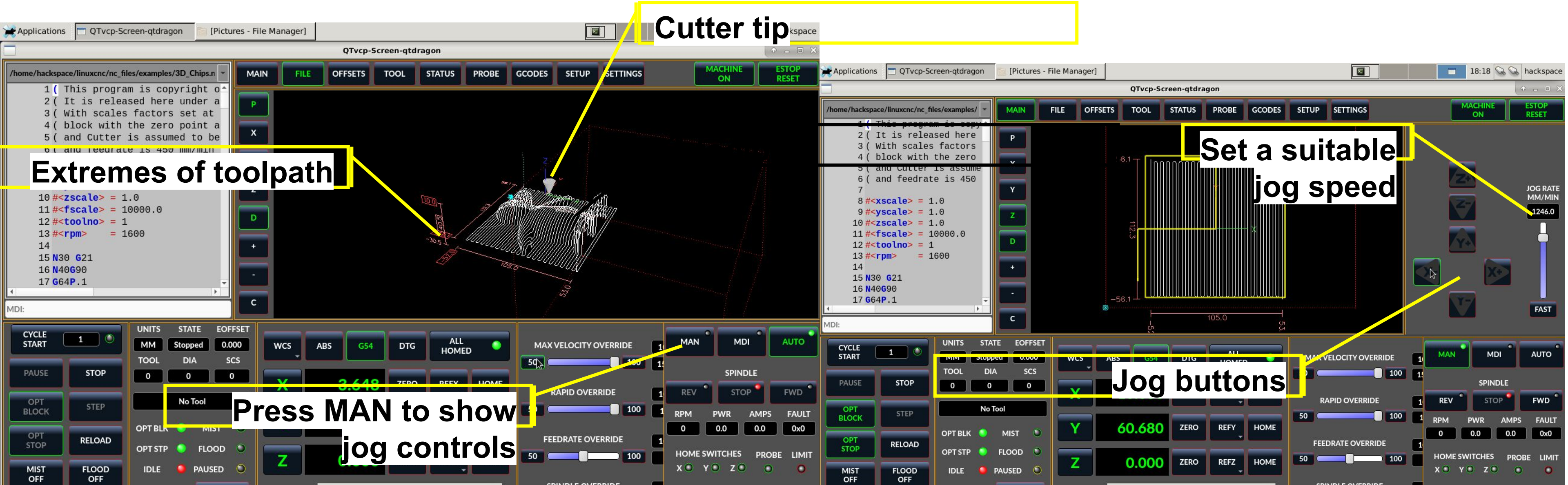




4) Copy the File to the LinuxCNC folder by clicking the arrow

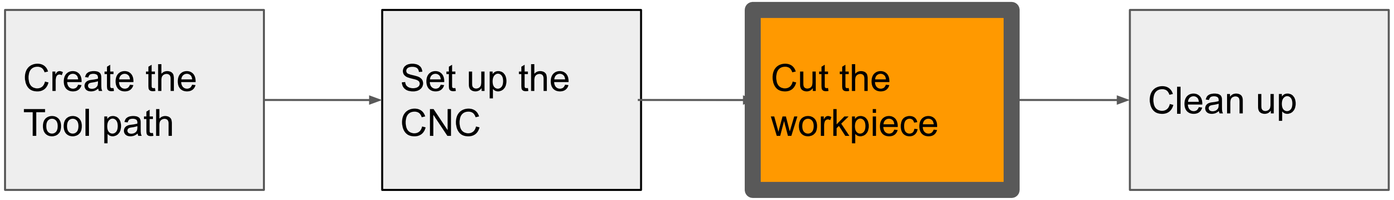
Checking the work envelope

With the G-code loaded, a render of the tool paths will appear in the main window
Use the jog buttons to manually drive the machine around the extremes of the tool path, to check the work fits on the material and the cutter won't hit anything it shouldn't. Be careful not to accidentally drive the cutter into anything!



CNC mill Operation

Executing the Cut



https://wiki.hackhitchin.org.uk/index.php?title=CNC_Micro_Mill

Run the cut

1. Drive the cutter up to be above the work, with no obstructions (clamps) between it and the start of the cut

2. Ensure you are wearing your safety glasses and hearing protection.

3. If using new cut parameters or workholding method, set the feedrate override to 25%

4. Start spindle
   a. Set the gear ratio Low (“L”) for 100-1000rpm, High (“H”) for 100-2000rpm.
   b. Set spindle direction (usually “F”)
   c. Turn red knob to appropriate speed.

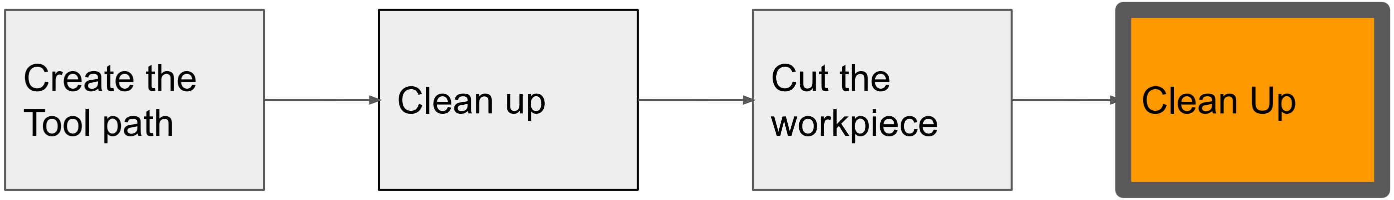
5. Click AUTO button
6. Click “CYCLE START” button
7. Depending on post processor, Z axis may immediately go upwards for a tool change, with a popup to confirm the correct tool is installed

8. Keep cursor near the “PAUSE” OR “STOP” buttons ~~and be prepared to hit the emergency stop.~~

9. Monitor the cut at all times.
   a. Watch for the tools coming close to the clamps
   b. Ensure the cutter and workpiece remains secure. If the cut depth is increasing, your cutter may be working loose.
   c. Press <Pause> If the cut needs to be stopped temporarily in order to remove loose material, swarf or built up chips.

CNC mill Operation

After the Cut



https://wiki.hackhitchin.org.uk/index.php?title=CNC_Micro_Mill

After the Cut

1. Ensure the cut has finished and the bed and head have stopped moving.
2. Raise spindle with Z+ jog button so cutter is well clear of work.
3. Dial the speed down with the red speed control knob, then turn the spindle off with the R/0/F switch.
4. Double check the cut is finished as expected, with profiles complete and through holes clear of bottom surface. Modify the job and rerun as required.
5. Remove workpiece - careful, the workpiece may be hot! It may also have very sharp edges or burrs.
6. Remove cutter - the cutter may also be hot!
7. Return all accessories, tools, cutters, collets and clamps to their storage locations.

Clean Up

1. Clean up all dust, swarf or debris on and around the machine.
2. Use the vacuum cleaner and a toothbrush or sash brush to get into all the crevices, nooks and cranies of the machine.
3. Sweep the floor around the CNC
4. Wipe the steel surfaces of the mill with a lightly oiled cloth to remove any wood dust residue and protect against corrosion.
5. Please leave the machine in as clean, or cleaner, state than you found it.
6. Log out and shut down the PC using the user interface. Please do not just turn the PC off!
7. Turn off mains power