

Student CNC Guide



Table of Contents

| | |
|---|----|
| Table of Contents..... | i |
| Introduction..... | ii |
| CNC Machining Safety | 1 |
| Technical Glossary | 2 |
| What is a CNC Milling Machine? | 3 |
| Intro | 3 |
| Coordinate Systems | 5 |
| Absolute Coordinate System | 6 |
| Work Coordinate System | 7 |
| G - Code Programming Language | 8 |
| Tools..... | 9 |
| Fixtures | 12 |
| Machine Overview | 14 |
| The CNC Mill..... | 14 |
| Tools and Toolholders | 19 |
| Accessories..... | 23 |
| Mach 3 Overview | 25 |
| Intro | 25 |
| Jogging | 30 |
| Setup Work Coordinate System Origin..... | 30 |
| MDI Overview | 31 |
| Wizards Overview | 31 |
| Using the Auto Tool Zero Probe | 31 |
| Using the Touch Probe | 32 |
| CAD/CAM Software Overview | 34 |
| Intro | 34 |
| Autodesk Fusion 360 | 34 |
| Autodesk Fusion 360 Design Workflow | 35 |
| Workflow of Machining a Part | 39 |
| Speeds and Feeds Table..... | 43 |
| Prefix Reference | 45 |
| G - Code Reference | 46 |
| M - Code Reference | 47 |

Introduction

This manual aims to quickly make you successful in operating the TORMACH CNC Mill and making very precise and high quality parts for your project.

The first sections of the manual take you through the very basics of what a CNC Mill is and what operations it can do, the basics of how the machine works, what G - Code is, the different types of tools that can be used and other practical things you need to know.

This is followed by an overview of the actual TORMACH CNC Mill. Clear diagrams with key components is marked out and explained. The available accessories and tools are reviewed.

Next the control software is explained, how it works and the minimum features needed to know.

This is followed by a quick introduction of the CAD/CAM software. Then the workflow of designing a part in Autodesk Fusion 360 and generate the G - Code for the CNC Mill is presented.

Now that the fundamentals of the CNC process has been explained, a step by step section is presented which shows all of all the steps needed to perform in order to go from drawing to finished part.

A number of useful tables can also be found in the manual.

I hope that the manual enables you to succeed in operating the CNC Mill!

Sincerely,
Derek Murphy

CNC Machining Safety

When a CNC machine is operated properly, using the machine is a relative safe process. *This is only under the circumstances that the following rules are followed!*



Safety Rules

- 1. Do not operate the machine if you are not sure of what you are doing, call for help by some who can!**
- 2. Always wear safety glasses when you are near the machine!**
A broken cutter / chip / material can shoot out of the machine during operation.
- 3. Always make sure that the spindle is tightened!**
A loose tool will cause vibrations and can be shoot away during cutting.
- 4. Make sure the stock material is properly tightened!**
The material need to be firmly secured to the table.
- 5. Hit the Emergency Stop / Escape Key if something goes wrong!**
This quickly stops the machine preventing danger.
- 6. Never put your Fingers / Hand / Head near the spinning cutter!**
Don't risk your fingers, stop the machine by Feedhold, adjust then resume.
- 7. Never leave the machine unattended!**
If something goes wrong, you need to be around to stop the machine.

Technical Glossary

| | |
|----------------|---|
| Stock Material | The raw material the part should be machined out of. |
| G - Code | The programming language of the CNC Mill. |
| Feedrate | The speed of the axis. |
| CAD | Software to draw parts in 3D. |
| CAM | Software to generate G - Code from the CAD files. |
| MDI | Manual G -Code input in controlling software. |
| DRO | Digital Read Out. Displays a number, typically an axis coordinates value. |

What is a CNC Milling Machine?

Intro

A CNC Milling machine is a computer controlled machine that can be used to make very precise parts.

The machine works by *removing* material from the *workpiece* with a *rotating cutting tool*.

The machine does this by guiding the tool in all three directions of the *cartesian coordinate system*, that meaning along the X, Y and Z axis.

The arrows in the figure below illustrate the three directions the CNC Mill can move in.

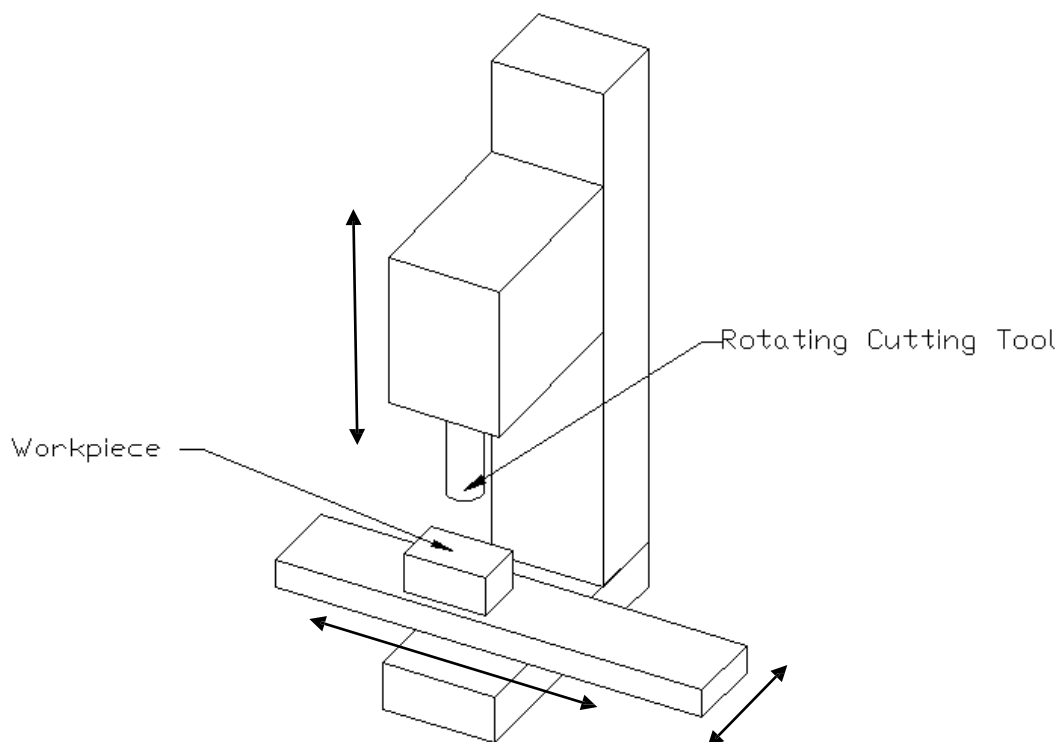


Figure 1: Illustration of a CNC Mill

The CNC Mill can machine many different materials, examples are steel, aluminum, brass, copper and plastic. A Mill is *not designed* for cutting wood and should not be used for that.

Examples of operations that can be made in a CNC Mill is to cut a profile shape, engrave text, mill a 3D surface, drill holes and mill bearing pockets.



Fig 2: Example of CNC Milling

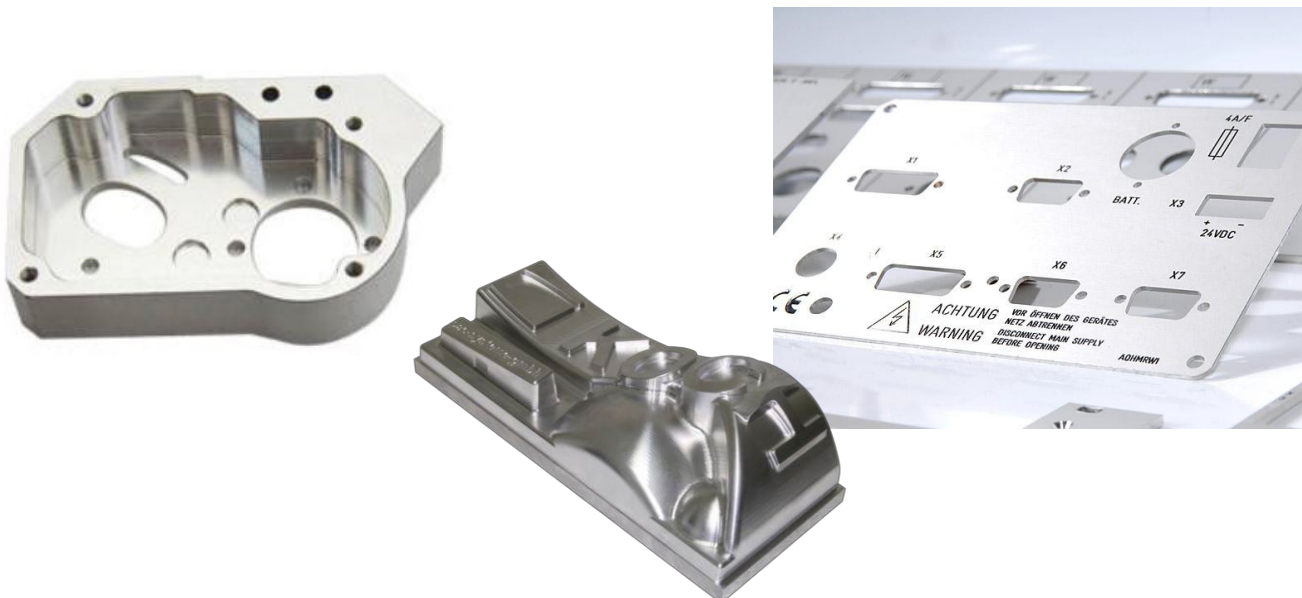


Fig 3: Example of CNC Milled parts

Coordinate Systems

The CNC Mill can as previously mentioned move in the X, Y and Z directions of the cartesian coordinate system. The coordinate system is defined as illustrated below.

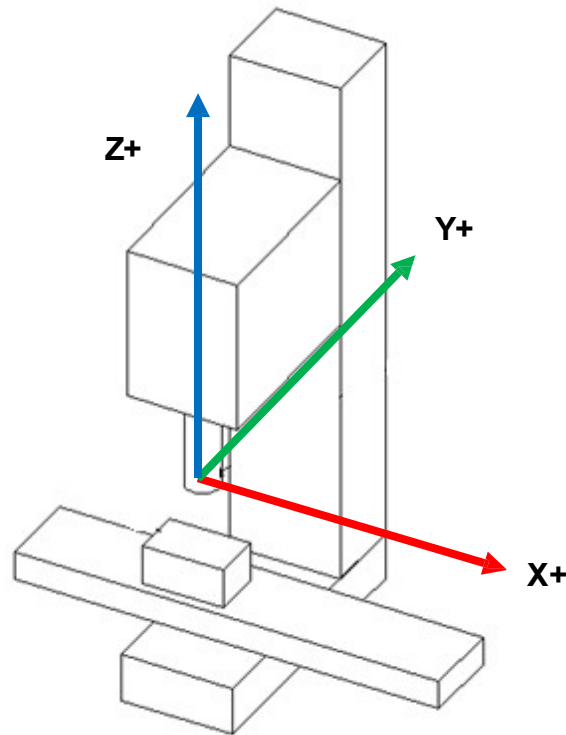


Figure 4: Coordinate system

Note that the coordinate system is defined *relative to the tool motion!*

This means that when the machine for example is cutting in the positive X direction, the tool should move to the right which is the positive direction as defined by the coordinate icon in Fig 4.

Since the tool cannot move in any other direction then up and down due to the design of the mill, the table has to move.

The table and the stock attached to it will thus have to travel in the opposite direction, i.e. to the left when the mill is cutting in the positive X direction in order for the relative motion to be to the right. The same logic applies to the Y axis, but not the Z axis.

Absolute Coordinate System

The absolute coordinate system of the CNC Mill is called the *Machine Coordinate System*, and its *Origin* is located at the endpoint of the three axis's. From the machine origin the X and Y axis can only move in the positive direction and the Z axis can only move in the negative direction.

The origin of the machine coordinate system is called *Home Position*. Every time the machine starts up the operator should perform a homing cycle (Reference all axis) to find the Home Position. This enables the machine to know where it is. The machine finds the Home Position by slowly driving each axis to their extreme limit (-X, -Y, +Z). As each axis reaches its mechanical limit, a sensor triggers and stops the motion. When this has been done to all three axes, the position of the machine is at the origin of the machine coordinate system.

It is very important that the machine is referenced to its absolute position since the control computer and G - Code programs base some movements with respect to this coordinate system. Safe axis limits are also based on this coordinate system.

The operator must perform a Reference the Axis on every startup of the machine!

Figure 5 illustrates the origin / home position of the machine coordinate system.

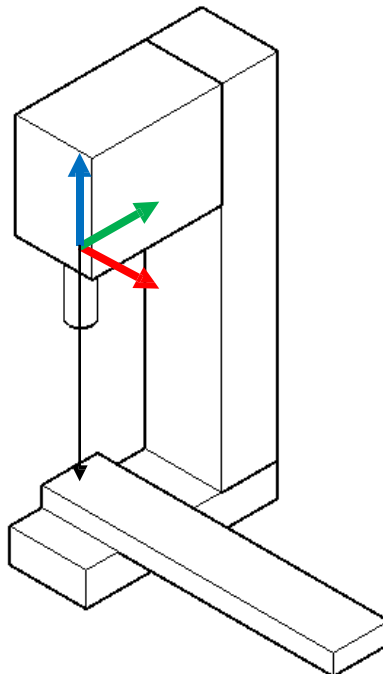


Fig 5: Machine Coordinates origin

Work Coordinate System

When programming the CNC Mill it is useful to define a custom coordinate system that can be set anywhere on the part to be milled.

A Work Coordinate System can be setup by moving the machine so that the cutter is in the appropriate place and call this the origin of the Work Coordinate System

The origin of the Work Coordinate System will typically be set at the center of the part and with the tip of the cutter directly above the part.

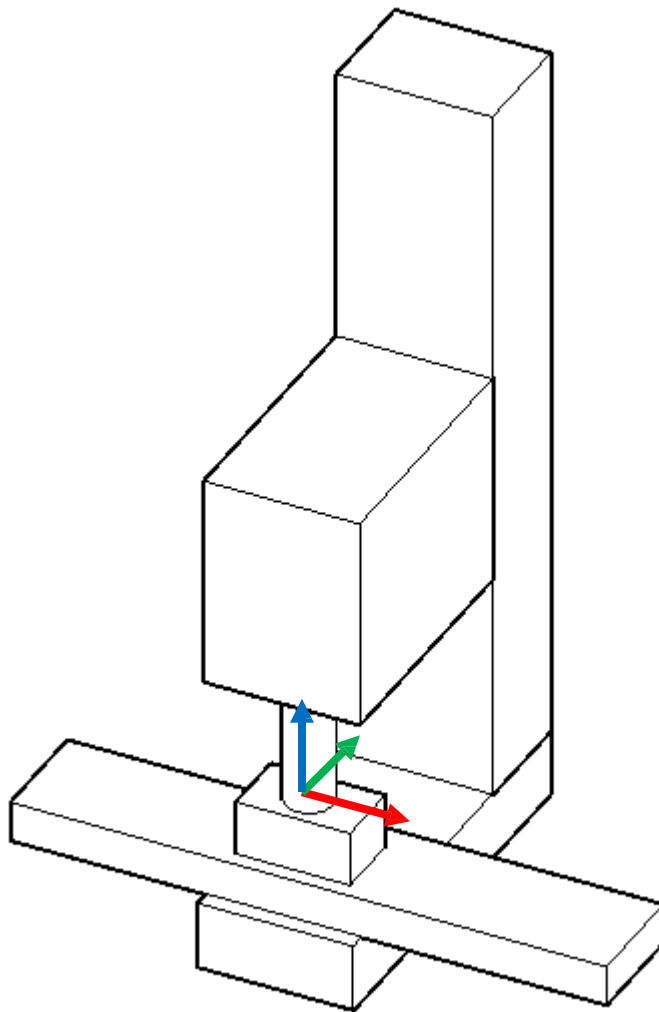


Figure 6: Work Coordinate System

G - Code Programming Language

So how do you make the machine move and actually machine a part? This is done by running a CNC program. A CNC program is loaded into the Control Computer, which then is executed causing the machine to perform the movements programmed in the file.

The *G - Code Programming Language* is the name of the language that the CNC machine can understand and convert into actual motion of the machine. The Control Computer reads the G - Code file line by line and commands the machine to do the movements.

The language consists of a set of codes called G - Codes and M - Codes, each instruction have a specific function, for example to command a motion from one point to another.

The following is an example of a G - Code program:

; Example program

G54 ; Use G54 work coordinates offset

G90 ; Absolute positioning

G0 X0 Y0 Z0 ; Rapid move to (0,0,0)

G1 Z-1 F50 ; Lower the tool to Z = -1 at a speed of 50 mm/min

G1 X30 Y30 F100 ; Move to X = Y = 30 mm simultaneously, 100 mm/min

M30 ; End of program









The G - Code program can be written manually in a file that then is executed, or inputted line by line and executed in real time through the *MDI* input of the Control Computer. These methods are useful if some very simple operations are to be made.

It's much more common to use a *CAM Software* to generate the G - Code. The CAM software imports a 2D or 3D model and generates a G - Code file to cut the part. There is also a simpler type of CAM software called *Wizards* that will generate G - Code for common operations like milling a circular pocket or drilling a hole. Wizards are very handy if a relatively simple part with few operations are to be made. These can be found in the CNC – Software and do not need a CAD model.

A complete list of all the programming codes can be found under the sections 'G - Code Reference', 'M - Code Reference' and 'Prefix Reference'.

Tools

The CNC machine uses different tools to perform various operations on the part. The following list describes some common tools.

| | | |
|-------------------|---|---|
| Square Endmill | The most common tool. Used to make square cuts. |  |
| Ball Endmill | Used to do 3D milling. |  |
| Bull Nose Endmill | Produces a small chamfer at the bottom of the cut. |  |
| Twist Drill | Used to drill holes. |  |
| Center Drill | Used to make a small guide hole for a Twist Drill. This makes sure the Twist Drill does not wobble. |  |
| Face Mill | Used to face mill, face milling produces a flat surface on the part. |  |
| Engraving Tool | Used to engrave text on parts or engrave circuit boards (PCBs). Also called V-bit. |  |
| Chamfer Tool | Used to produce a chamfer on a hole or around the profile of a part. |  |

An endmill can be Center Cutting or Non Center Cutting. A center cutting endmill can be plunged straight down into the material since its cutting edges / flutes go all the way into the center. A non center cutting endmill cannot be plunged and must be ramped into the material.

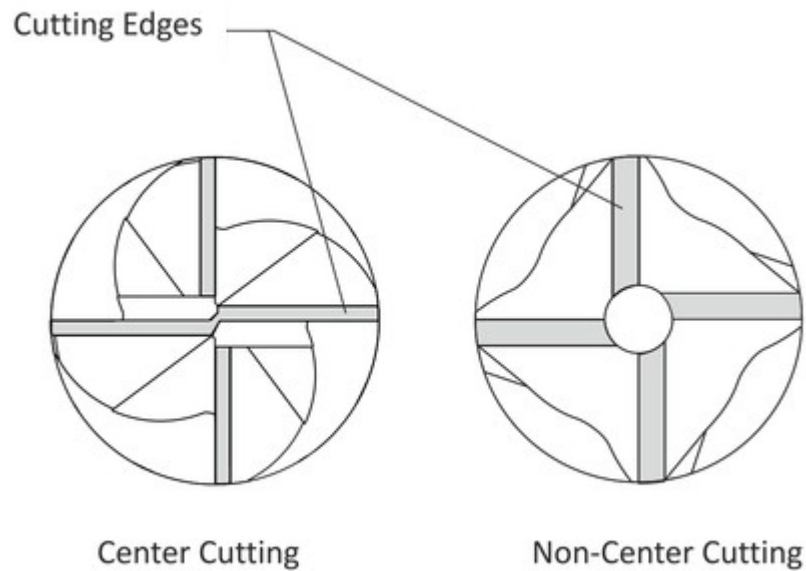


Fig 7: Center Cutting vs Non Center Cutting

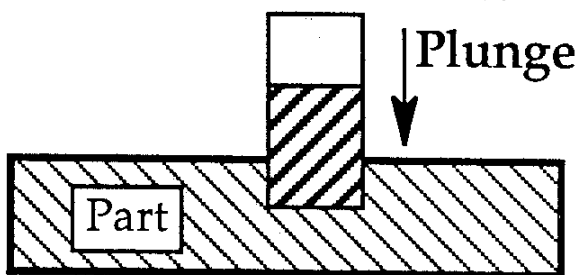


Fig 8: Endmill plunging into material

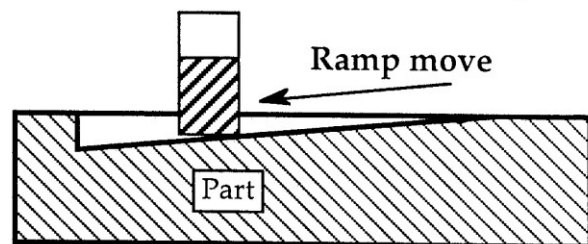


Fig 9: Endmill ramping into material

Climb milling is preferred to use on a CNC milling machine.

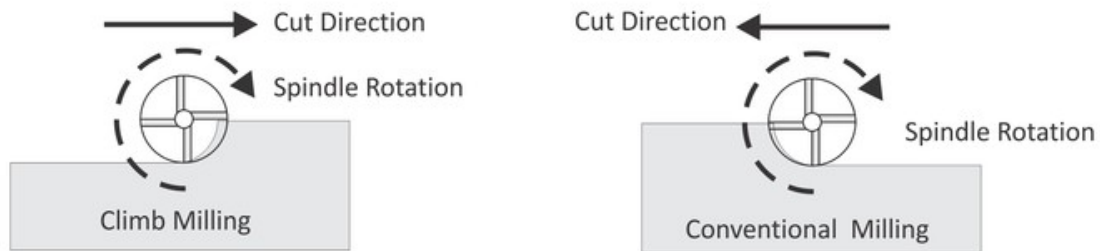


Fig 10: Climb Milling vs Conventional Milling

The number of cutting flutes / edges on the endmill determines the type of material it is designed to cut.

| No. of Flutes vs Intended Material to Cut | |
|---|----------|
| 2 Flutes | Aluminum |
| 3 Flutes | Plastic |
| 4 Flutes | Steel |

Fixtures

The stock material should be clamped to the table in one of the following ways.

Never clamp the stock material directly on the table! Always put a safety sheet of material under the stock in order to prevent the mill cutting into the table!

Clamping on Table

The stock can be clamped onto the table with clamps.

Always place a safety sheet of material under the stock so that the mill does not cut accidentally into the table!

This also enables to cut a bit deeper than the thickness of the stock.

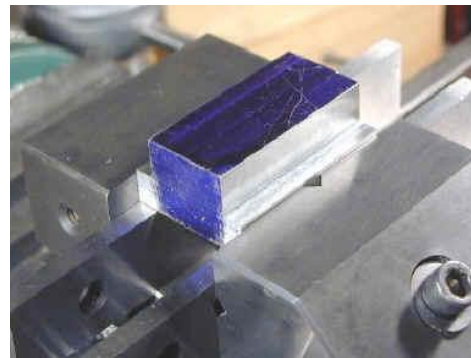


Clamping in Vise

The stock material is often clamped between the jaws in the vise.

Use a pair of parallels to make sure that the stock is clamped parallel to the table.

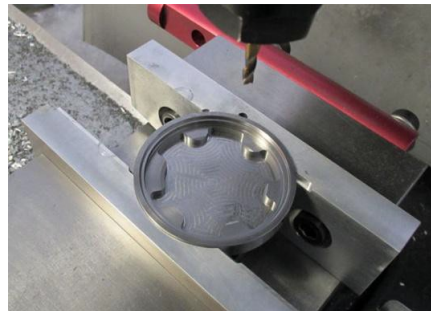
Parallels are a pair of flat bars that can be positioned at the bottom of the vise for the stock material to sit on top of.



Clamping in Vise with Soft Jaws

A pair of soft jaws can be made from aluminum and installed in the vise. The soft jaws can then be machined to accept a stock material with non-square shape.

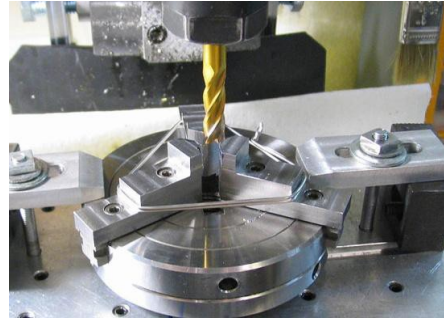
This should not be done to the standard steel jaws of the chuck!



Clamping in Chuck

A lathe chuck can be clamped to the table in which the stock material can be clamped in.

This is useful to do lathe like machining operations on round stock.



Machine Overview

The CNC Mill

The Student CNC machine is a TORMACH PCNC CNC Mill with the following specifications.

| | PCNC 1100 (Series 3) | PCNC 770 (Series 3) | PCNC 440 |
|----------------------------------|--|--|----------------------------------|
| Table Size | 34" x 9.5" | 26" x 8" | 18" x 6.3" |
| Vertical Limit | 17" | 13.25" | 11" |
| Max Spindle to Column | 11" | 8.6" | 6.5" |
| Motion Limits (X x Y x Z) | 18" x 9.5" x 16.25" | 14" x 7.5" x 13.25" | 10" x 6.25" x 10" |
| Typical System Footprint* | 80" x 45" x 82" | 61.5" x 45" x 73" | 42" x 36" x 72" |
| Typical System Weight† | 1580 lbs | 908 lbs | 600 lbs |
| Table Slots | 3 slots 5/8" | 3 slots 5/8" | 3 slots 3/8" |
| Spindle Taper | R8 | R8 | R8 |
| Spindle Power | 1.5 hp Sensorless vector motor | 1 hp Sensorless vector motor | 2/3 hp Brushless DC motor |
| Spindle Speed | 100-5140 RPM | 175-10,000 RPM | 300-10,000 RPM |
| Transmission | 2 ratio V belt | 2 ratio poly V belt | 2 ratio poly V belt |
| Max Feed Rate | 110 IPM (X,Y) 90 IPM (Z) | 135 IPM (X,Y) 110 IPM (Z) | 135 IPM (X,Y) 110 IPM (Z) |
| Tapping | Tension/Compression, Auto-Reversing Thread Mill | Tension/Compression, Auto-Reversing Thread Mill | Thread Mill |
| Power Requirement | 200-250 VAC, single phase** 50/60 Hz | 115VAC, single phase 50/60 Hz | 115VAC, single phase 50/60 Hz |

The maximum size of stock material the CNC Mill can machine is given by its X, Y and Z travel. Keep in mind that this is further reduced by the tool diameter and the length of the tool and tool holder.

The illustration and the description on the next pages points out the basic parts of the CNC Mill that is essential to understand what they are in order to operate the machine.



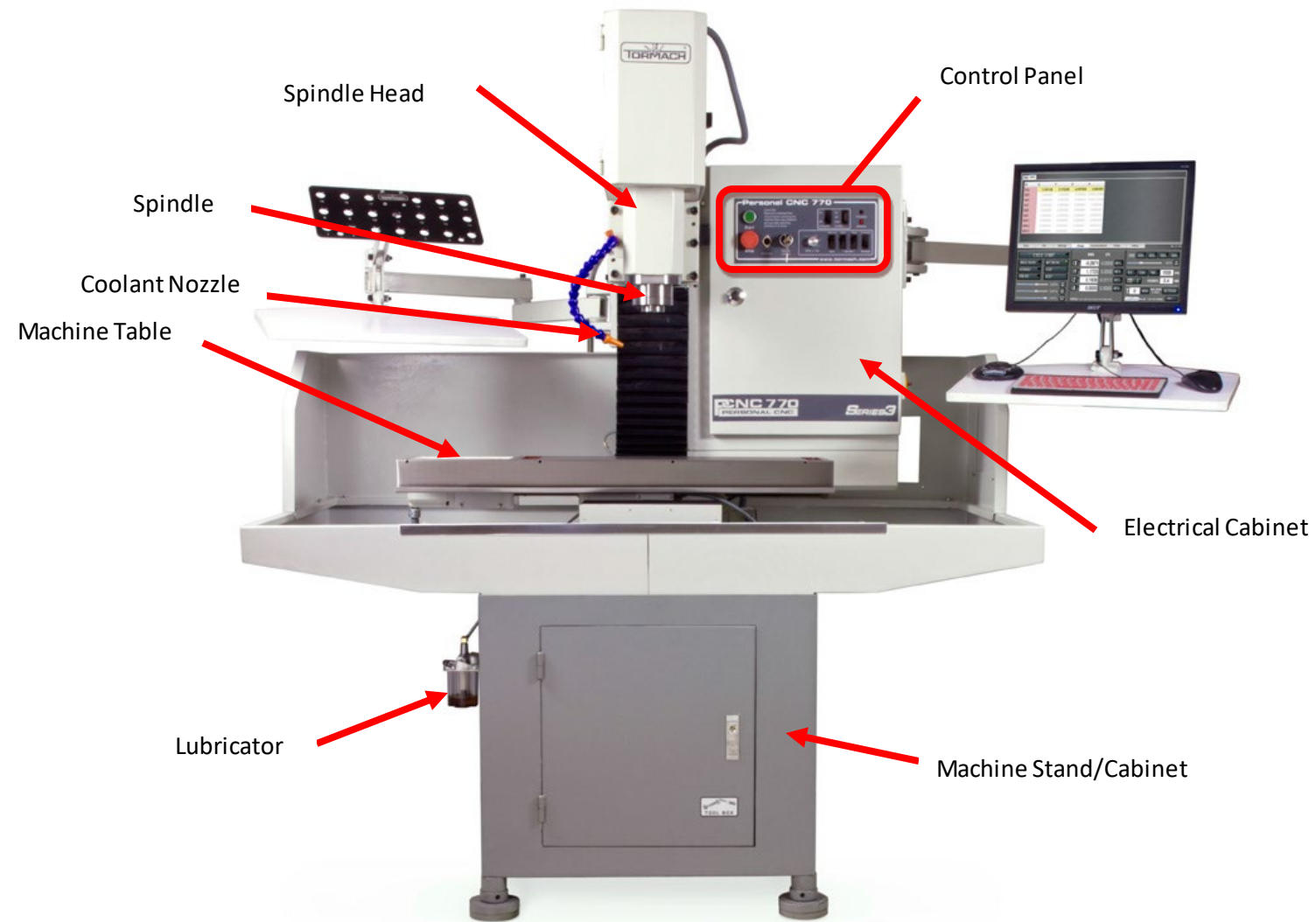
PCNC 1100 (Series 3)



PCNC 770 (Series 3)



PCNC 440





| | |
|------------------------|--|
| Start Button: | The main switch to turn on the power to the machine. |
| Table: | The working area that the CNC Mill can work on, this is what you clamp your work onto. Moves in the X and Y direction. |
| Spindle: | The spindle motor of the machine. This is where you attach the tool or tool holder that is to be used. |
| Lubricator: | The lubrication system of the machine. Pull and release the handle to lubricate the machine. This should always be done before the machine is to be used! |
| Emergency Stop: | Emergency button, will stop the machine if pressed. Must be reset in order for the machine to operate! |
| Spindle Head: | The structure that contains the spindle motor. Moves up and down, is controlled by the Z axis |
| Coolant Nozzle: | The nozzle where the coolant / compressed air is sprayed out from. |
| Machine Stand/Cabinet: | Tools and accessories to the CNC Mill are stored in here. |

Tools and Tool holders

There are many tools available to use with the CNC Mill. Some of the tools are attached to the machine directly by mounting them in the spindle, other tools such as endmills and drills can only be attached to the machine by the use of a tool holder. The TORMACH Machine uses the TTS Tool Holding system.

If using the TTS Tooling system refer to the TTS Tooling system, if using the TTS Tooling system please refer to Technical Document TD10416 – Tormach Tooling System Basics.

Figure 11 illustrates some of the available TORMACH TTS tools and tool holders, Figure 12 illustrates the TTS collet holder.



Fig 11: TTS Tools and Tool holders

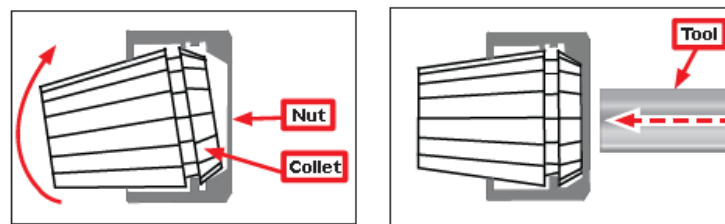


Fig 12: TTS Collet Tool Holder

The TTS system uses a precision $\frac{3}{4}$ " collet in combination with a drawbar and interchangeable TTS tool holders. Many different TTS tool holders are available.

Prior to using TTS for the first time, the TTS collet must be installed. To install the collet:

1. Using a clean rag, apply a degreasing agent to the inside taper of the spindle and also the entire surface of TTS collet. Wipe clean and dry.

2. Apply a small amount of Anti-seize grease (PN 31273) to the following surfaces: drawbar threads, outside taper of TTS collet, inside taper of R8 spindle. Do not apply grease to the inside surface of the TTS collet.

3. Open the spindle door.

Spindle locking fork

4. Swing the spindle locking fork so it engages with the flats on the top of the spindle. If necessary turn the spindle by hand until the flats line up.

Drawbar



5. Using one hand, insert the TTS collet into the bottom of the spindle. Twist the collet while applying light upward pressure until the collet groove aligns with the spindle alignment pin and the collet is pushed completely inside the spindle taper. With the other hand, insert the drawbar into the top of the spindle and rotate the drawbar several turns to engage several threads in the TTS collet.

6. To finish tightening the drawbar, place and hold a TTS tool holder inside the TTS collet and against the spindle nose. Use a 13-mm wrench to rotate the drawbar and tighten the collet against the TTS tool holder.

To change a TTS tool holder, loosen the drawbar by one turn with a 13-mm wrench. Then, grasp the tool holder in one hand and use a mallet to gently strike the top of the drawbar. Remove the tool holder and repeat steps 5 and 6 (above) to install a new tool holder.

Accessories

Clamps are used to clamp stock or accessories such as a vise or a lathe chuck to the table. The table has T - Slots to enable clamps to be used. A clamp is assembled from a T - Slot nut, threaded pin, clamping bar and a step block.



Fig 14: Clamps

Parallels are very precise pairs of flat steel bars in various thickness and height. The parallels are intended to be positioned at the bottom of the vise to make sure that the stock material sits parallel to the table and is clamped as high up in the vise as possible.

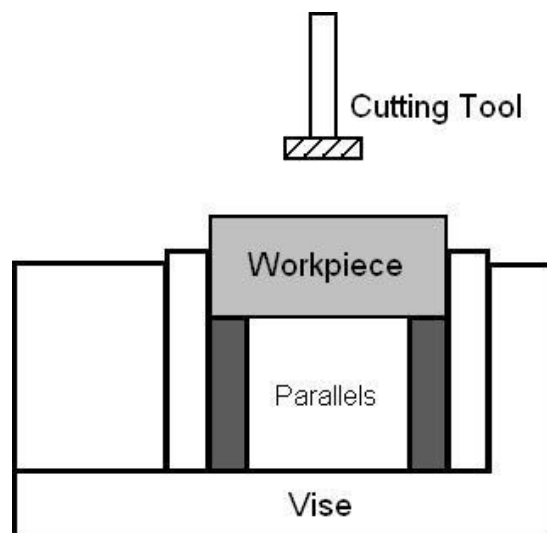


Fig 15: Parallels in Vise

A rotary axis is available to the CNC Mill. The rotary axis is plugged in to the jack labeled 'Rotary Axis' on the side of the machine.

Power on Procedure

Turn the controller on by switching the Main Disconnect from the power Off to the power On position.



Then turn on the controller by setting the Controller On/Off switch to the on position. The Controller On/Off rocker switch is used to power the PathPilot controller on and off.



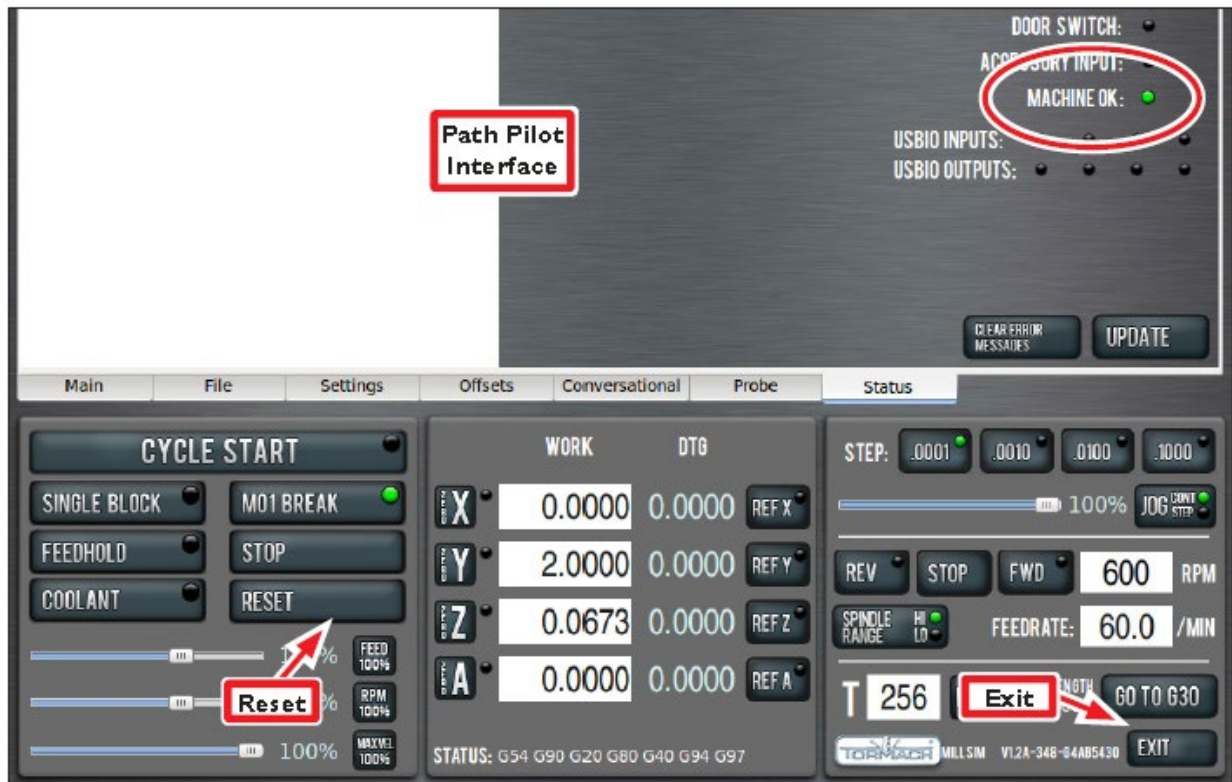
After the PathPilot Interface is loaded, make sure the red E-stop is in the Power On position.



Press the green Start button on the Operator Panel momentarily to power on the mill. When the mill tool has power, the Machine LED light on the Operator Panel illuminates

NOTE: Once the red E-stop has been depressed, the green Start button is inoperative until the E-stop is released.

Click the blinking Reset button on the PathPilot Interface to establish communications between the controller and the mill. When communication is established, the Controller LED light on the Operator Panel illuminates.



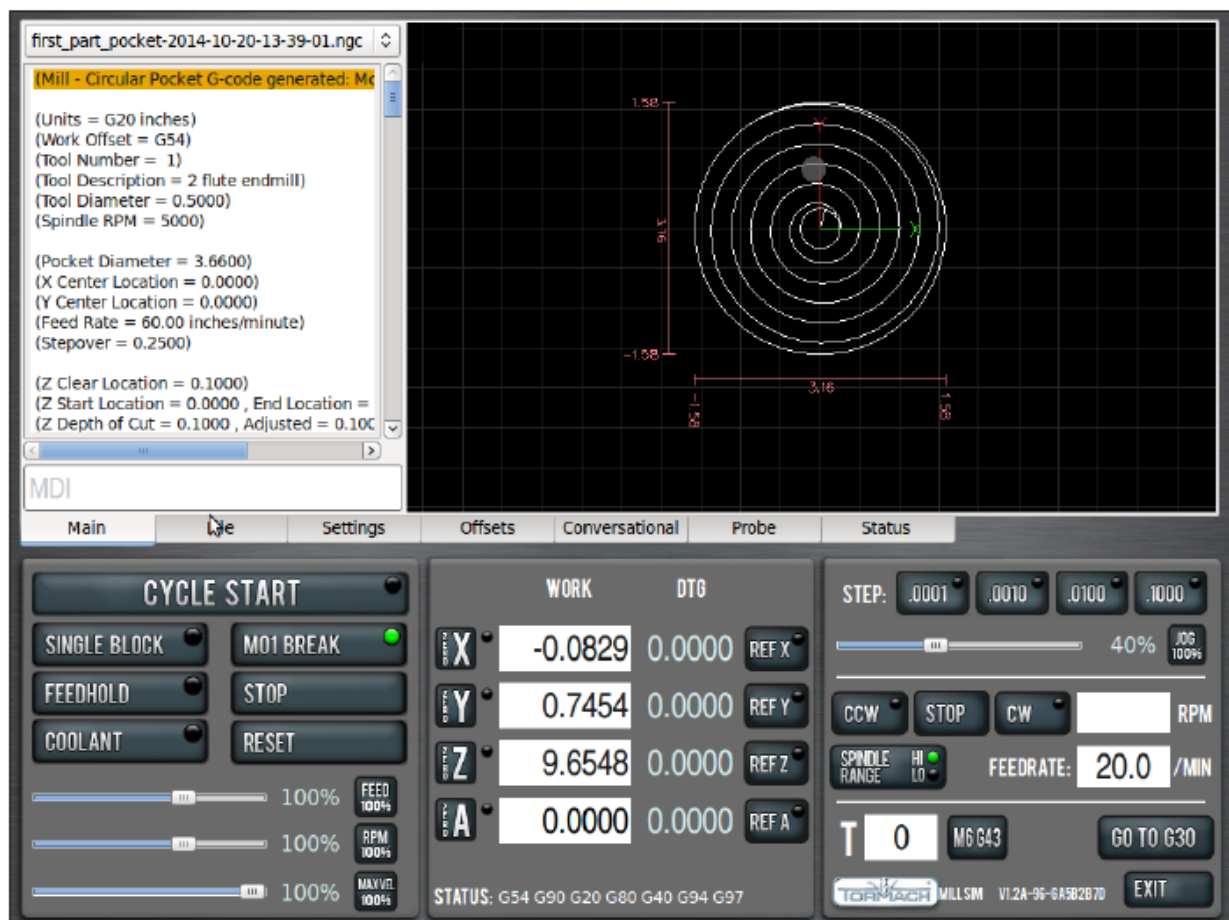
Path Pilot Overview

Intro

The control software of the CNC Mill is the very PathPilot from TORMACH. The software runs under Linux and is easy to use and has many powerful features.

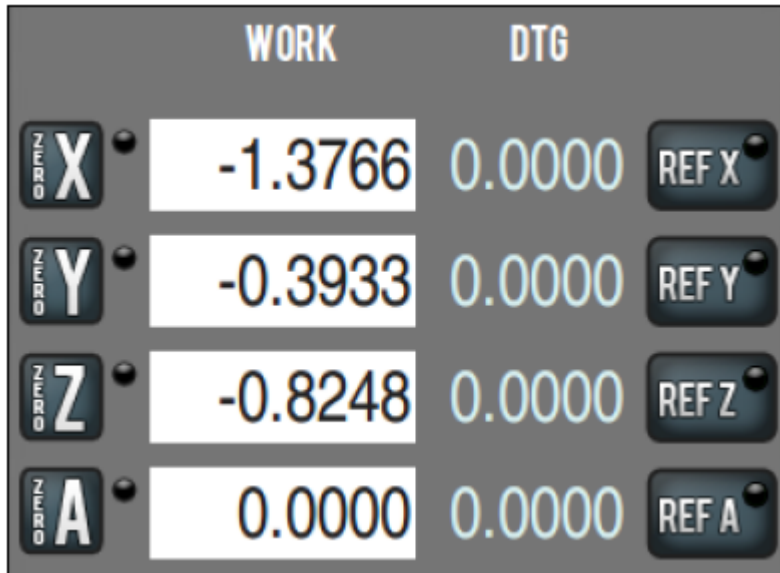
PathPilot will launch automatically when you follow the power up procedures listed above.

This is the PathPilot Main Screen.



Referencing the Mill

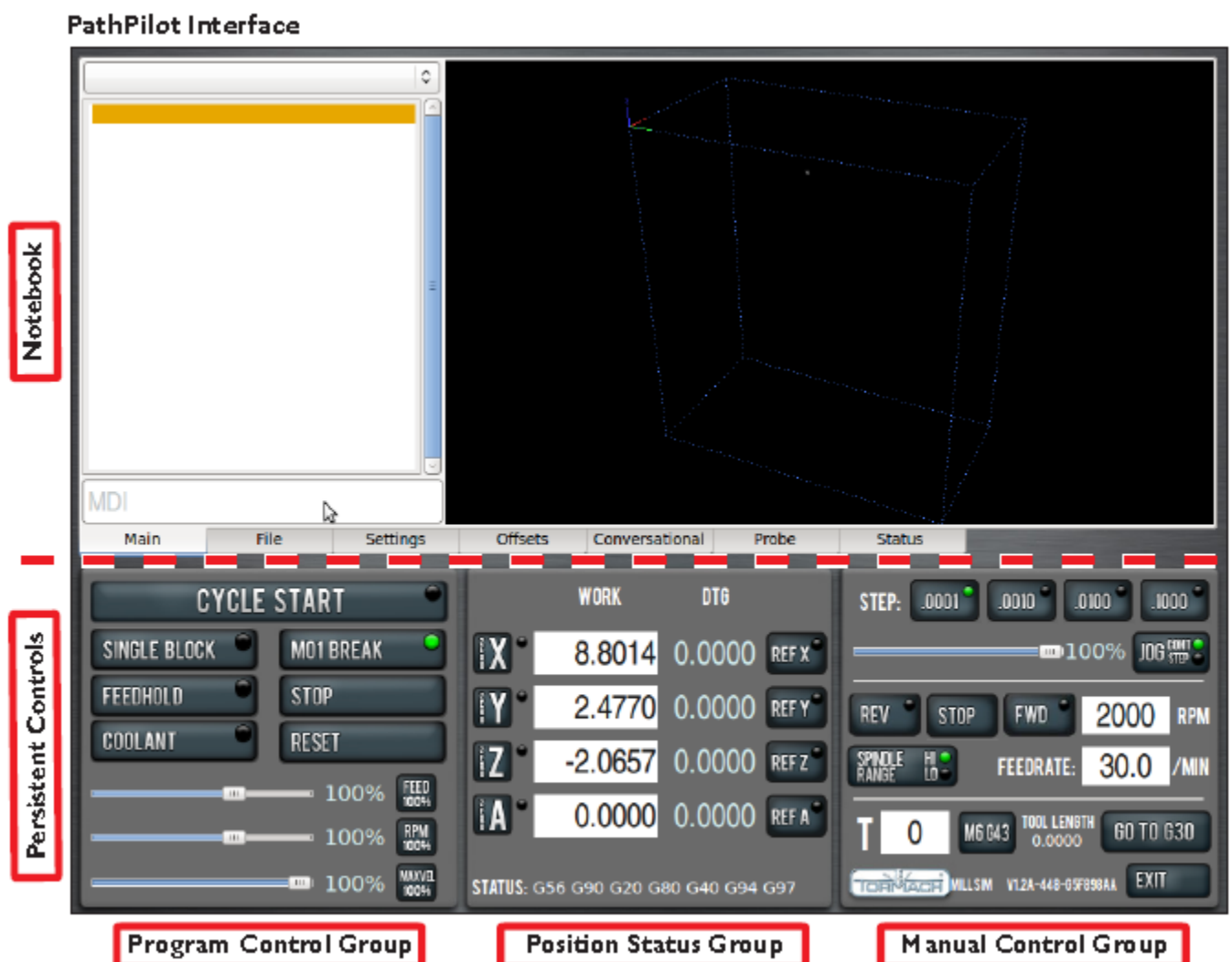
Once the mill is powered on correctly and after clicking the flashing Reset button, you must reference the X-, Y-, and Z-axes. You should reference the Z-axis first to help avert a crash as it moves the tooling as far as possible from a workpiece or vise. All three axes can be referenced simultaneously by pressing the Ref buttons in rapid succession.



The axes should be referenced before operating the mill to establish soft limits to protect the mill from over travel and to give meaning to work offset values. After referencing the axes, the LEDs on the Ref X, Ref Y, and Ref Z buttons turn green, indicating that the mill has been referenced. While you can jog the mill before referencing, you should not run parts until the mill has been referenced.

PathPilot Interface

The PathPilot™ Interface is divided into two sections, the Notebook and the Persistent Controls. The Persistent Controls make up the bottom half of the screen and include three control groups: the Program Control Group, the Position Status Group, and the Manual Control Group. The top half of the screen is the Notebook, which includes seven tabs including Main, File, Settings, Offsets, Conversational, Probe, and Status. Depending on the mill accessories, there may also be optional tabs including ATC (automatic tool changer), Injection Molder, and Scanner. These tabs are used to select different Notebook pages, each of which displays various buttons, digital readouts (DROs), and information pertinent to the functioning of the PathPilot Interface.



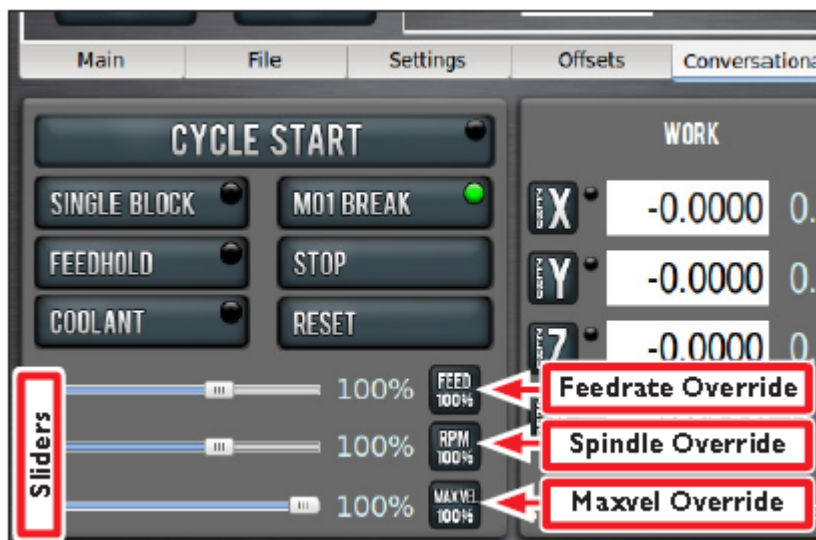
Persistent Controls

The Persistent Controls on the lower half of the screen are always present – they don't move or disappear as you page through the Notebook that makes up the top half of the interface.

These are divided into three logical families: Program Control Group, Position Status Group, and Manual Control Group.

Program Control Group

The buttons, sliders, and readouts of the Program Control Group are all functions that relate to tasks you will perform while running a G-code program. They may be used at any time while running a program, or before running a program to set modes like Single Block or M01 Break.



Cycle Start – The Cycle Start button is used to start a program. While running a program the LED in the upper right-hand corner of the button illuminates.

If Single Block is active, the Cycle Start button causes the mill to execute one line of G-code per click of the button. When running a program, if motion is paused due to Feedhold, M01 Break, Single Block, or because the mill is waiting on a manual tool change, the Cycle Start button LED flashes on and off until the Cycle Start button is pressed again.

You will receive an error if:

- Cycle Start is pressed when the Main tab of the notebook is not active
- Cycle Start is pressed when no G-code program is loaded
- Cycle Start is pressed before the mill has been referenced

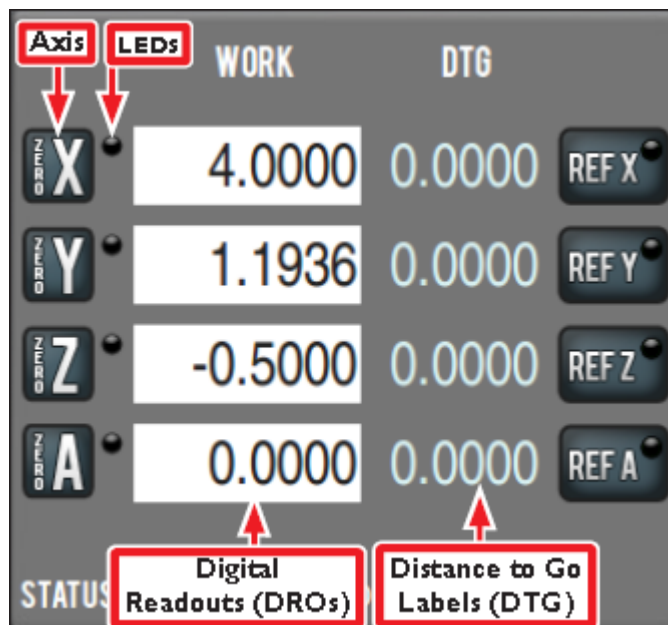
Reset – The Reset button brings the mill out of an E-stop condition, resets G-code modalities, clears alarm messages, and rewinds the G-code program. When the mill is first powered on, or after an emergency stop (E-stop), the Reset button flashes Studica

back and forth between red and white. When this button is flashing (after power has been restored to the mill), clicking the Reset button starts and verifies communication between the mill and the controller.

Position Status Group

The buttons, labels, and DROs of the Position Status Group pertain to mill position, active G-code modalities, and feed/speed settings. These controls may be used at any time before or after running a G-code program or MDI move. They are unavailable for operator input while mill is moving.

The X, Y, Z, and A work offset DROs display the current mill position expressed in the currently active work offset coordinate system (G54, G55, etc.).



NOTE: When the mill is at rest, these readouts are also operator entry fields. Change the current work offset position by clicking in the DRO field, which illuminates. Type a number, for example 4.0, and click Enter on keyboard. Press the Esc key to return to the original value.

Remember when entering a value into a DRO you must hit enter on the keyboard for the value to update.

Manual Control Group

The Manual Control Group's buttons and slider allow the operator to perform tasks related to manual control of the mill, including jogging the mill axes, changing the current tool number, feed rate, or spindle speed, and starting or stopping the spindle.

Jogging Controls – Tormach mills can be jogged with either the Jog Shuttle or with the keyboard's arrow keys:

- The right arrow jogs X-axis in the positive X direction (moves the table to the left of the operator)
- The left arrow jogs X-axis in the negative X direction (moves table right of operator)
- The up arrow jogs Y-axis in the positive Y direction (moves table towards operator)
- The down arrow jogs Y-axis in the negative Y direction (moves table away from operator)
- The Page Up key jogs the Z-axis in the positive Z direction (moves spindle up)
- The Page Down key moves the Z-axis in the negative Z direction (moves spindle down)

NOTE: Jogging is not permitted during G-code program execution or during MDI moves.



Jog Shuttle

This Jog Shuttle is an optional accessory that may increase productivity, especially on short-run jobs requiring extensive setting up of the workpiece and tooling.

The X, Y, Z and A buttons are used to jog axes X, Y, Z and A respectively. The A LED light beside an axis DRO indicates that this axis is selected for jogging. The Step button cycles through the available jog step sizes. The size that is active is indicated by an illuminated LED. Continuous jogging is done with the Shuttle Ring by turning it counter clockwise (minus) and clockwise (plus).



Spindle Controls

The REV, Stop, and FWD buttons can be used to manually control the spindle. Rev is the equivalent of typing M04 in the MDI Line – it starts the spindle counter clockwise at the RPM specified in the Spindle RPM DRO.

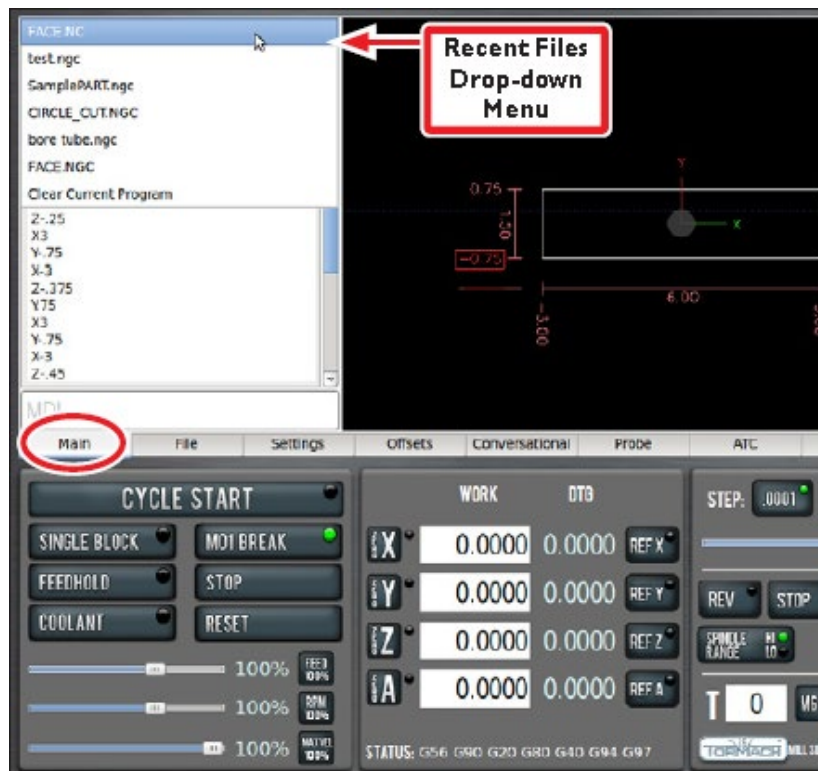


The Stop button stops the spindle, similar to the M5 command. The FWD button starts the spindle clockwise at the set RPM.

Main Tab

Clicking the Main tab of the notebook brings up the Main screen. This screen is active by default when the PathPilot controller first powers on.

The Main screen contains four controls – the Recent Files/Current File Display, the Tool Path Preview, the G-code window, and the MDI Line.

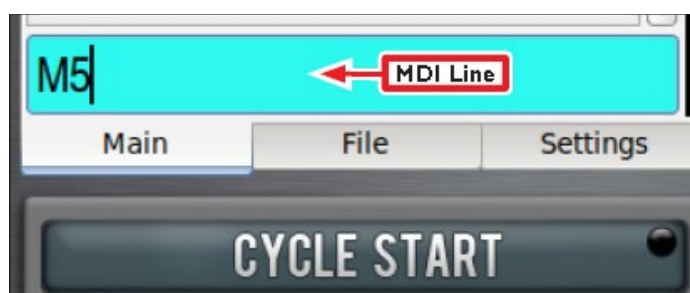


G-code Window/Set Start Line

The G-code window displays the G-code in the currently loaded file. You may use the scroll bars to scroll down to see the entire file. The start line (the line at which the G-code program is executing) is highlighted in green. This usually is the first line of the code; to change this, place the mouse pointer over the preferred start line, right click, and choose the Set Start Line option.

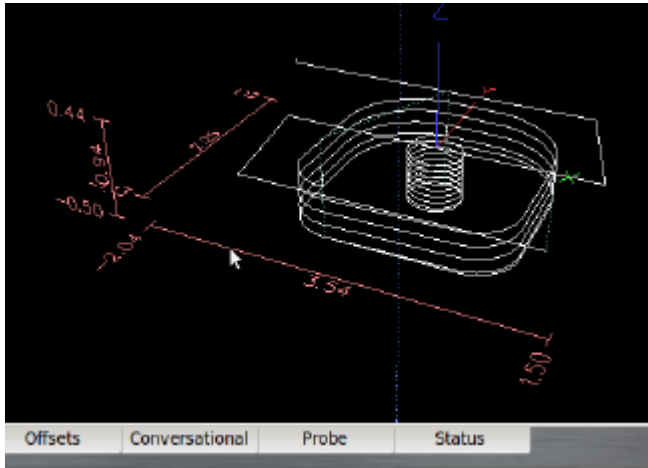
MDI Line

When running a G-code part program, commands to the mill are read from a file. It is often convenient to send G-code commands to the mill directly. This can be done by typing a command into the Manual Data Input line (MDI Line)



Tool Path Display

The tool path window displays a graphic representation of the tool path that is executed for the currently loaded G-code file. Preview lines are shown in white, the tool path as it is cut is red, and jogging moves are yellow. The boundary box, drawn in dotted blue, represents the ends of travel of the axes.



Four views are available: top, front, right, and ortho. The default view after loading a new G-code program is top. To switch between views, right click anywhere in the tool path display and choose the desired view.

File Tab

The File screen is used for transferring files to and from a USB drive, copying, deleting, and renaming files and folders. On the left is a window showing files and folders on the controller hard drive, and on the right, is a window showing files and folders on a removable USB drive.

Note that the controller does not run programs from the USB drive. Programs must first be copied to the controller before they can be loaded and run.

Edit G-code

The Edit G-code window is a preview window displaying the contents of the selected .nc file.

1. To edit the G-code, highlight the file and click the Edit G-code button.
2. A text editor opens the file in a new window for editing the contents of the file. Make the appropriate changes to the file and click Save.
3. Close the text editor by clicking on the X in the upper right-hand window of the screen.
4. When asked to re-load the file, click OK.

Load G-Code

The Load G-Code function is only available for files stored on the controller.

1. Navigate to the desired .nc file in the control window.
2. With the file highlighted, click Load G-Code button.
3. Click on the Main file tab to open the Main screen.
4. In the Main screen, verify the G-Code file name appears in the drop-down menu.

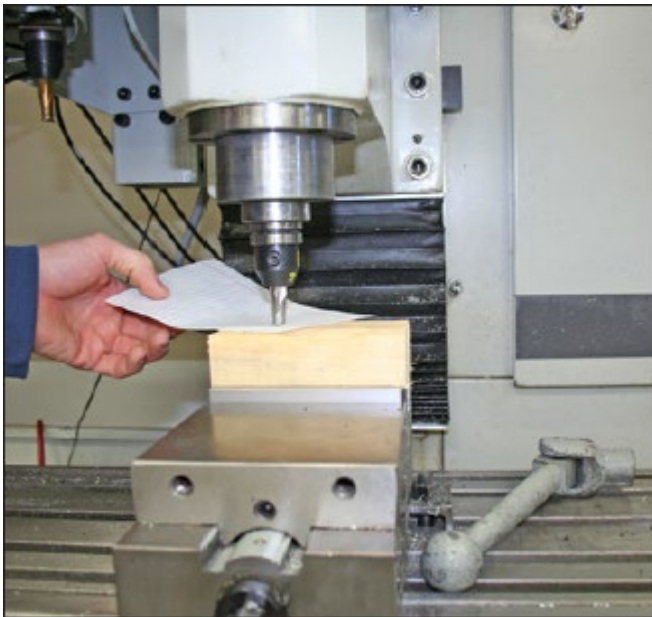
Setup Work Coordinate System Origin

The Work Coordinate system is setup by moving the endmill to the place of where the operator would like to place the origin of the Work Coordinate System, and then Zero all the coordinates values of the three axis X, Y and Z.

The accuracy required of the position of the origin of this coordinate system can vary.

If the stock material is sufficiently oversized of the part to be milled, the position of the origin can be eyeballed. The endmill is simply positioned approximately where the origin is set in the CAD/CAM program and then the X, Y and Z axis is zeroed.

If more precision is required, use an edge finder and the touch tool method.

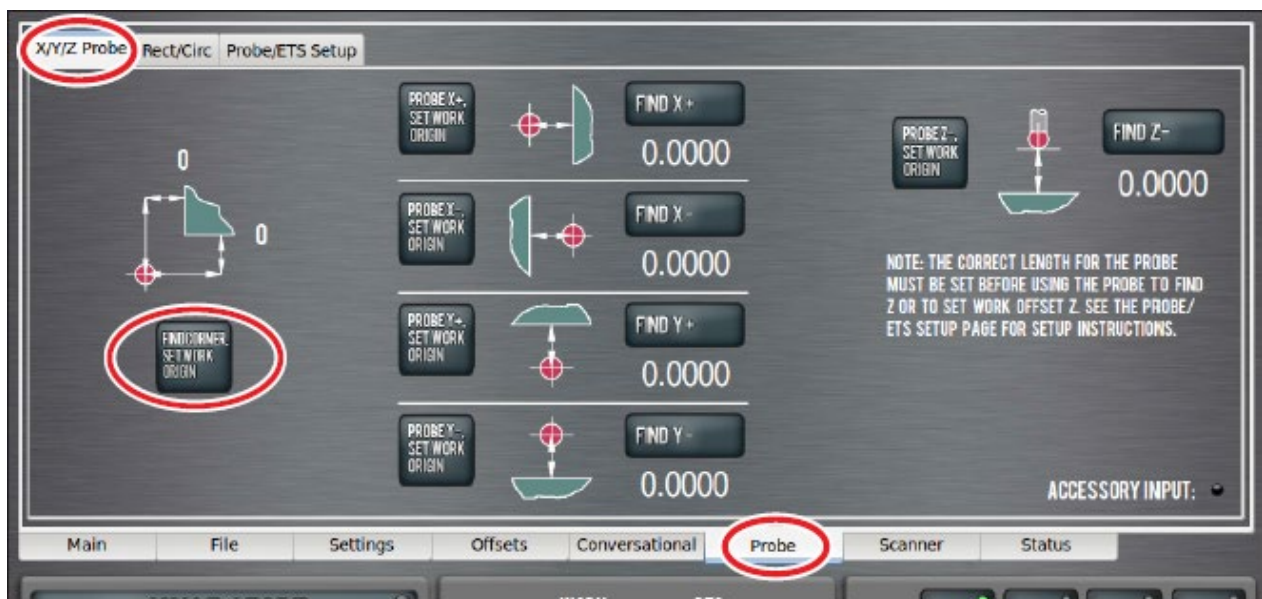


If more accuracy is required a TTS probe or ZERO Master can be used to find the Z height of a surface and the Touch probe can be used to find edges and centers of holes. In this way, the origin of the Work Coordinate System can be set exactly at the tip of a corner for an example.

Probe Tab

The Probe tab of the notebook contains automated functions to find X/Y/Z locations, set work offsets, probe pockets, slots or bosses, as well as instructions on probe and toolsetter setup and calibration. The Probe tab contains a separate, smaller notebook that consists of three tabs:

1. X/Y/Z Probe
2. Rect/Circ Probe
3. Probe/ETS Setup



Tool 99 (the probe tool) must be the current tool in spindle before using any of the probing functions. All probing moves occur at a feed rate specified by the current F command.

CAD/CAM Software Overview

Intro

The design of a part starts with a drawing it in the CAD program. The part should be drawn as a 3D model. When this is done the next step is to use the CAM part of the software to specify the cutting operations that should be done in order to produce the part from a stock material, this can then be exported as a G - Code file to be executed on the CNC Mill.

The workflow is thus as follows:

3D Modeling → Define Stock Size → Specify Cutting Operations → Export G - Code

Autodesk Fusion 360

Autodesk Fusion 360 is a CAD and CAM software package that can be used to do 3D modeling and generate toolpaths / G-code to be executed on the CNC mill.

The software is free for students and can be downloaded from Autodesk after creating an Autodesk account.

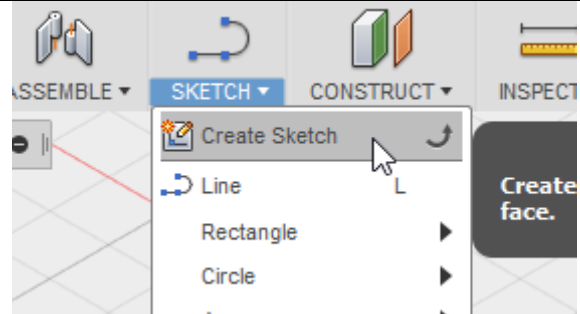
Autodesk Fusion 360 Design Workflow

CAD Modeling in Autodesk Fusion 360

1.

Start Autodesk Fusion 360, a new file 'Untitled' will be created automatically. Save it in a project folder and give it a describing name.

Create a new sketch and select the plane to draw it in.

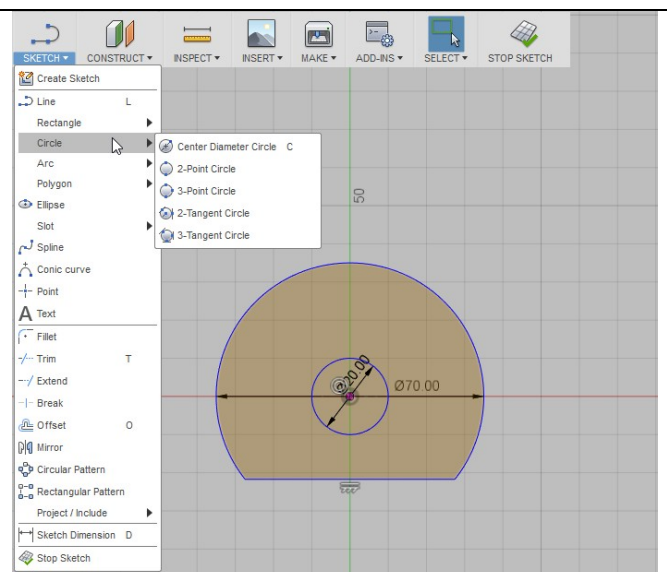


2.

Draw the 2D geometry of the part.

Use the drawing tools in the 'SKETCH' menu.

When the sketch is done, return to 3D view by clicking 'STOP SKETCH'.



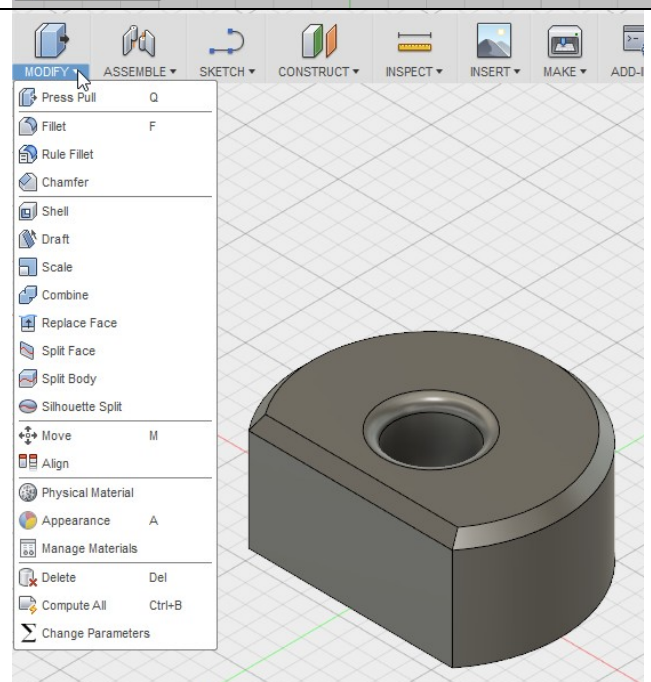
3.

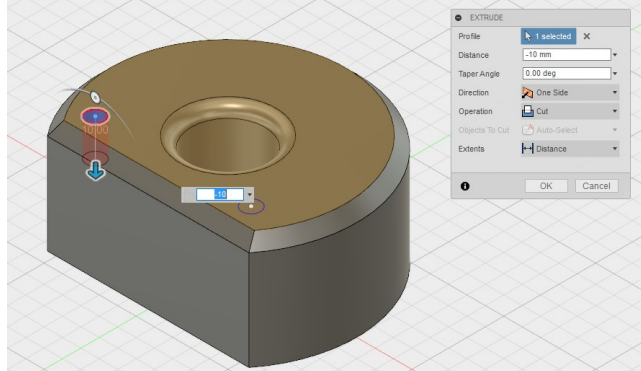
Use the tools in the 'MODIFY' menu to pull the sketch into a 3D geometry and modify it.

Press Pull = Raise 2D into 3D

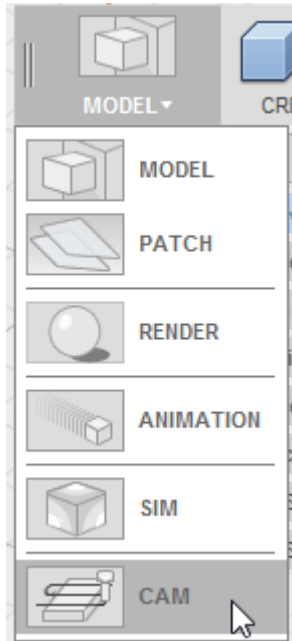
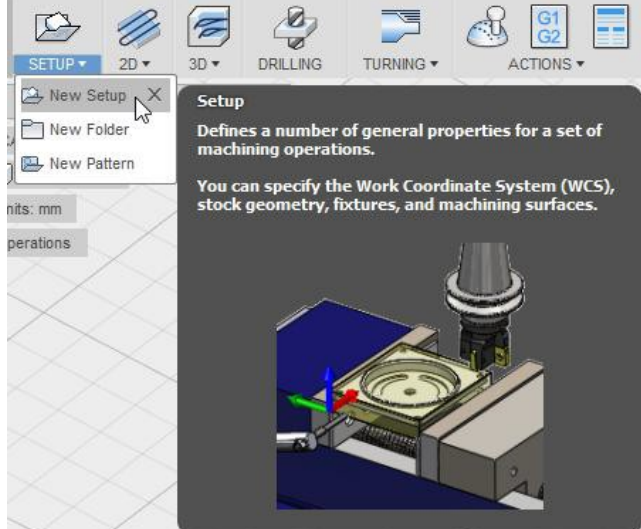
Chamfer = Create a chamfered edge

Fillet = Create an edge with a radius



| | | |
|----|--|--|
| 4. | Create additional sketches if necessary. Modify them into 3D geometry. |  |
|----|--|--|

CAM Process in Autodesk Fusion 360

| | | |
|----|--|--|
| 1. | Select CAM mode. |  |
| 2. | <p>Create a new setup.</p> <p>Select the origin point under the 'Setup' tab.</p> <p>Specify the stock material shape and size under the 'Stock' tab.</p> |  |

3.

Create the necessary cutting operations.

Choose the cutting operation from the 2D menu.

Select the cutting tool to be used.

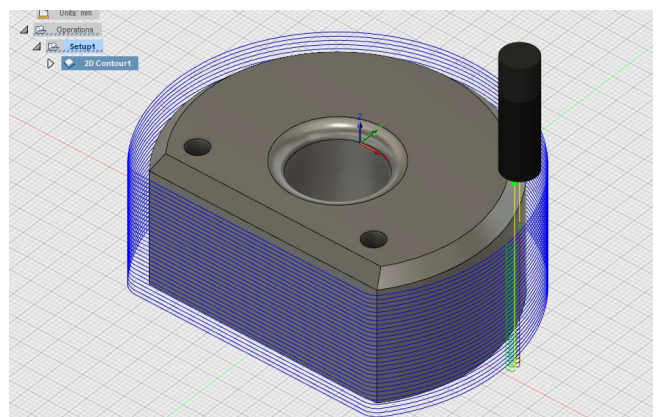
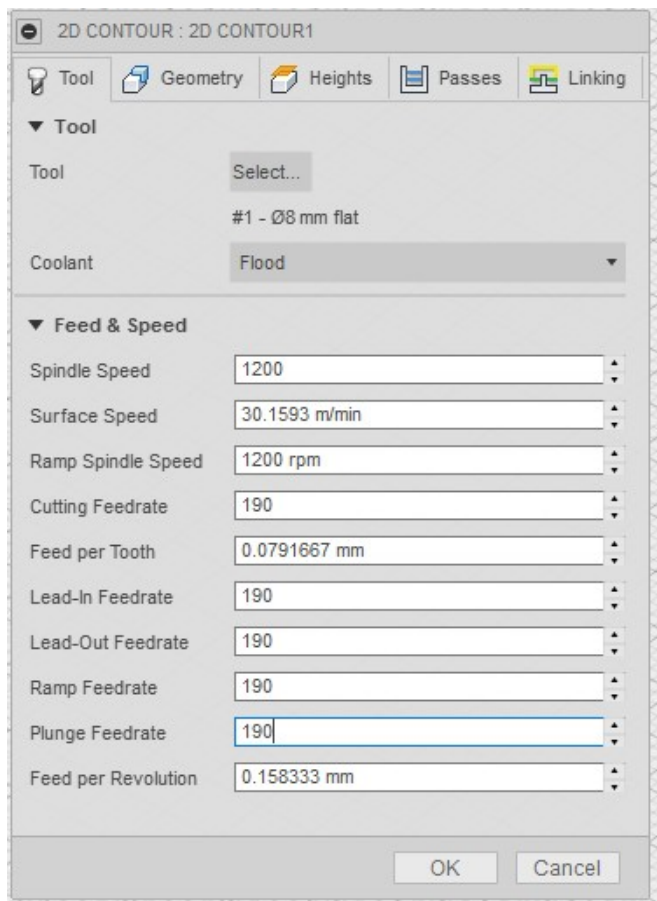
Fill in the Spindle Speed, Ramp Spindle Speed, Cutting Feedrate, Lead-In Feedrate, Lead-Out Feedrate, Ramp Feedrate and Plunge Feedrate according to the Speeds and Feeds Table.

Select the geometry to be cut.

Select the heights of the depth of the cut.

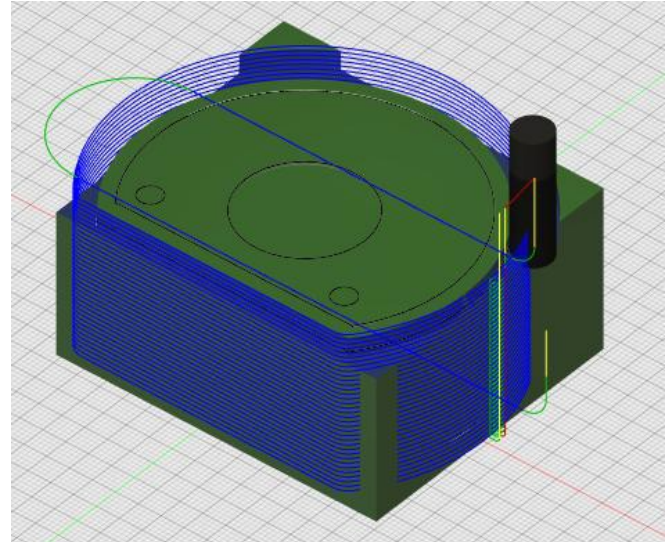
Select the depth per pass under 'Passes' tab.

Keep in mind how the part should be fixtured / clamped to the table. It is common to machine it halfway and then flip it around in order to machine the second half.



4.

Simulate to make sure that the toolpaths are correct.

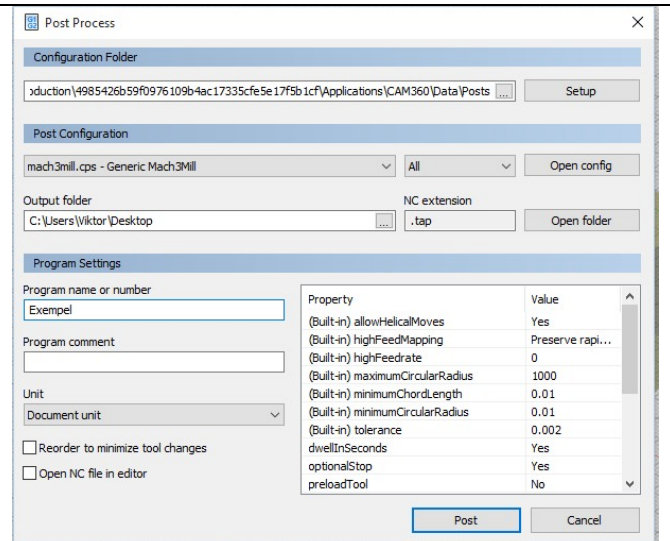
**5.**

Generate the G -Code by selecting 'Post Process'

Select 'mach3mill.cps' under 'Post Configuration'

Give the program a name under 'Program name or number'

Hit OK , save the file and head to the CNC Mill!





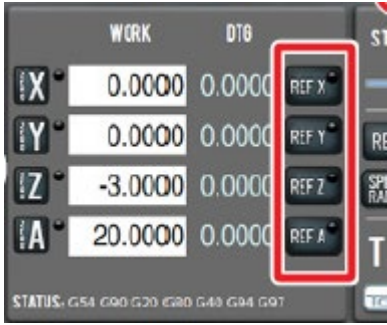

Workflow of Machining a Part

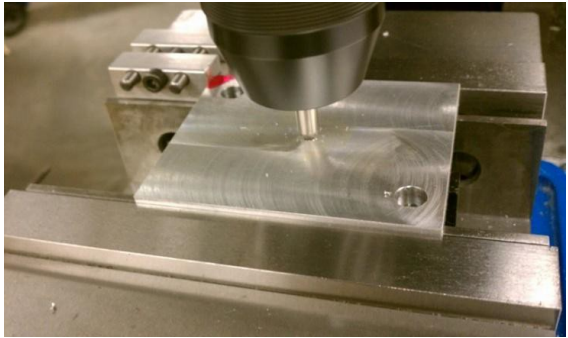

This section describes step by step how to run a typical job on the CNC machine.

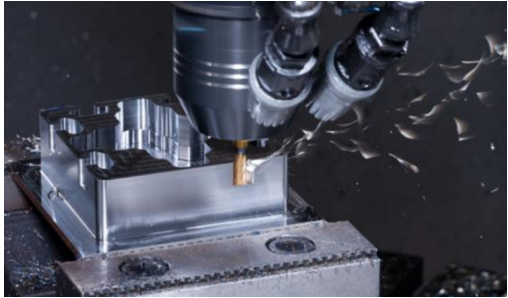
Make sure to take the time to read the previous sections, you will not be successful in operating the machine if you don't understand the basics of CNC machining. It is also important to follow the safety rules in order to not harm yourself or others.

Read and understand the Safety Rules before operating the machine!

| Workflow of Machining a Part | | |
|------------------------------|--|---|
| 1. | <p>Generate the G - Code either by:</p> <ul style="list-style-type: none"> - Autodesk Fusion 360 - Mach 3 Wizards - Writing it manually <p>Make sure that the Feeds, Speeds and Maximum Depth Of Cut are set accordingly to the 'Speeds and Feeds' table.</p> |  |
| 2. | Find a suitable stock material. Cut it to be a bit oversized to the part that you are machining. | |
| 3. | Turn on the Path Pilot Controller | |
| 4. | Turn on the power to the mill | |
| 5. | Pull and release the oiler to lubricate the machine | |
| 6. | Reset Path Pilot by clicking the 'Reset' button. |  |

| | | |
|------------|---|---|
| 7. | <p>Reference the machine by performing the homing cycle. Press the 'REF' button for each axis,</p> <p>This will reference the Machine Coordinates, making sure that the offsets work.</p> |  A photograph of a CNC control panel. The panel displays 'WORK' and 'DRO' coordinates for X, Y, Z, and A axes. The X-axis is at 0.0000, Y at 0.0000, Z at -3.0000, and A at 20.0000. To the right of these displays are buttons labeled 'REF X', 'REF Y', 'REF Z', and 'REF A'. These four buttons are enclosed in a red rectangular box. |
| 9. | <p>Mount the stock material to the table either by:</p> <p>Clamping it directly to the table with a safety sheet of material underneath.</p> <p>Clamping it in the vise. Use parallels to make sure it is parallel to the table.</p> <p>Clamping it in the lathe chuck.</p> |  A photograph showing a rectangular piece of light-colored stock material mounted on a CNC table. The material is held in place by a metal vise on the left and a parallel bar on the right. A black safety sheet is visible underneath the material. |
| 10. | <p>Insert the cutting tool into the spindle. Make sure that it is securely tightened</p> | |

| | | |
|-------------------|---|---|
| <p>11.</p> | <p>Set the origin point of the coordinate system (G54 Work Offset).</p> <p>Do this by jogging the machine until the tip of the endmill is in the appropriate spot in the XY plane and directly above the stock material. Zero the X, Y and Z axis.</p> <p>The Touch Probe can also be used to automatically and very accurately set the Z height.</p> |  |
| <p>12.</p> | <p>Move the Z axis up 1 inch. Do this with the G - Code 'G0 Z1' in the MDI box on the tab MDI'.</p> <p>Then zero the Z axis.</p> <p>This is so that the program first can be tested without cutting into the material. This is known as 'cutting air'. This is to determine that the program works as indented before the real cut is made.</p> | |
| <p>13.</p> | <p>Load the G - Code into Path Pilot by clicking 'Load G - Code' under the file tab.</p> | |
| <p>14.</p> | <p>Start the program by pressing 'Cycle Start'.</p> |  |
| <p>15.</p> | <p>When the program is verified to work correctly:</p> <p>Stop the program with the 'Stop' button.</p> | |

| | | |
|------------|---|--|
| | <p>Lower the Z axis 1 inch with the G - Code command 'G0 Z-1'.</p> <p>Zero the Z axis.</p> <p>Start the program by pressing 'Cycle Start'.</p> | |
| 16. | <p>Monitor the machine while it is working. Do not leave it unattended. You need to be around and stop it if something goes wrong.</p> <p>When it is running, chips need to be blown away and cutting oil applied if steel materials are cut.</p> |  |
| 17. | <p>When you are done using the machine it should be closed down in the order:</p> <p>Choose Exit in Path Pilot and follow the prompts.</p> | |
| 18. | <p>Cleaning. The machine should be completely cleaned when you are done using it.</p> <p>Use a dustpan and brush to clean the chips.</p> <p>Wipe off oil from the machine with paper towels.</p> <p>Vacuum clean the machine and the floor.</p> | |
| 19. | <p>Clean all the tools you have used and put them back in their designated place in the tool cabinet.</p> | |

Speeds and Feeds Table

| Material | Tool Diameter (mm) | Feed Rate (mm/min) | | | Plunge Feed Rate | Spindle RPM | Maximum Cutting Depth (mm) |
|------------|--------------------|--------------------|-----|-----|------------------|-------------|----------------------------|
| | | Cutting Flutes | | | | | |
| | | 2 | 3 | 4 | | | |
| Mild Steel | 3 | 95 | 140 | 190 | 50 | 3100 | 0.3 |
| | 4 | 95 | 140 | 190 | 50 | 2400 | 0.4 |
| | 6 | 95 | 140 | 190 | 50 | 1600 | 0.5 |
| | 8 | 95 | 140 | 190 | 50 | 1200 | 0.8 |
| | 10 | 95 | 140 | 190 | 50 | 950 | 1 |
| | 12 | 95 | 140 | 190 | 50 | 800 | 1 |
| | 15 | 95 | 140 | 190 | 50 | 700 | 1 |
| | 20 | 95 | 140 | 190 | 50 | 750 | 0.5 |
| | 63 | 95 | 140 | 190 | 50 | 1000 | 0.5 |
| | | | | | | | |
| Hard Steel | 3 | 40 | 60 | 75 | 25 | 1300 | 0.15 |
| | 4 | 40 | 60 | 75 | 25 | 1000 | 0.2 |
| | 6 | 40 | 60 | 75 | 25 | 600 | 0.25 |
| | 8 | 40 | 60 | 75 | 25 | 500 | 0.35 |
| | 10 | 40 | 60 | 75 | 25 | 400 | 0.5 |
| | 12 | 40 | 60 | 75 | 25 | 300 | 0.5 |
| | 15 | 40 | 60 | 75 | 25 | 250 | 0.75 |
| | 20 | 40 | 60 | 75 | 25 | 500 | 0.5 |
| | 63 | 40 | 60 | 75 | 25 | 1000 | 0.25 |
| | | | | | | | |
| Aluminum | 3 | 238 | 360 | 500 | 100 | 5000 | 0.5 |
| | 4 | 238 | 360 | 500 | 100 | 5000 | 0.5 |
| | 6 | 238 | 360 | 500 | 100 | 4000 | 0.8 |
| | 8 | 238 | 360 | 500 | 100 | 3000 | 1 |
| | 10 | 238 | 360 | 500 | 100 | 2400 | 1 |
| | 12 | 238 | 360 | 500 | 100 | 2000 | 1 |
| | 15 | 238 | 360 | 500 | 100 | 1600 | 1 |
| | 20 | 238 | 360 | 500 | 100 | 1200 | 1 |
| | 63 | 238 | 360 | 500 | 100 | 1000 | 1 |

| Material | Tool Diameter (mm) | Feed Rate (mm/min) | | | Plunge Feed Rate | Spindle RPM | Maximum Cutting Depth (mm) |
|----------|--------------------|--------------------|-----|-----|------------------|-------------|----------------------------|
| | | Cutting Flutes | | | | | |
| | | 2 | 3 | 4 | | | |
| Plastic | 3 | 95 | 140 | 190 | 100 | 3100 | 0.3 |
| | 4 | 95 | 140 | 190 | 100 | 2400 | 0.4 |
| | 6 | 95 | 140 | 190 | 100 | 1600 | 0.5 |
| | 8 | 95 | 140 | 190 | 100 | 1200 | 0.8 |
| | 10 | 95 | 140 | 190 | 100 | 950 | 1 |
| | 12 | 95 | 140 | 190 | 100 | 800 | 1 |
| | 15 | 95 | 140 | 190 | 100 | 700 | 1 |
| | 20 | 95 | 140 | 190 | 100 | 500 | 1 |
| | 63 | 95 | 140 | 190 | 100 | 1000 | 1 |

Prefix Reference

| | |
|---|---|
| A | A axis of machine |
| B | B axis of machine |
| C | C axis of machine |
| D | Tool radius compensation number |
| F | Feedrate |
| G | G - Code |
| H | Tool length offset index |
| I | X axis offset for arcs |
| | X offset in G87 canned cycle |
| J | Y axis offset for arcs |
| | Y offset in G87 canned cycle |
| K | Z axis offset for arcs |
| | Z offset in G87 canned cycle |
| L | Number of repetitions in canned cycles/subroutines |
| | L1 / L2 tool offsets settings / fixture offset (with G10) |
| M | M -Code |
| N | Line number |
| O | Subroutine label number |
| P | Dwell time in canned cycles |
| | Dwell time with G04 |
| | Tool / Fixture number (with G10) |
| | Tool radius (G41, G42) |
| Q | Feed increment in G83 canned cycle |
| | Repetitions of subroutine call |
| R | Arc radius |
| | Canned cycle retract level |
| S | Spindle speed |
| T | Tool selection |
| X | X axis of machine |
| Y | Y axis of machine |
| Z | Z axis of machine |

G - Code Reference

| |
|--|
| G0 Rapid positioning |
| G1 Linear interpolation |
| G2 Clockwise circular/helical interpolation |
| G3 Counterclockwise circular/Helical interpolation |
| G4 Dwell |
| G10 Set tool table entry |
| G17 XY Plane select |
| G18 XZ plane select |
| G19 YZ plane select |
| G20/G21 Inch/Millimeter unit |
| G28 Return home |
| G28.1 Reference axes |
| G30 Return home |
| G33 Spindle Sync motion |
| G40 Cancel cutter radius compensation |
| G41/G42 Start cutter radius compensation left/right |
| G43 Apply tool length offset (plus) |
| G49 Cancel tool length offset |
| G53 Move in absolute machine coordinate system |
| G54 Use fixture offset 1 |
| G55 Use fixture offset 2 |
| G56 Use fixture offset 3 |
| G57 Use fixture offset 4 |
| G58 Use fixture offset 5 |
| G59 Use fixture offset 6 / use general fixture number |
| G61/G61.1 Path control mode |
| G70/G71 Inch/Millimeter unit |
| G73 Canned cycle - peck drilling |
| G80 Cancel motion mode |
| G81 Canned cycle - drilling |
| G82 Canned cycle - drilling with dwell |
| G83 Canned cycle - peck drilling |
| G85/G86/G88/G89 Canned cycle - boring |
| G90 Absolute distance mode |
| G90.1 Absolute IJK mode |
| G91 Incremental distance mode |
| G91.1 Incremental IJK mode |
| G92 Offset coordinates and set parameters |
| G92.x Cancel G92 etc. |
| G93 Inverse time feed mode |
| G94 Units Per Min. |
| G98 Initial level return / R-point level after canned cycles |

M - Code Reference

| | |
|-----|---------------------------------|
| M00 | Program stop |
| M01 | Optional program stop |
| M02 | Program end |
| M03 | Start spindle clockwise |
| M04 | Start spindle counterclockwise |
| M05 | Stop spindle |
| M06 | Tool change |
| M07 | Mist coolant on |
| M08 | Flood coolant on |
| M09 | All coolant off |
| M30 | Program end and rewind |
| M48 | Enable speed and feed override |
| M49 | Disable speed and feed override |
| M98 | Call subroutine |
| M99 | Return from subroutine/repast |