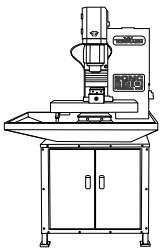
Tormach PCNC440 Mill

Manual

The
Creative
School

Design +
Technology LAB

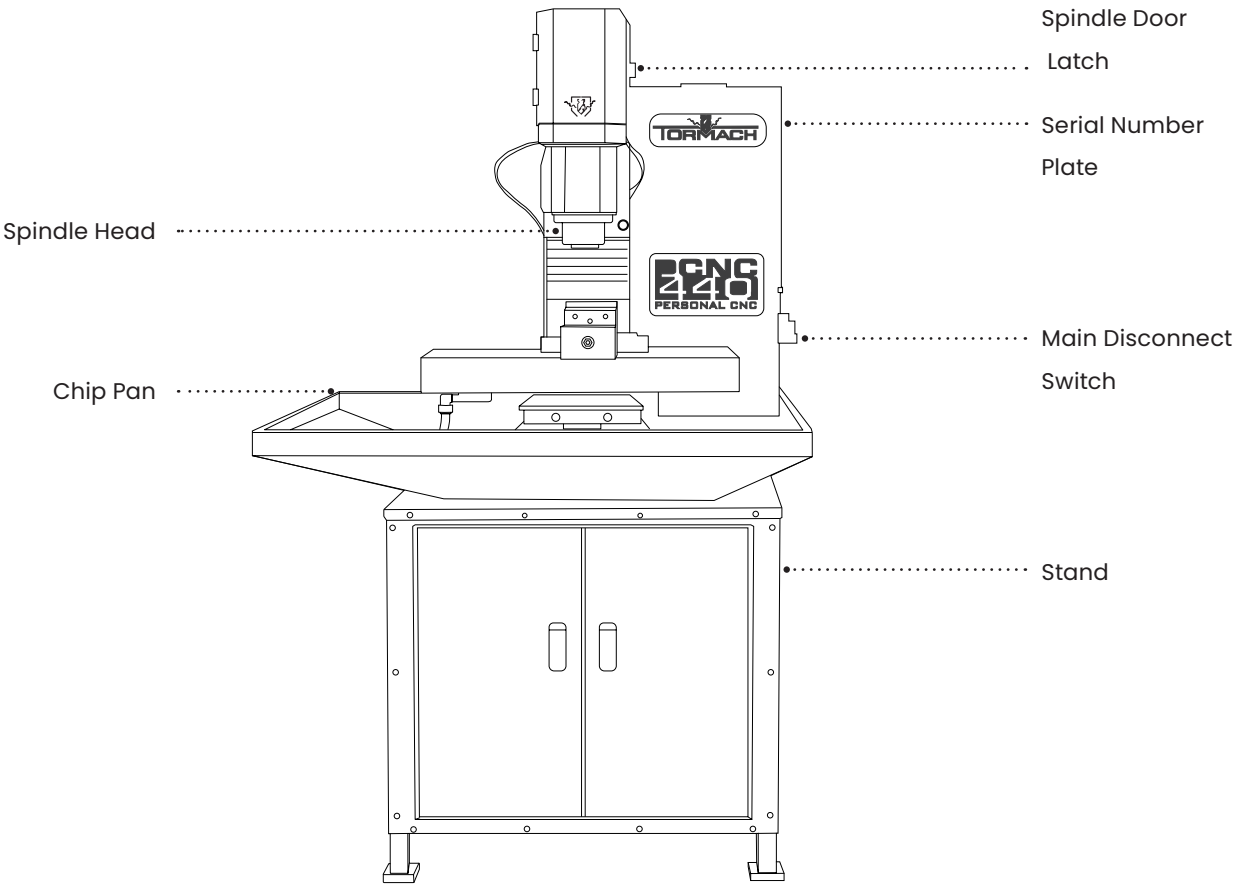


01:
The PCNC Mill

The following manual presents foundational information on the PCNC (Personal CNC) Mill at the Design + Technology LAB.

Please read this manual before booking a PCNC Mill appointment with the Design + Technology LAB.

Questions and Concerns: If you have any trouble or questions regarding machine use please email the Staff Members at: Designtechnologylab@ryerson.ca.



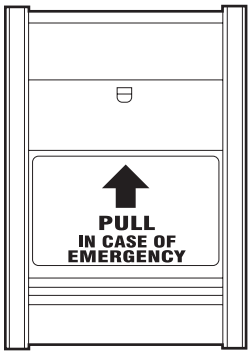
Tormach PCNC 440 Mill

D+TL

General Workshop Safety

Required Personal Protective Equipment (PPE)

Recommended Personal Protective



Emergency Pull Station

Workshop Safety Practices

1. No person will work alone in any areas of the machine room. Design + Technology LAB members working after hours alone must get permission from the lead Technician Member and notify Ryerson security.
2. Safety glasses must be worn while operating any power machinery/tools.
3. Distracting conversations and use of electronics (including cellphones and earbuds) can be dangerous in work areas and must not take place. Please take phone calls in the hall outside the lab.

Clothing and PPE

Before entering the Design + Technology LAB: Please ensure that you have read and adhere to the workshop general safety guidelines and are wearing all required PPE. The following rules concern appropriate Workshop apparel:

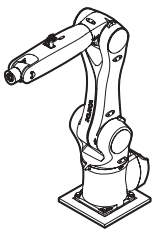
- Long sleeves must be rolled up.
- Loose scarves or clothing must be secured
- Heavy or loose jewelry must be removed (i.e., rings, watches, bracelets, necklaces, etc.)
- Long hair must be tied back.
- Non-slip, hard-toe, flat shoes must be worn.

What to do in the case of an accident.

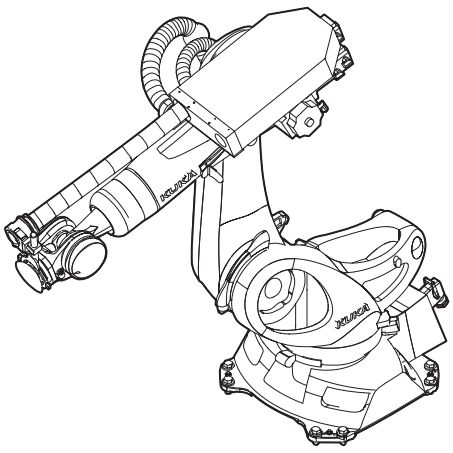
In the case of an accident notify a Staff Member immediately and pull the EPS (Emergency Pull Station) switch to notify security. If possible have the injured person sit down to prevent fainting, collapsing, or further injury.

D+TL

General Workshop Safety



KUKA KR10 Robotic Arm



KUKA KR150 Robotic Arm

Robotic Operation Safety

There are a number of robotic arms located in the Design + Technology LAB. There is 1 large Kuka KR150 in the High Bay (B33A) and 2 Kuka KR10's in the Workroom (B31). Be mindful of robotic operation at the Design + Technology LAB. Do not enter rooms when the following signage is posted.

This sign is most commonly posted on the entrance to the High Bay. Interlocks are installed on doors to the Kuka Cage and High Bay. Do not open doors during Robot use or it will trigger the workshop's Emergency Stop system. If you require access to the Print Lab when a robot in the High Bay is in operation, ask the Design + Technology LAB staff for assistance.

Noise and Air Quality

The Design + Technology LAB is a new facility with a strong interior ventilation system and various dust and fume collectors. Air quality in the facility can be affected by the machining of various materials. The dust and fume collectors work to reduce your risk of exposure to off-gassing materials and fine dusts. In special circumstances, we recommend you wear an N95 mask as an extra precaution.

The dust collectors and fume collectors in the metal and woodworking shop are noisy. Please use earplugs if the noise bothers you. The Design + Technology LAB has disposable ear plugs available.

03: PCNC Mill Safety

1: Pre - Operation Checklist

Are the power cords free of frays and damage?

Do you know where the e-stop feature is located (button/switch/cord)?

Are guards in place and in good working order?

Is the machine/equipment secure and level?

Is the area around the machine/ equipment free of slip/trip hazards?

Are flammable/combustible materials removed from the immediate work area?

Has the debris/material from previous operations been removed?

Are all tools/wrenches removed from the work area?

Is the machinery/equipment free of defects?

Are all your measurements and layouts complete before using machinery/ equipment?

2: Post - Operation Checklist

1. Turn the machine off. Do not walk away until it comes to a complete stop.

2. Clean up the machine and the surrounding area.

MANUFACTURER SAFETY [1]

General Safety

- Wear OHSA-approved safety glasses, safety shoes, and ear protection.
- Remove loose-fitting clothing, neckties, gloves, and jewelry.
- Tie up long hair or secure under hat.
- Never operate a machine after consuming alcohol or taking medication.
- Keep work area well lit and deploy additional lighting, if needed.

Operational Safety

- Understand CNC machines are automatically controlled and may start at any time.
- Do not leave machine unattended during operation.
- Always power off machine when not in use.
- Never operate with unbalanced tooling or spindle fixtures.
- Remove all tools (wrenches, chuck keys, etc.) from spindle and machine surface before starting operations; loose items can become dangerous projectiles.
- Use adequate work clamping; loose workpieces can become dangerous projectiles.

03: PCNC Mill Safety

MANUFACTURER SAFETY [1] (continued)

Operational Safety

- Protect your hands. Stop machine spindle and ensure machine motion has stopped before:

  1. Reaching into any part of the machine motion envelope
  2. Changing tools, parts or adjusting the workpiece
  3. Changing belt/pulley position
  4. Clearing away chips, oil or coolant; always use a chip scraper or brush.
  5. Making an adjustment to part, fixture, coolant nozzle or when taking measurements
  6. Removing protective shields or safeguards; never reach around a guard
  7. Keep work area clear of clutter as machine motion can occur when keys are accidentally pressed or objects fall on keyboard, resulting in unexpected motion.

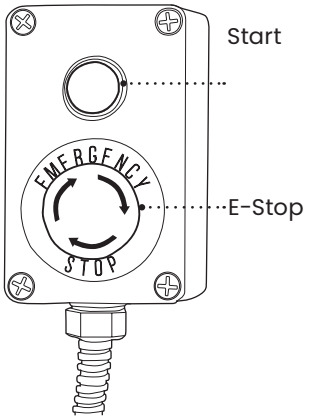
03: PCNC Mill Safety

MANUFACTURER SAFETY (continued)

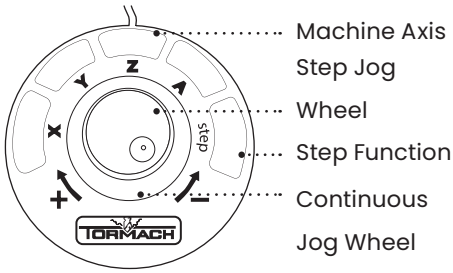
- Position clamping attachments clear of tool path. Be aware of workpiece cutoffs that could be cut free during operations and become dangerous projectiles.
- Always use proper feeds/speeds, as well as depth/width of cut to prevent tool breakage.
- Check for damaged tools/workpieces and cease operations if detected; replace before re-starting operations as these can become dangerous projectiles. Never use longer or larger tools than necessary.
- Chips and dust from certain materials (e.g., magnesium) can be flammable. Fine dust from normally non-flammable materials may be flammable or even explosive.
- Chips, dust, and vapors from certain materials can be toxic. Always check the Materials Safety Data Sheet (MSDS) for each material.

04: Machine Overview

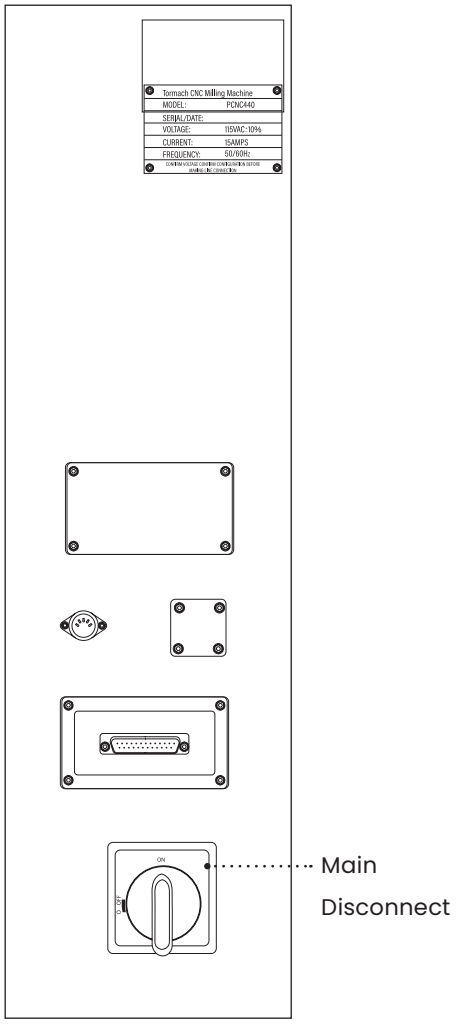
Machine Controls



Operator Box



Jog Shuttle Controller

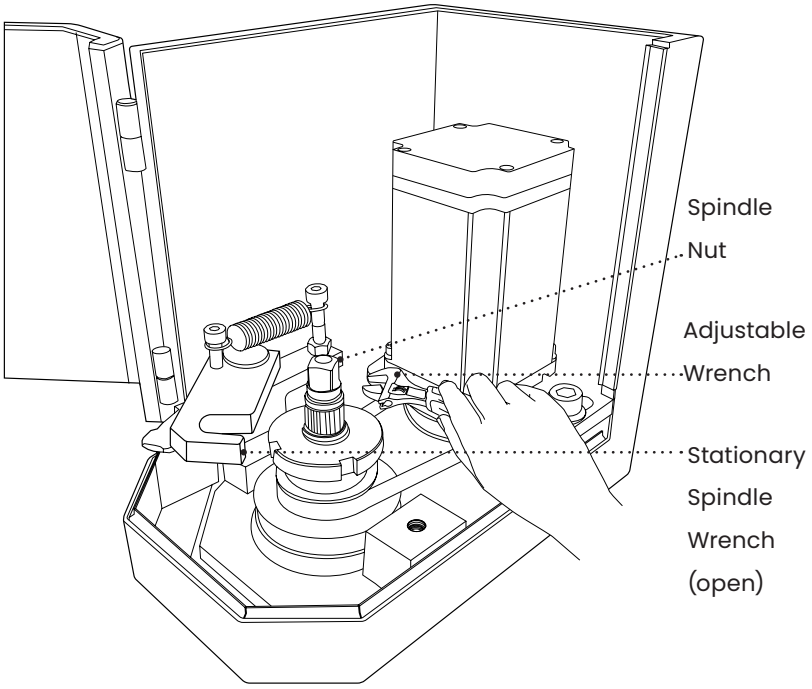


The Jog Shuttle Controller allows you to control the position of the X, Y and Z axes. You can choose between continuous jogging or stepping, in increments of 0.0001", 0.001", 0.01" or 0.1".

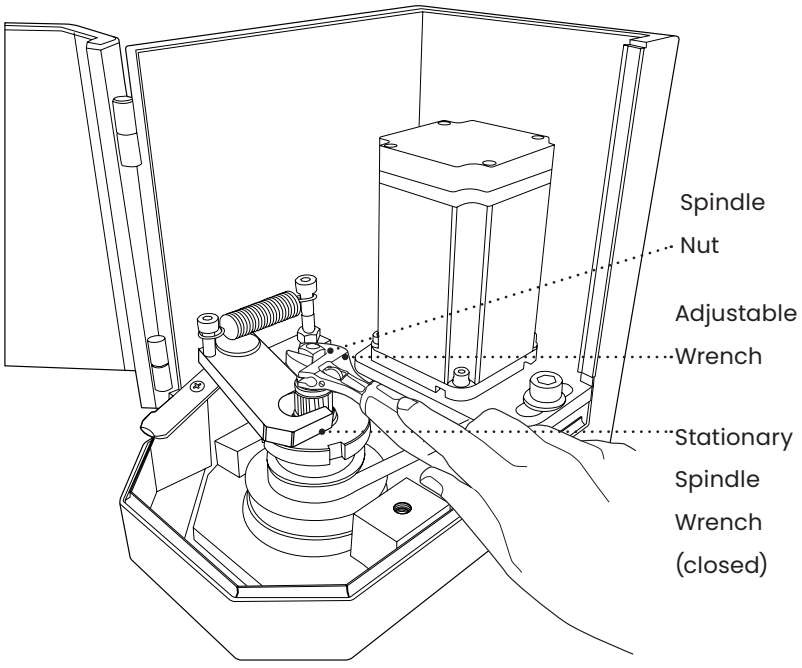
Jog Shuttle Control	Machine Movement
x- axis + positive jog wheel movement	Bed moves LEFT
y- axis + positive jog wheel movement	Bed moves FORWARD
z- axis + positive jog wheel movement	Spindle moves UP

04: Machine Overview

A Design + Technology LAB Staff member will perform tool changes, if necessary.



Open Top Spindle (Axonometric View)

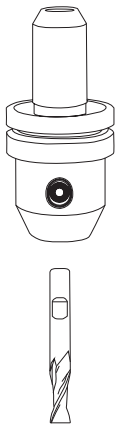


Open Top Spindle (Axonometric View)

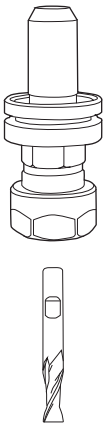
04: Machine Overview

TYPES OF TOOL-HOLDERS:

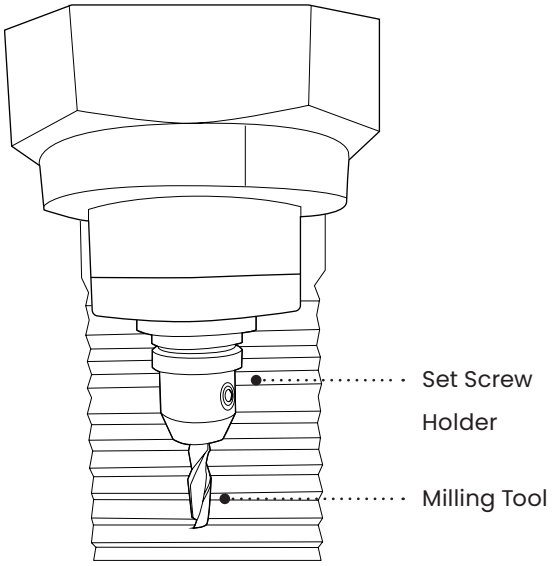
1. Set Screw Holder



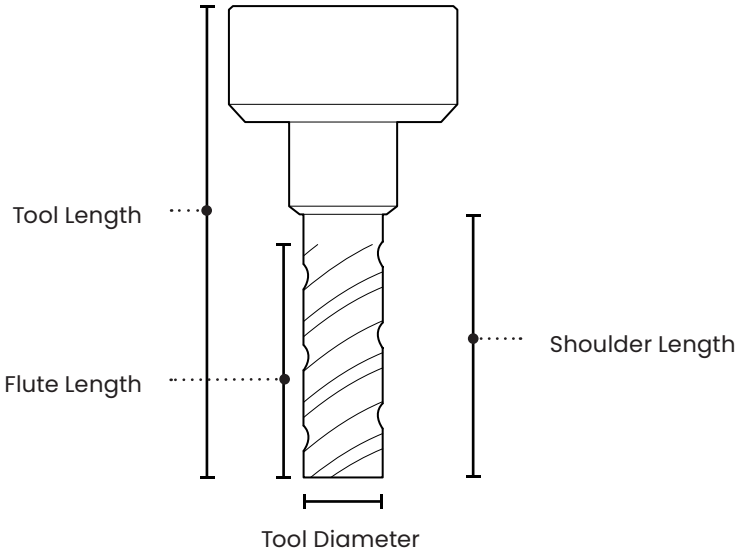
2. Collet Holder



Machine and Tool Components



Spindle with Set Screw Holder and Tool

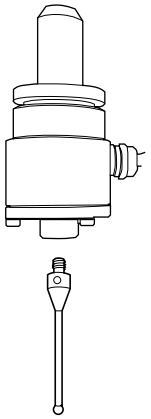


Milling Tool Details

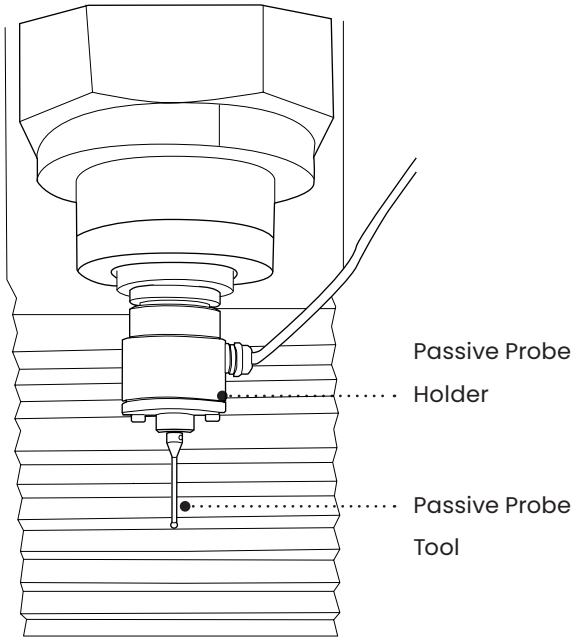
04: Machine Overview

TYPES OF TOOL-HOLDERS:

3. Passive Probe Holder



Machine and Tool Components



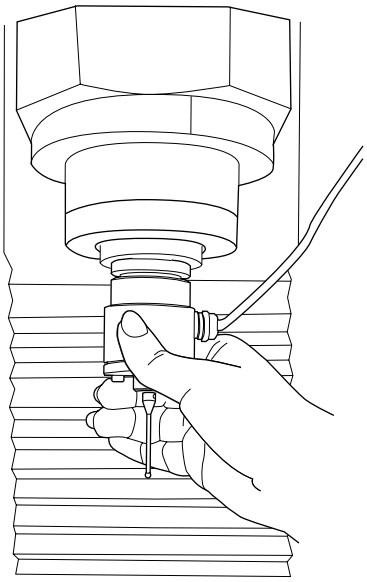
Spindle with Passive Probe Holder and Tool

05: Removing Mill Tools

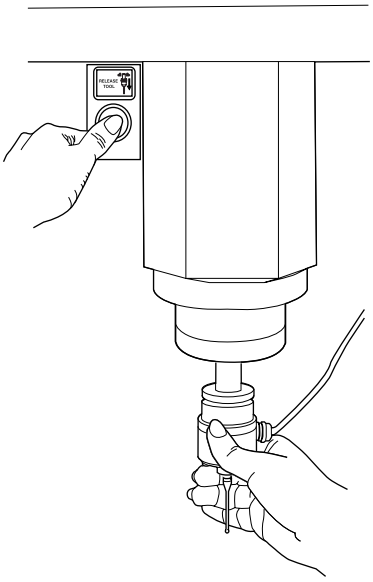
A Design + Technology LAB Staff Memberwill perform tool changes, if necessary.

1. Open the Top Spindle door latch to ensure the tool will not run.
2. Make sure you have a firm grip on the tool to prevent damage during tool release. Keep your hand towards the bottom of the tool to avoid the collet holder grip.
3. Press the black “RELEASE TOOL” button beside the Spindle and pull the tool downward.
4. Ensure the tool is fully released from the collet holder and spindle.

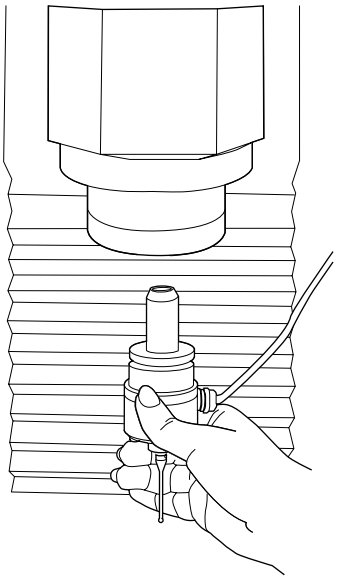
STEP 2



STEP 3

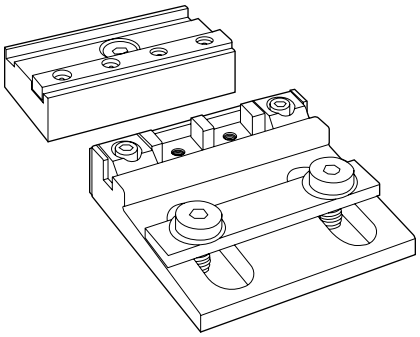


STEP 4

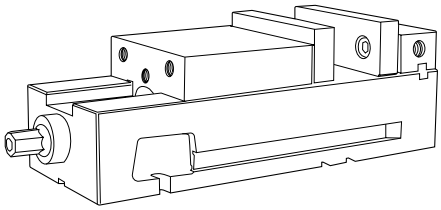


06: Milling Accessories

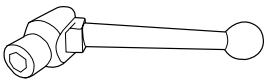
Below are diagrams of additional accessories you might need when preparing your mill job on the PCNC Mill.



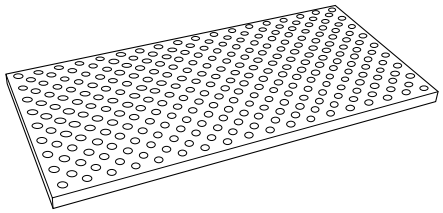
Two-part Vise



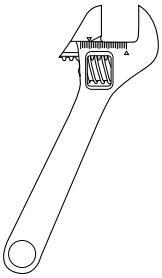
Vise



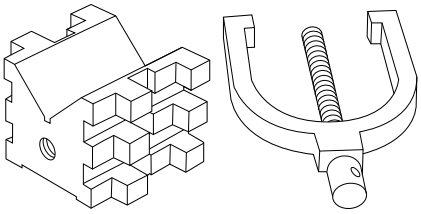
Vise Wrench



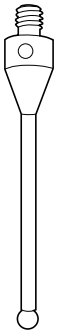
Fixture Plate



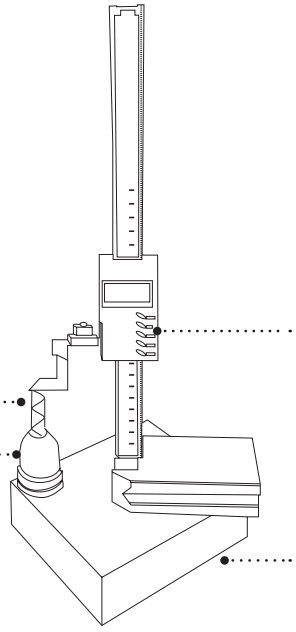
Adjustable Wrench



V-Block / clamp set



Passive Probe



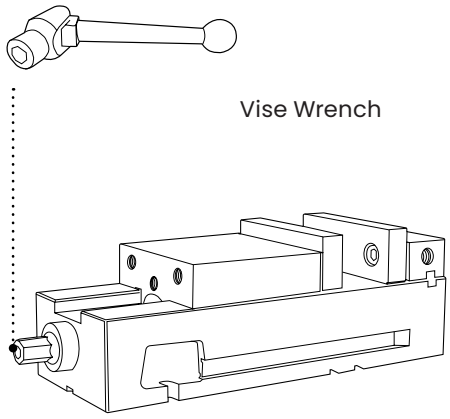
Digital Height Gauge

07: Workholding

Workholding (securing your workpiece) is one of the most challenging aspects of the milling process. There are several workholding accessories to aid with this challenge, but the best option is dependent on your stock size and the milling requirements of your project model.

Workholding accessories at your disposal include a 4" vise, fixture plates and a variety of clamps.

Vises: A machinist vise is the go-to method to secure workpieces, especially those with straight edges. They can be adapted for workpieces with rounded edges with the help of soft jaws which can be milled to perfectly support your workpiece.

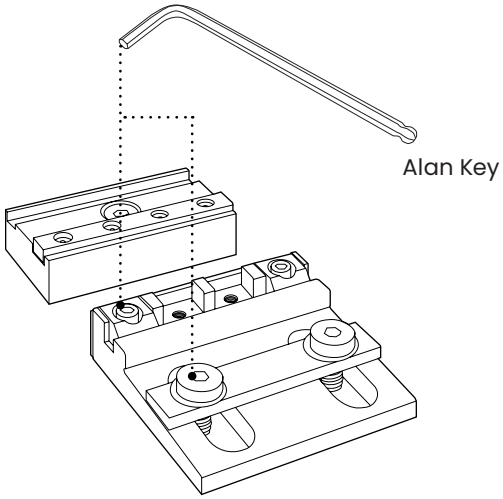


Vise Wrench

Vise

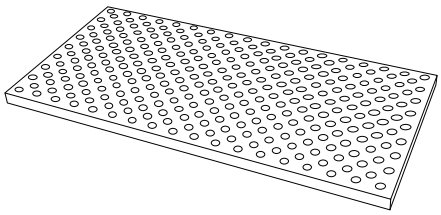
07: Workholding

Two-Part Vise: A machinist vise is the go-to method to secure workpieces, especially those with straight edges. The two-part vise can accommodate large areas on the fixture plate for workpieces at varying sizes with the help of soft jaws to perfectly support your workpiece.



Two-part Vise

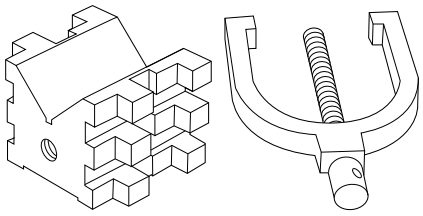
Fixture Plates: Aluminum fixture plates, also known as tooling plates, are a low-cost, disposable alternative to others on the market, and can be used to mount things quickly. These blank plates are easily customized to fit your machining needs to make sure that workholding is done consistently every time.



Fixture Plate

Clamps: The most basic way to secure a workpiece is to clamp it. This typically includes using T-nuts, studs, clamps, and flange nuts.

07: Workholding



V-Block / clamp set

V-Block: These blocks have a 'V' shaped channel and a clamp to secure cylindrical stock material horizontally or vertically. Stock with a square profile may also be held to mill along an edge.

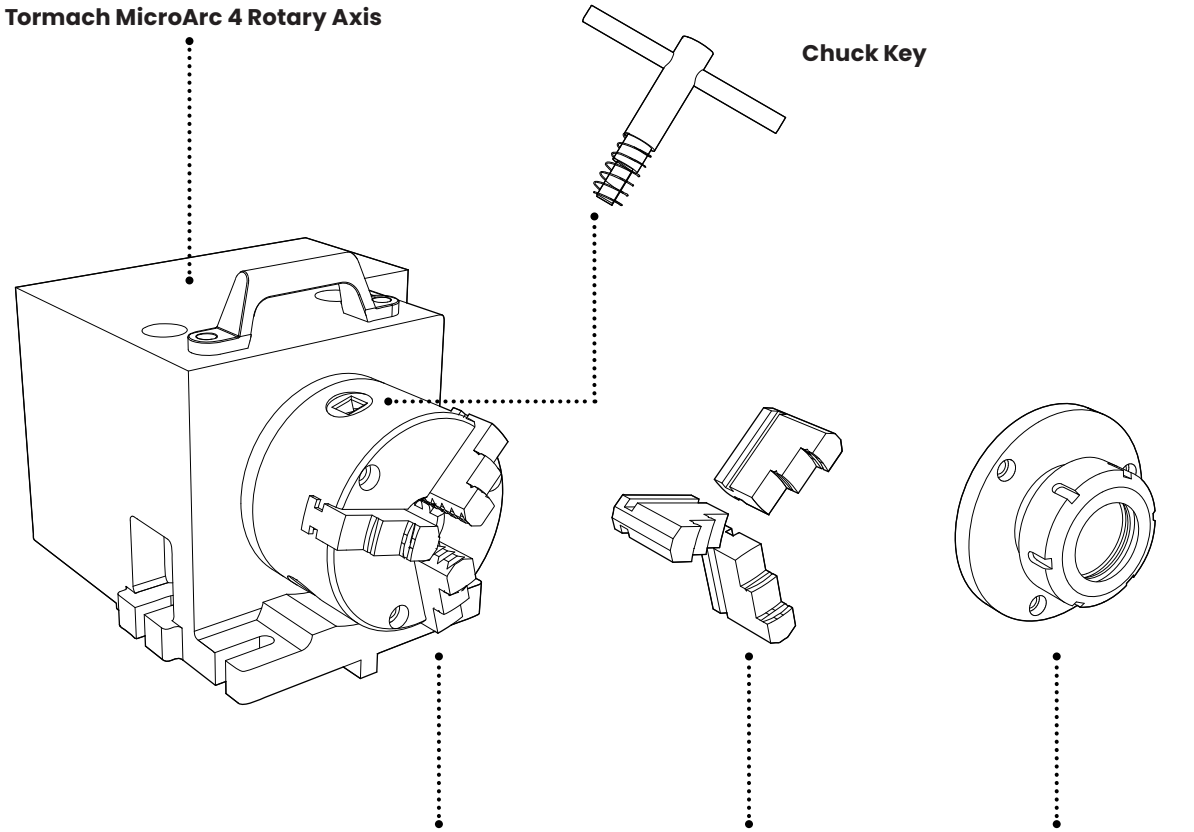
07: Workholding

MicroArc Rotary Axis: There are two types of mounting options for the rotary axis:

1) 100mm 3 jaw chuck with standard and reversed jaws

2) ER 40 Collet Face Plate

Depending on the size and shape of your stock material, you may need to mill a mortise or tenon into it prior to mounting it on the rotary axis. This will be explained further in chapter *11 The PCNC Process: Step 3.*

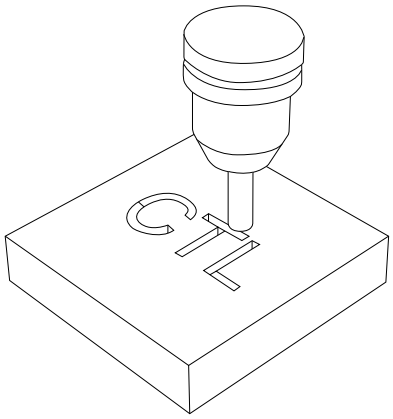


08: Milling Operations

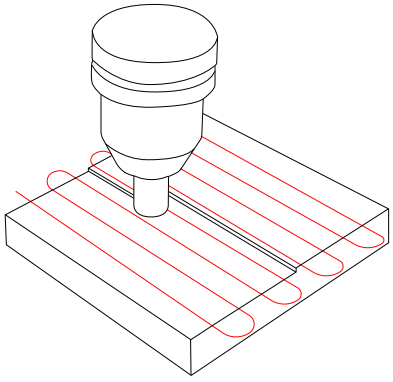
PCNC milling operations can be broken down into two subcategories: **2D milling operations** and **3D milling operations.** When using the PCNC Mill, remember you are able to prepare both types of operations in a single 3D Machining Model.

2D Milling Operations

Engraving: The Engraving Toolpath operation is designed to move along the model contours employing v-shaped chamfer tools. The profile contours can be selected by choosing the edges, sketches and solid faces, while the tool tip can be used to create sharp edges along the cavity corners.

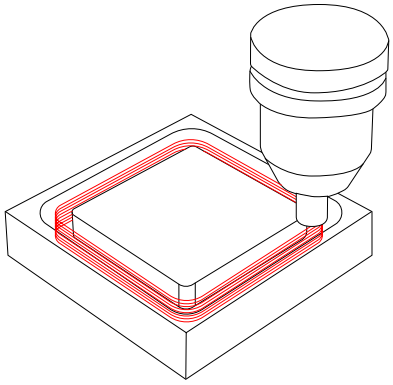


Facing: The Facing Toolpath operation is designed for top layer resurfacing of your stock. This process flattens your stock and prepares it for additional milling.

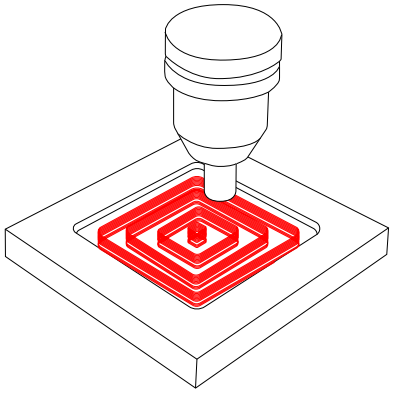


08: Milling Operations

2D Contouring: The 2D Contouring Toolpath operation is designed for machining profiles. Similar to 2D Adaptive Clearing and 2D Pocketing, the machining area for 2D Contouring can be selected by choosing the edges, sketches and solid faces. Although 2D Contouring is typically a finishing operation, it can be used to create multiple cuts.



2D Pocketing: The 2D Pocket Toolpath operation is designed the rough removal of material, in addition to finishing passes. Similar to 2D Adaptive Clearing, this tooling process can allow you to clear material in cavities, open pockets etc., The toolpaths for 2D Pocketing are offsets of your machining boundary, and can be selected by choosing edges, sketches or solid faces. This operation has additional options for milling tapered walls and pre-drilling entry positions.



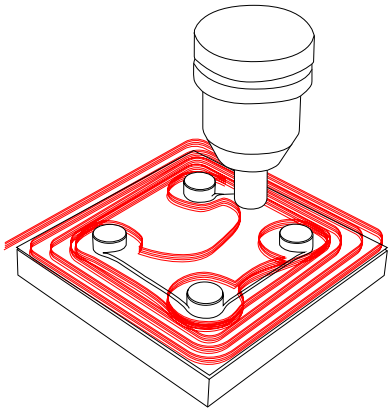
08: Milling Operations

2D Milling Operations

3D Milling Operations

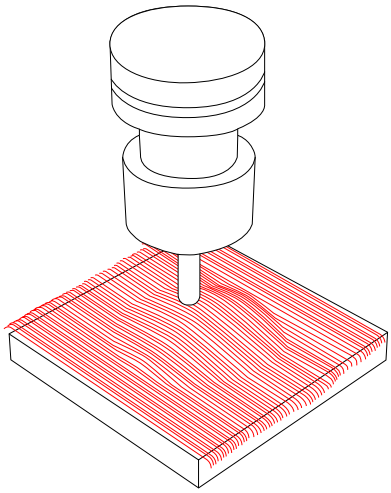
4-Axis Milling Operations

2D Adaptive Clearing: The 2D Adaptive Clearing operation is designed for material removal. This process can allow you to clear material in cavities, open pockets etc., and can eliminate the need for conventional cutting and sudden directional changes. Areas for this machine process can be selected by choosing edges, sketches or solid faces.

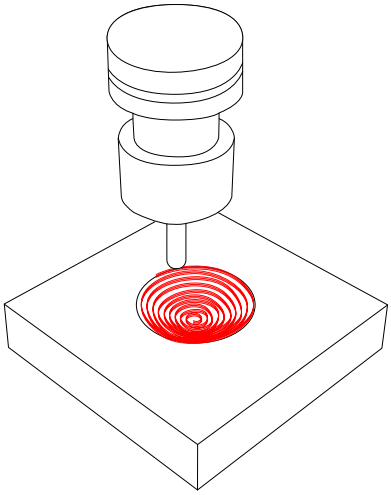


08: Milling Operations

Parallel Finishing: The Parallel Finishing Toolpath operation is designed to create equally spaced finishing paths, with the ability to control the angle of each row.

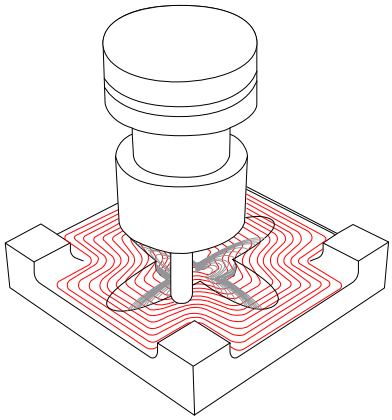


Spiral Finishing: The Spiral Finishing Toolpath operation is designed to create spiral paths to the base surface of your model. This process is especially useful with round, flat surfaces.

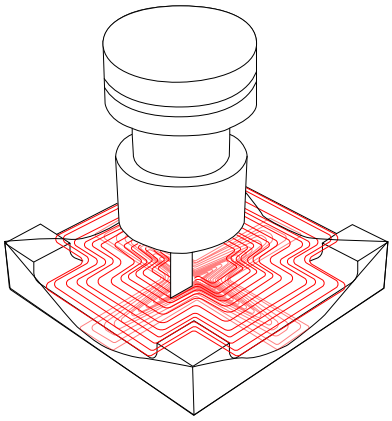


08: Milling Operations

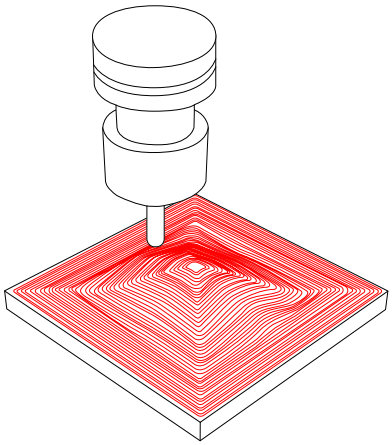
Pocket Clearing: The Pocket Clearing Toolpath operation is designed for the removal of material from large areas.



Adaptive Clearing: The Adaptive Clearing Toolpath operation is a high speed machining operation optimized for material removal.

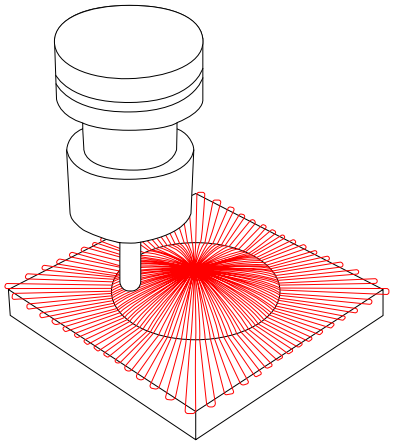


Morphed Spiral: The Morphed Spiral Toolpath Operation is designed for milling organic models.



08: Milling Operations

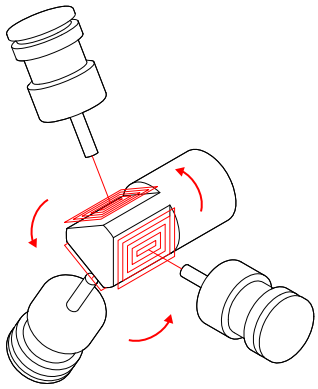
Radial Finishing: The Radial Finishing Toolpath operation is designed to create radial paths along a selected arc. These radial lines are projected onto the surface of your model.



08: Milling Operations

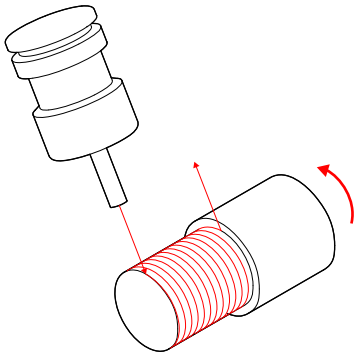
4-Axis Milling Operations

All 2D and 3D milling operations can be done in conjunction with the rotary axis. This means that the PCNC can perform a milling operation on one face, rotate the stock, and then perform another operation on another face.



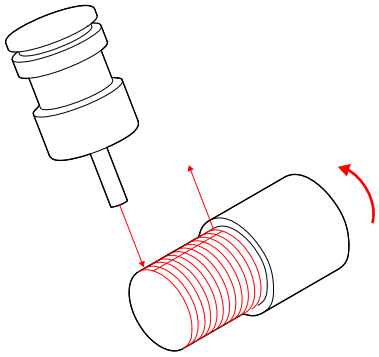
The 4th axis allows for simultaneous rotational milling. This means that the stock can rotate around the x-axis while the spindle moves up and down along the z-axis. There are three types of simultaneous rotational milling operations: Spiral, Circular, and Linear.

Spiral: This operation performs one continuous cut along the length of your model.

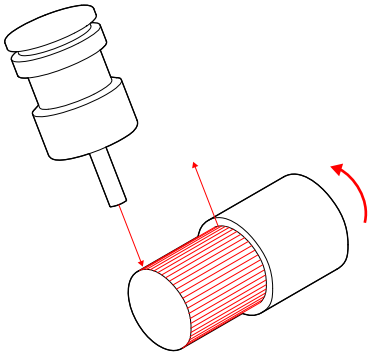


08: Milling Operations

Circular: The stock makes a full rotation with the tool set at a fixed point to machine a circle. It then steps over once to machine the next circle



Linear: The stock traverses along the x-axis allowing the tool to mill along the length of your model. It then rotates one step over to repeat the operation.



Note that these are finishing operations which mean that you will still need to remove as much material as possible with an adaptive clearing pass on ALL faces of your stock before performing a Spiral, Circular or, Linear 4-axis operation.

09: Materials

The PCNC Mill is capable of milling many different materials. These materials, referred to as stock, include; wood, unhardened mild or alloy steels, aluminum, and plastics. Talk to a Staff Member if you wish to mill a material that falls outside this list:

- Unhardened mild or alloy steels
- Aluminum
- Plastics
- Wood

10: Coordinates and the PCNC Mill

The PCNC Mill is a physical representation of the virtual coordinate grid of your digital models.

The **X-Axis** runs left and right on the PCNC Mill.

The **Y-Axis** runs forward and backward on the PCNC Mill.

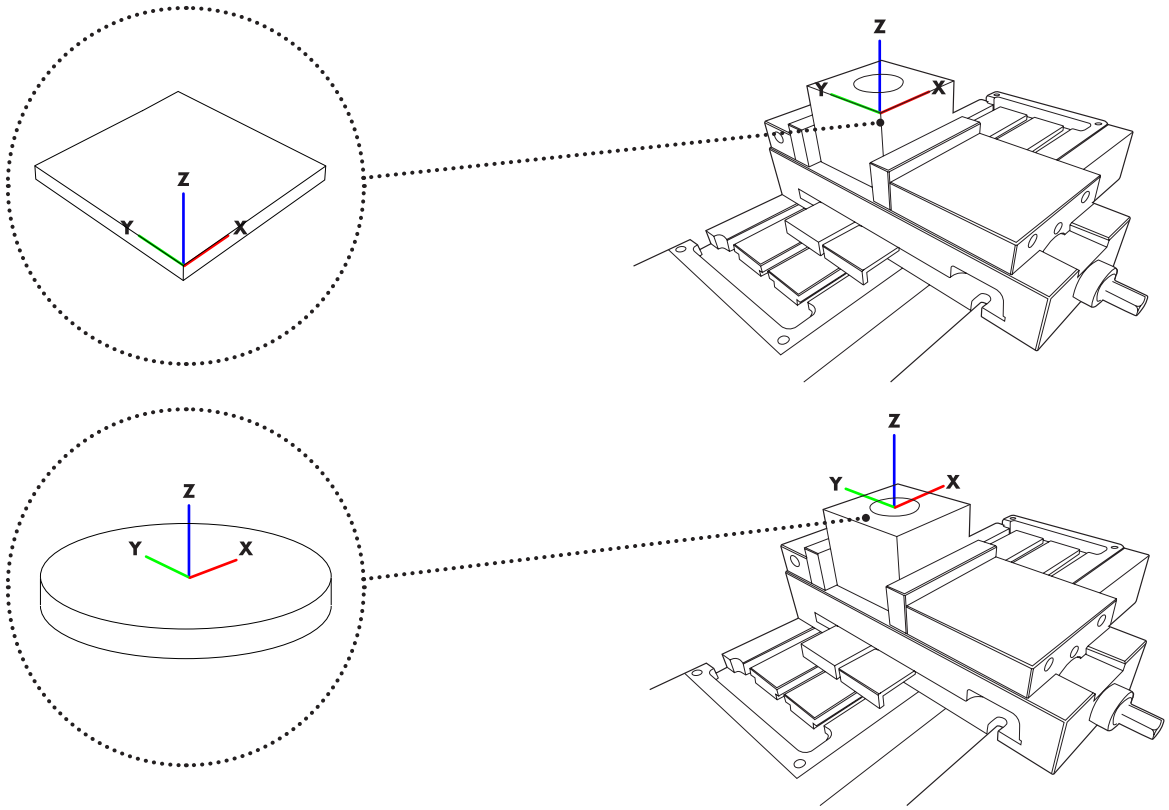
The **Z-Axis** runs 'vertical' from the PCNC Mill.

The Origin point in the digital space (software) and the physical space (the mill) is the intersection of all 3 axes (X, Y, Z).

It is imperative that the origin point and orientation of your 3D machining model coincides with the location of your material on the physical PCNC mill. Depending on the shape and the orientation of your stock, you will need to pay particular attention to the location and direction of the origin.

10: Coordinates and the PCNC Mill

For 2D and 3D milling operations on a **rectangular stock** set the origin point at any corner with the Z+ axis oriented vertically upward. Be sure that your toolpaths are located within your material. For **circular stock** set the origin at it's center with the Z+ axis oriented vertically upward.



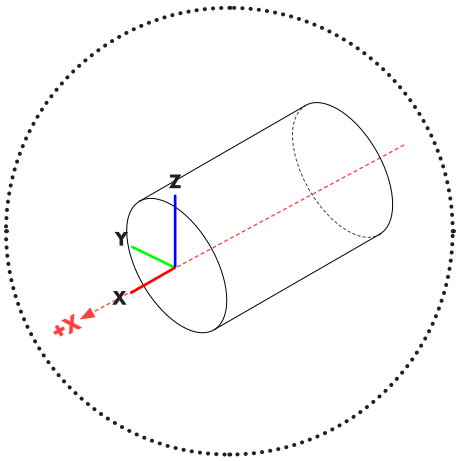
3D Project & Machining Model: FUSION 360.

Corresponding origin (0,0,0) point probed on the PCNC440 Mill using PathPilot.

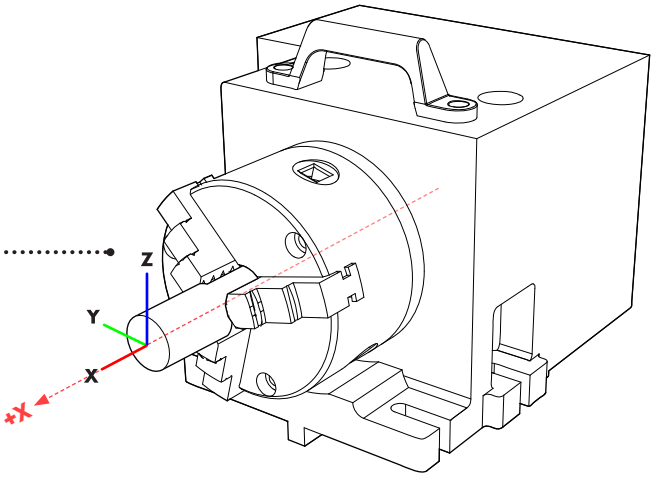
10: Coordinates and the PCNC Mill

For **4-axis milling operations,** set the origin point at the **center** of your stock. Keep in mind that the **X+ axis** should be oriented along the length of your stock as well as the X+ axis of the MicroArc 4 rotary axis. The orientation of the Z+ axis will always run perpendicular to the face(s) of your stock.

The origin on the PCNC mill is set during the probing process. It should correspond to where it is set in your 3D project and machining model. This will be explained in further detail in *Step 3* of *Mill Your Project.*



3D Project & Machining Model: FUSION 360



Corresponding origin (0,0,0) point probed on the PCNC440 Mill using PathPilot. NOTE: the MicroArc 4 rotary axis is set up on the Design + Technology LAB Tormach PCNC440 mill so that the X+ axis runs along the center of the chuck and points AWAY from the rotor.

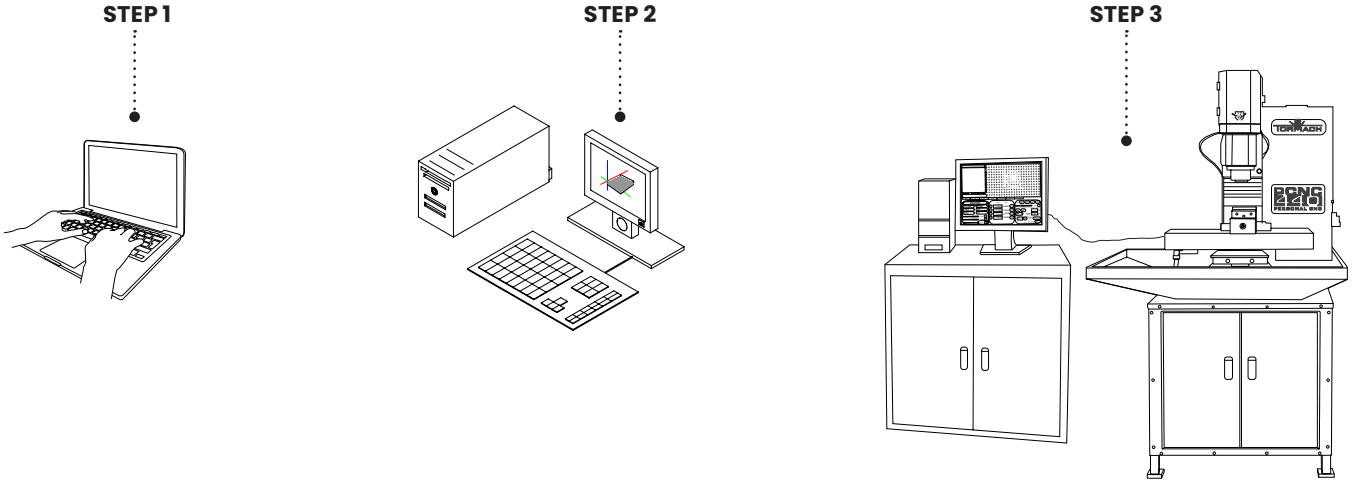
11: The PCNC Process

The PCNC Mill is a digitally driven, coordinate CNC machine. The CNC (Computer Numerically Controlled) process for the PCNC Mill is as followed:

STEP 1: Create your 3D Project model. Model your part in a 3D modeling software (Rhino, Fusion 360, etc).

STEP 2: Develop your 3D Machining model. Import your project model into Fusion360 and program your toolpaths in MANUFACTURE mode. Export (post) the 3D Machining model to create a g-code file. Remember to select the Tormach PathPilot post. G-code is computer language that instructs the PCNC Mill.

STEP 3: Mill you project. Import the g-code file into Tormach's PathPilot interface via USB. A Staff Member will assist you with machine calibration, workpiece setup, and final milling adjustments within PathPilot's Persistent Controls menu (bottom half of the screen), before beginning the milling process.



Step 1:

Create your 3D Project Model

Now that you have developed a foundational understanding of the PCNC Mill and its functions, you are ready to create your digital files. As mentioned earlier, **you will need two 3D digital files** for your PCNC milling appointment; a 3D Project model and a 3D Machining model.

Project model vs. Machining model:

- **Project model:** a 3D digital model of the part you wish to mill on the PCNC. You can use any 3D modeling software such as Rhino, Fusion 360 , etc.) to create your project model. This will serve as a reference for the Staff Member, so they can understand what you wish to achieve from your PCNC appointment.
- **Machining model:** a 3D digital model that is used to create tool paths and generate g-code for the PCNC. Fusion 360 is the primary software for this at the Design + Technology LAB. You will develop your machining model within the Manufacture mode in Fusion360 prior your appointment. This will be reviewed by a Staff Member during your appointment.

Step 1:

Create your 3D Project Model

It is important to design your project within the size limitations of the PCNC Mill. The following size limitations are the MAXIMUM extents in which your project can be milled safely. These dimensions will ensure that the tool does not collide with the machine, trigger the machine's limit switches, or cause too much *chatter* (vibration). Keep in mind that the overall size of your project will also be determined by the dimensions of your workpiece (or stock) material.

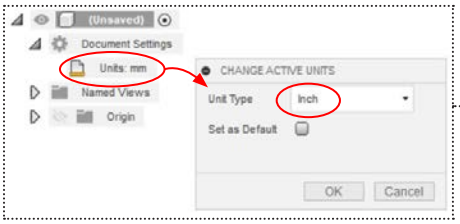
- **3-axis milling projects:** the Tormach PCNC 440 has a maximum travel distance of 10"x 6.25"x 10" (x,y,z); Therefore, design within a 8"x 4"x 8" (203 x 101 x 203 mm) boundary.
- **4-axis milling projects:** the MicroArc 4 rotary axis has maximum holding diameter of 4" (100mm). Design within a 4"dia. x 6" length (100mm dia.. x 150mm length) boundary.

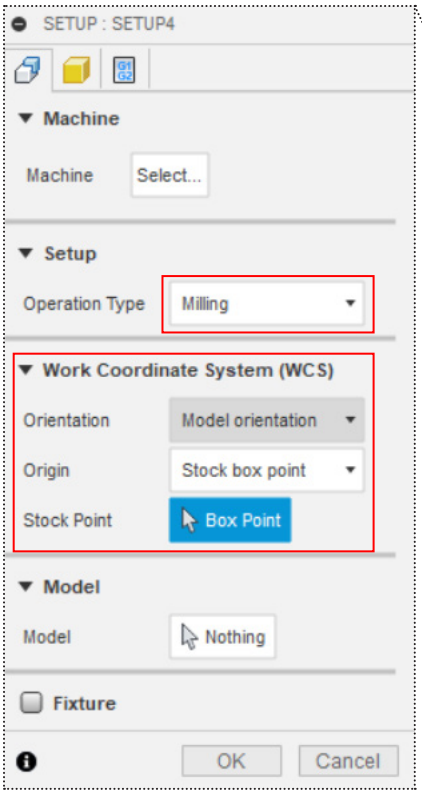
Step 2:

Develop your 3D Machining Model

The following section outlines the procedural order for preparing your 3D machining model file in Fusion360. This model will be developed by you and reviewed by a Staff Member prior to milling.

After creating your project model file in a 3D modeling software:



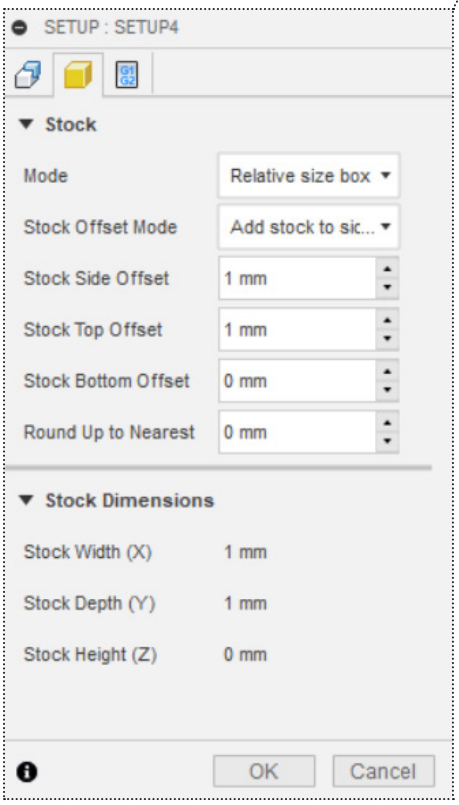


1. Upload the model into Fusion 360.
2. Set the units to **inches**. This can be done while in either *Design* or *Manufacture* mode.
3. Check X, Y, Z model orientation. On occasion, the Y-axis faces up in Fusion360. To change the orientation of the XYZ plane click your user icon at the top right of the window and open:

   Preferences > *Default modeling orientation* > *Z up*
4. To begin creating tool paths from your project model, go into **Manufacture** mode to develop your machining model. Select *New Setup* in the Setup drop-down. This will open a pop-up window through which the overall parameters of the machining setup can be determined.
5. The **Setup** tab in this is where the location and orientation of the origin point is determined. Select where to locate and orient the origin under *Work Coordinate System (WCS)*- default settings are shown to the left. Be sure that *Operation Type* is set to *Milling.*

Step 2:

Develop your 3D Machining Model



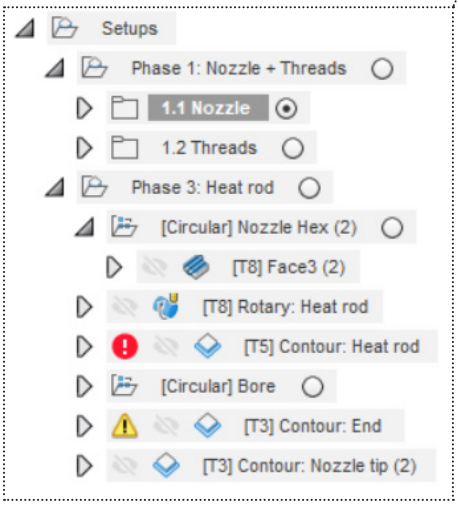
6. The **Stock** tab is where the parameters of your workpiece material is determined. Under the *Mode* drop-down menu, you may select from several options: fixed or relative sized box, cylinder, or tube. If you have modeled a custom stock material, select *From solid* in the drop-down menu and input your desired geometry.

7. Select the appropriate milling operation for your job under the **Milling** tab within the Manufacture interface. All 2D, 3D, and 4-axis milling operations previously mentioned in chapter *08 Milling Operations* are available by default with the exception of some **Multi-Axis** operations. *Autodesk* may require you to manually enable access to these plugins. Clicking on any one of these process will open a pop-up window where you will input detailed information about tools, feed & speeds, model geometry, heights & clearances, passes & depths, and more. This step is very important as it will dictate tool paths, milling time and the finish quality of your mill job.

8. The Design + Technology LAB will provide access to the latest **tool library** available. If you require a specific tool that isn't in the existing tool library, consult a Design + Technology LAB Staff well in advance to ensure the tool can be shipped, installed, and calibrated for your scheduled milling time.

Step 2:

Develop your 3D Machining Model



9. Your milling process will appear under **Setups**. The tool number and any error warnings will be displayed beside every operation. It is important to consider the order in which each operation appears - each will be milled first to last from top to bottom. Any rough, shallow, and/or internal passes should appear first prior to any finished, deep, and/or external passes. Note: depending on the complexity of your job, consider naming and group your operations to keep the file organized. (Refer to the corresponding example)

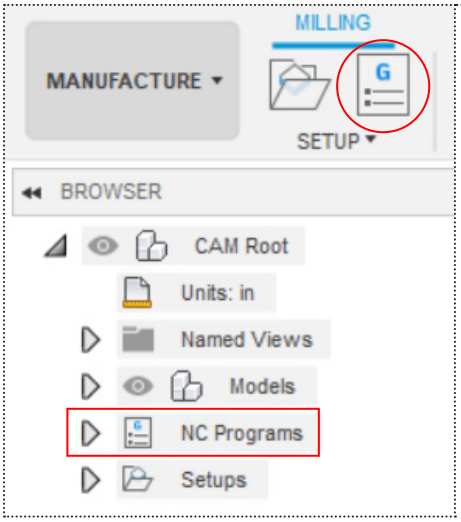
10. Simulate your milling operations and adjust the settings as needed.

11. Once you have completed the steps outlined above, ask a Staff Member to review your work.

Step 2:

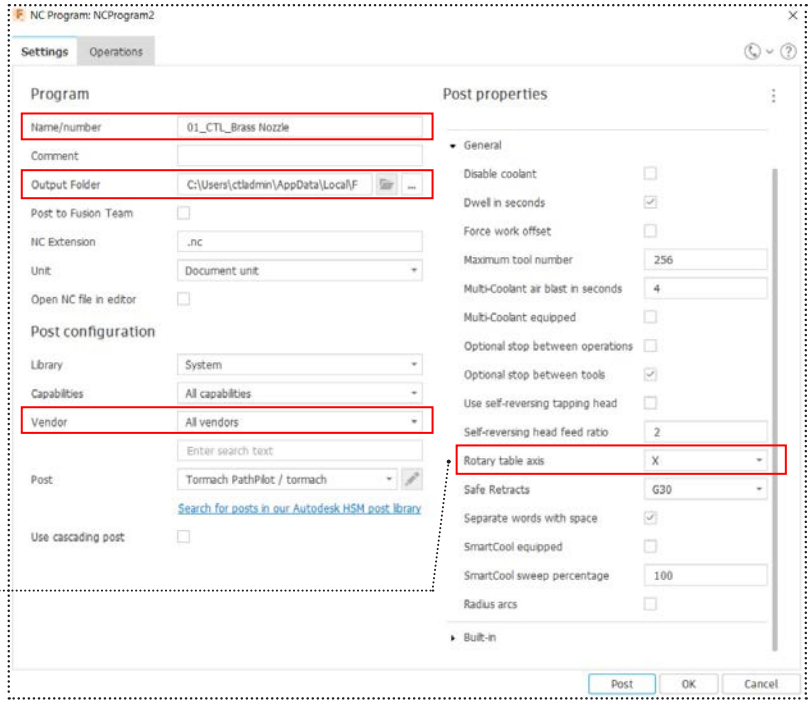
When you and a LAB Staff are satisfied with your machining model, post your file as g-code.

Develop your 3D Machining Model



1) Create an **NC program** of your setup. This will allow you to modify your post settings throughout the process before posting the final G-code. NC Programs will appear above Setups in the manufacture interface.

2) Double click your NC Program to modify the settings. Under the *Settings* tab be sure to name your G-code file and set the *Output Folder* to your USB. Be sure to select **Tormach PathPilot / Tormach** in the drop-down menu under *Post Configuration*. The *Operations* tab will allow you to include or omit any operation from the G-code sequence.



IMPORTANT: When milling with the MicroArc 4th axis, be sure to set the *Rotary table axis* to X

Step 2:

Develop your 3D Machining Model

When naming your G-code file, follow this naming system for ease of identification:

Your Initials_Project Name_Post Attempt #.nc

(ex. *DT_Lab Gear_Brass Nozzle 01.nc*)

The *Post Attempt Number* will make it easier to locate your file on the PathPilot interface should you need to re-post it more than once.

Save and Post your file after reviewing all the NC Program parameters above.

Step 3:

Mill your project

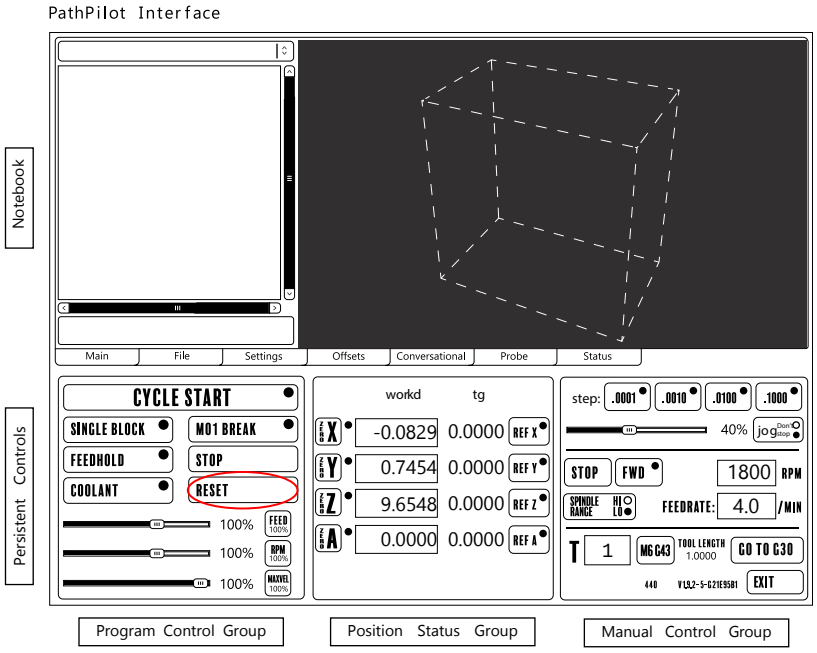
Import the g-code file into Tormach's PathPilot interface via USB. A Staff Member will assist you with tooling, calibration, workpiece setup, and final milling adjustments within PathPilot's Persistent Controls menu, before beginning the milling process.

POWERING ON PATHPILOT + PCNC MILL

Please execute the six steps listed below, in order:

1. Turn the PathPilot controller power strip ON.
2. Once the software has loaded, turn the Main Disconnect switch ON (located on the right side of the machine).
3. Turn the red Emergency Stop (E-stop) button clockwise to release.
4. Press the green button beside the E-Stop.
5. Click RESET on the screen.

PathPilot Interface



Step 3:

Mill your project

POWERING ON PATHPILOT + PCNC MILL

TORMACH PATHPILOT INTERFACE

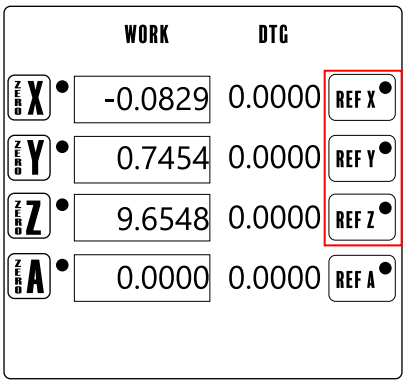
1. **Reference the Mill**
2. **Workpiece Preparation**
3. Tooling Preparation
4. Tool Offsets
5. Work Offsets
6. Import g-code file into PathPilot interface
7. Running the g-code file on the PCNC Mill

POWERING OFF PATHPILOT + PCNC MILL

TORMACH PATHPILOT INTERFACE

1. **Reference the Mill:** The X, Y and Z - axes should be referenced before operating the mill to establish the soft limits to protect the mill from over travel and to give meaning to the work offset values.

   Reference all three axes simultaneously by clicking the **Ref X, Ref Y and Ref Z** buttons. The LEDs on these buttons will turn green once they have been successfully referenced.



2. **Workpiece Preparation:** make sure your workpiece fits within the size limitations of the machine (refer to *Step 1*) and is held securely to the bed using any combination of the hold-down tools listed in Chapter *06: Milling Accessories.* When milling with the 4th axis rotor, make sure that the appropriate set of jaws are installed onto the chuck. Depending on the size and shape of your workpiece, you may create a mortise or tenon BEFORE mounting it onto the chuck. Be sure that your workpiece is at least 1.5" longer than the length of your design to account for a mortise or tenon. This means that your project model should be positioned 1.5" away from one end of the stock - this will prevent tools from colliding with the chuck jaws.

Step 3:

Mill your project

POWERING ON PATHPILOT + PCNC MILL

TORMACH PATHPILOT INTERFACE

1. Reference the Mill
2. Workpiece Preparation
3. **Tooling Preparation**
4. **Tool Offsets**
5. Work Offsets
6. Import g-code file into PathPilot interface
7. Running the g-code file on the PCNC Mill

POWERING OFF PATHPILOT + PCNC MILL

3. **Tooling Preparation:** Ensure that all the tools you require for your tooling operations are available and ready for use. You must consult a Staff Member to confirm that the tools you program in Fusion correspond to the existing tool library in PathPilot. Setting tool offsets in the PathPilot tool library will be explained in the following step.

4. **Setting Tool Length Offsets:** All the available tools are listed in the Tool Table, allowing you to refer to it and input the correct **Tool #** into the **Tool (T) DRO**.

   Please do not edit the Tool Table without consulting with a Staff Member.

   Using the **Digital Height Gauge** is the easiest method for determining tool lengths. This device can be used offline, and the lengths can be entered manually into the **Tool Table** in the **Offsets Tab**.

   If you have your own tools you would like to use on the PCNC Mill, talk to a Staff Member. The Staff Member will assist you in accurately measuring your tool lengths using the Digital Height Gauge, if they approve your request.

Tool	Description	Diameter	Length
1	2 flute endmill	0.5000	3.2250
2	engraver	0.1250	3.1466
3		0.0000	0.0000
4		0.0000	0.0000
5		0.0000	0.0000
6		0.0000	0.0000
7		0.0000	0.0000
8		0.0000	0.0000
9		0.0000	0.0000
10		0.0000	0.0000
11		0.0000	0.0000

PathPilot Tool Table

Step 3:

Mill your project

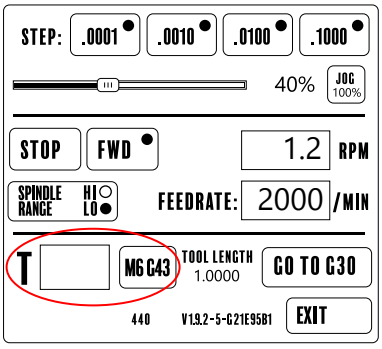
POWERING ON PATHPILOT + PCNC MILL

TORMACH PATHPILOT INTERFACE

1. Reference the Mill
2. Workpiece Preparation
3. Tooling Preparation
4. **Tool Offsets**
5. **Work Offsets**
6. Import g-code file into PathPilot interface
7. Running the g-code file on the PCNC Mill

POWERING OFF PATHPILOT + PCNC MILL

Enter the Tool # of the tool of your preference and press the **M6 G43** button.



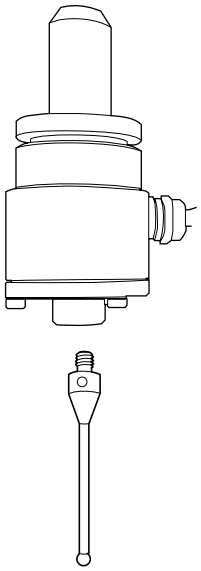
PathPilot Interface

5. **Work Offsets**

Work Offsets allow the operator to think in terms of the X, Y, Z - coordinates with respect to the part instead of thinking of them with respect to the mill position.

Setting the X, Y Workpiece Coordinates

1) If there is a tool in the spindle, carefully remove it along with the collet and replace it with the **Passive Probe** tool and its collet.



Passive Probe Tool

Step 3:

Mill your project

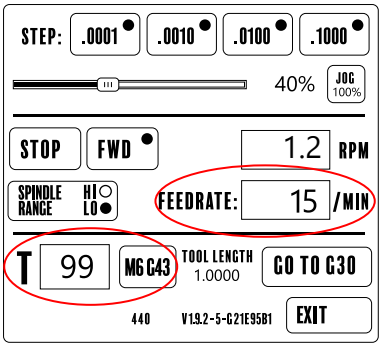
POWERING ON PATHPILOT + PCNC MILL

TORMACH PATHPILOT INTERFACE

1. Reference the Mill
2. Workpiece Preparation
3. Tooling Preparation
4. Tool Offsets
5. **Work Offsets**
6. Import g-code file into PathPilot interface
7. Running the g-code file on the PCNC Mill

POWERING OFF PATHPILOT + PCNC MILL

2) Type 99 into the **Tool (T) DRO** and press the **M6 G43** button. Tool 99 represents the Passive Probe while the M6 G43 button tells the PathPilot operating system that the tool has been changed and that the probe will read material coordinates.



PathPilot Interface

3) Open the Spindle Door to ensure the tool does not spin during operation.

4) Lower the **Feed Rate** to ensure the tool moves at a slow and steady rate to avoid tool damage.

Step 3:

Mill your project

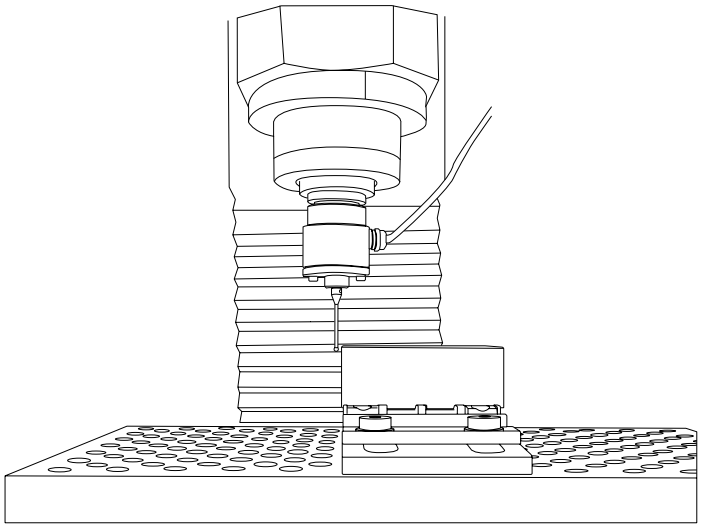
POWERING ON PATHPILOT + PCNC MILL

TORMACH PATHPILOT INTERFACE

1. Reference the Mill
2. Workpiece Preparation
3. Tooling Preparation
4. Tool Offsets
5. **Work Offsets**
6. Import g-code file into PathPilot interface
7. Running the g-code file on the PCNC Mill

POWERING OFF PATHPILOT + PCNC MILL

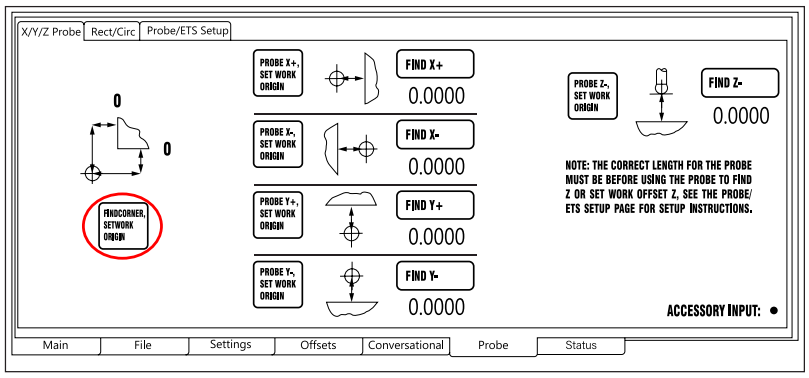
5) **Setting XY Workpiece Coordinates:** Carefully jog the Probe to the corner of your work piece using the Jog Shuttle. The Probe tip must sit within 1" away from the workpiece and just below the surface in order to read the material.



Passive Probe Tool

6) Once the Probe is in position, you may proceed to locate the origin (X0Y0Z0) either at the corner or center of the workpiece. This sequence is suitable for 3-axis milling procedures.

Findcorner Setwork Origin will set the XY workpiece coordinates. The spindle will automatically probe to each adjacent edge to set X0, Y0 at the corner of the workpiece.



PathPilot Interface

Step 3:

Mill your project

POWERING ON PATHPILOT + PCNC MILL

TORMACH PATHPILOT INTERFACE

1. Reference the Mill
2. Workpiece Preparation
3. Tooling Preparation
4. Tool Offsets
5. **Work Offsets**
6. Import g-code file into PathPilot interface
7. Running the g-code file on the PCNC Mill

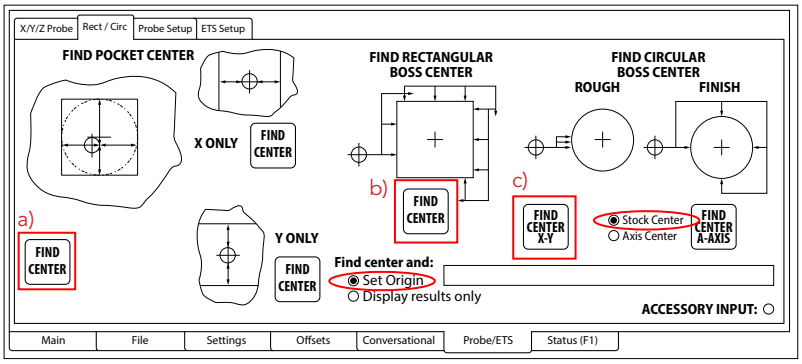
POWERING OFF PATHPILOT + PCNC MILL

There are three **Find Center** commands that will set X0Y0 coordinates at the center of a circular or square workpiece or pocket. Be sure that **Stock Center** and **Set Origin** is selected below.

a) To **Find Pocket Center**, jog the probe to any point within the pocket that is slightly below the top surface of the workpiece then click Find Center. The probe will jog to four points at the pocket edge to calculate its center.

b) To **Find Rectangular Boss Center**, jog the probe within 1" to the LEFT of the workpiece below its top surface then click Find Center. The probe will jog to four edges of the workpiece to calculate its center.

c) To **Find Circular Boss Center**, jog the probe within 1" to the LEFT of the workpiece below its top surface then click FIND CENTER X-Y. The probe will jog to multiple points along the circumference of the workpiece to calculate its center.



Step 3:

Mill your project

POWERING ON PATHPILOT + PCNC MILL

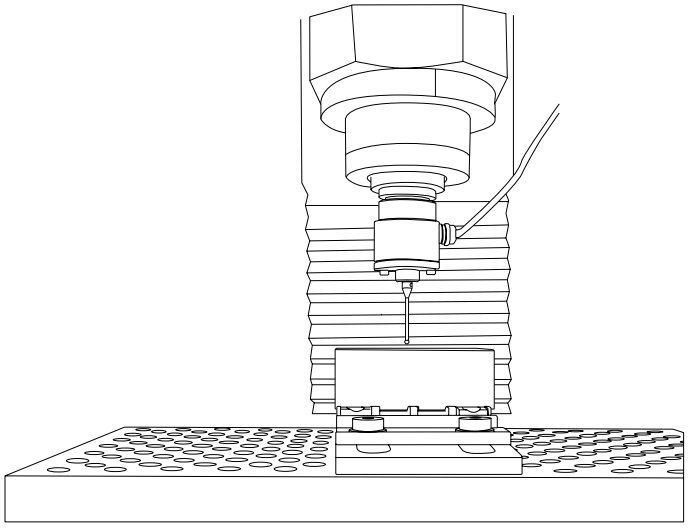
TORMACH PATHPILOT INTERFACE

1. Reference the Mill
2. Workpiece Preparation
3. Tooling Preparation
4. Tool Offsets
5. **Work Offsets**
6. Import g-code file into PathPilot interface
7. Running the g-code file on the PCNC Mill

POWERING OFF PATHPILOT + PCNC MILL

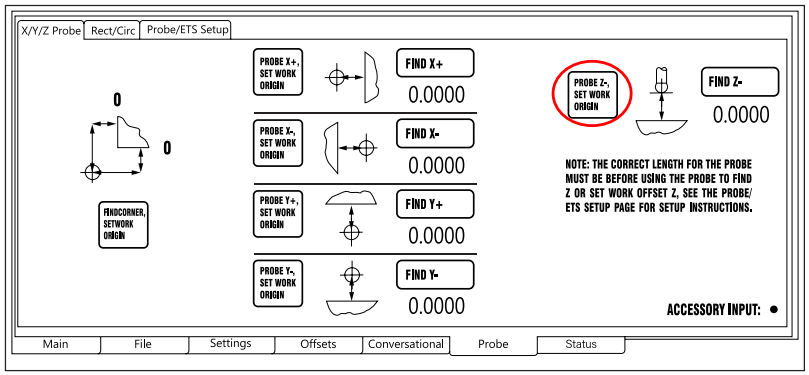
Setting Z Workpiece Coordinates

7. Carefully jog the Probe above your work piece using the Jog Shuttle. The Probe tip must sit within 1" above the workpiece to read the material.



Passive Probe Tool

8. Once the Probe is in position, click the **Probe Z-, Set work Origin** button in order to set the Z workpiece coordinates. The Probe will automatically plunge and set Z0 at the top of the workpiece. The location of the origin point has been set.



PathPilot Interface

Step 3:

Mill your project

POWERING ON PATHPILOT + PCNC MILL

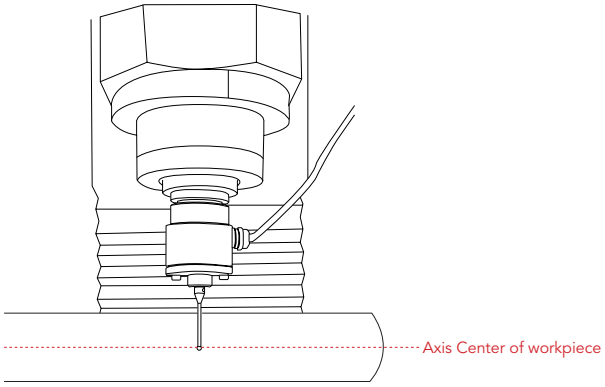
TORMACH PATHPILOT INTERFACE

1. Reference the Mill
2. Workpiece Preparation
3. Tooling Preparation
4. Tool Offsets
5. **Work Offsets**
6. Import g-code file into PathPilot interface
7. Running the g-code file on the PCNC Mill

POWERING OFF PATHPILOT + PCNC MILL

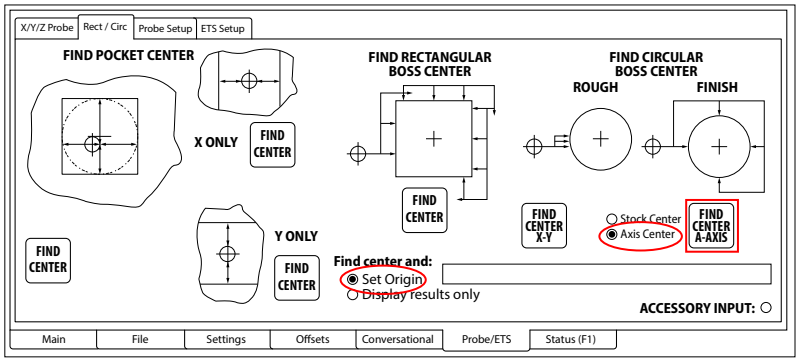
Setting Workpiece Axis Center Coordinates

9. The following procedure is suitable with 4-axis milling operations. Once the workpiece is mounted securely on the chuck, carefully jog the Probe to the side closest to you. Jog the probe in the Z direction to be approximately level with the center axis of your workpiece.



Passive Probe Tool

10. Make sure that **Axis Center** and **Set origin** is selected below. Once the probe is in position, click Find Center A-Axis. The 4th axis will automatically rotate for the probe to contact multiple points on the workpiece. This will calculate the location of the X-axis of the workpiece and the rotor.



Step 3:

Mill your project

POWERING ON PATHPILOT + PCNC MILL

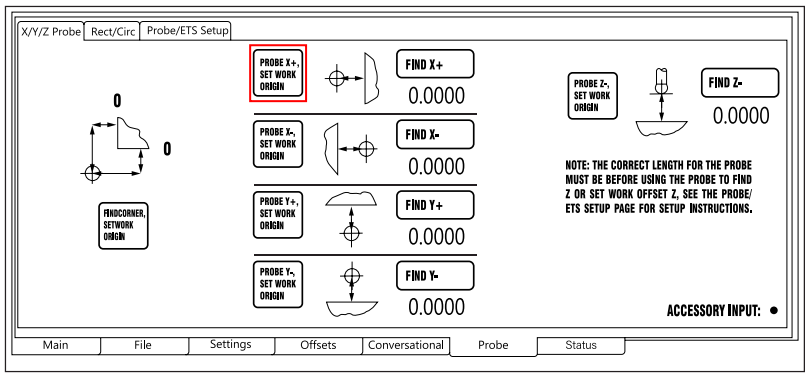
TORMACH PATHPILOT INTERFACE

1. Reference the Mill
2. Workpiece Preparation
3. Tooling Preparation
4. Tool Offsets
5. **Work Offsets**
6. Import g-code file into PathPilot interface
7. Running the g-code file on the PCNC Mill

POWERING OFF PATHPILOT + PCNC MILL

Setting X Workpiece Coordinates

11. After locating the x-axis, you must locate the origin point along that datum. Typically, it could be located at either end of your workpiece: at the jaws of the chuck or at the opposite end. Be sure that it corresponds with the location of the origin you set in your machining model. To locate the origin (X0,Y0,Z0), jog the probe approximately 1" from either end and click **Probe X-, Set Work Origin.** This will shift the X coordinate of the origin along the X-axis to the probed face.



Step 3:

Mill your project

POWERING ON PATHPILOT + PCNC MILL

TORMACH PATHPILOT INTERFACE

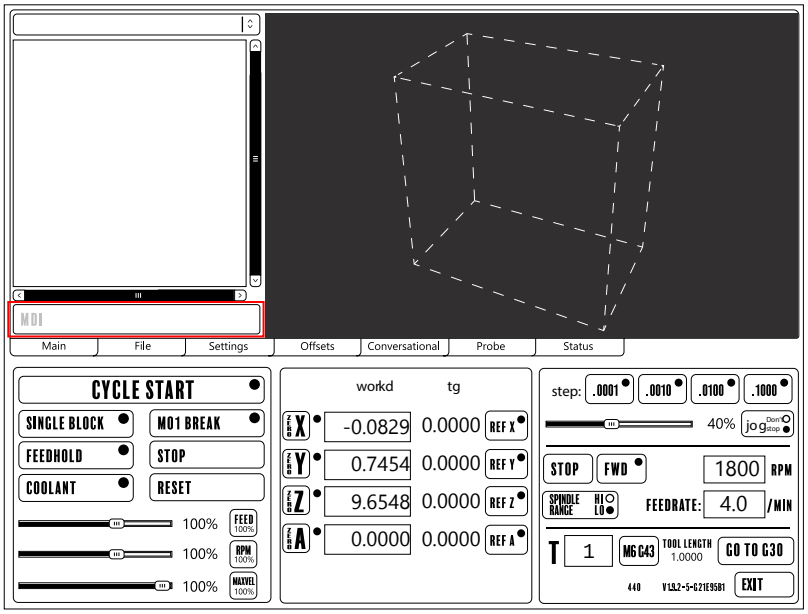
1. Reference the Mill
2. Workpiece Preparation
3. Tooling Preparation
4. Tool Offsets
5. **Work Offsets**
6. Import g-code file into PathPilot interface
7. Running the g-code file on the PCNC Mill

POWERING OFF PATHPILOT + PCNC MILL

Homing to Origin

12. Check the location of the origin by using the G0 command with the *MDI command bar* in the *Notebook* section of the *Main* PathPilot interface. Jog the probe a safe distance away from the workpiece. Type in 'G0' and then any combination of the 'X0Y0Z0' coordinates (ex. G0X0Y0) - this will bring the probe to an approximate location above or beside the origin. SLOWLY jog the probe closer to the origin point in either the x,y,or z direction.

PathPilot Interface



Step 3:

Mill your project

POWERING ON PATHPILOT + PCNC MILL

TORMACH PATHPILOT INTERFACE

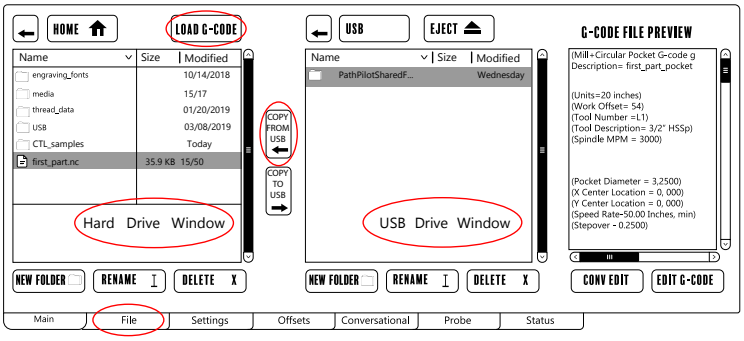
1. Reference the Mill
2. Workpiece Preparation
3. Tooling Preparation
4. Tool Offsets
5. Work Offsets
6. **Import g-code file into PathPilot interface**
7. Running the g-code file on the PCNC Mill

POWERING OFF PATHPILOT + PCNC MILL

Import g-code file into Pathpilot Interface:

1. Insert your USB with the g-code file into the PathPilot computer.
2. Open the **File tab** within PathPilot. Select your file within the centrally located USB Drive Window and hit the Copy from USB button to the left. This moves a copy of the selected file from your USB to the computer's hard drive, and will appear in the **Hard Drive Window** to the left. Finally select your file and hit the Load g-code button above.

   This should automatically open a preview of your file in PathPilot's Main tab.



PathPilot Interface

Step 3:

Mill your project

POWERING ON PATHPILOT + PCNC MILL

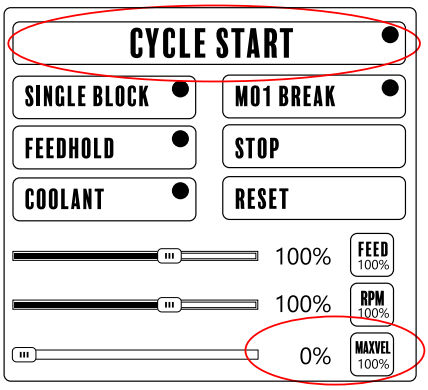
TORMACH PATHPILOT INTERFACE

1. Reference the Mill
2. Workpiece Preparation
3. Tooling Preparation
4. Tool Offsets
5. Work Offsets
6. Import g-code file into PathPilot interface
7. **Running the g-code file on the PCNC Mill**

POWERING OFF PATHPILOT + PCNC MILL

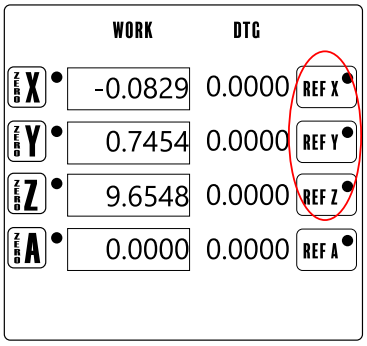
Running the g-code file on the PCNC Mill

1. Turn the **MaxVel 100%** slider down to **0%,** by clicking and dragging it.
2. Turn on the coolant pump and Click **Cycle Start.**



PathPilot Interface

3. NOTE: On occasion, the machine requires re-referencing before beginning the cycle. If this is necessary, the **Status Tab** will turn red and clicking on it will display an error message regarding the axes. To resolve, press **Ref X, Ref Y and Ref Z** buttons to re-reference the mill.
4. Go back to the **Main tab** and click the **Cycle Start** button again. This should start the spindle. Carefully, slide the **MaxVel 100% slider to 5%** and pay attention as the machine re-positions itself to begin milling. When the tool is a couple of inches above your workpiece, reduce the **MaxVel 100% slider back to 0%.**



PathPilot Interface

Step 3:

Mill your project

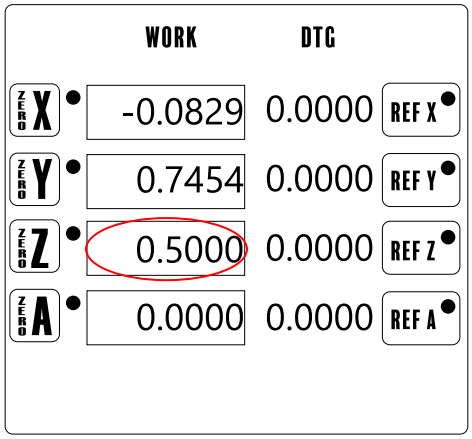
POWERING ON PATHPILOT + PCNC MILL

TORMACH PATHPILOT INTERFACE

1. Reference the Mill
2. Workpiece Preparation
3. Tooling Preparation
4. Tool Offsets
5. Work Offsets
6. Import g-code file into PathPilot interface
7. **Running the g-code file on the PCNC Mill**

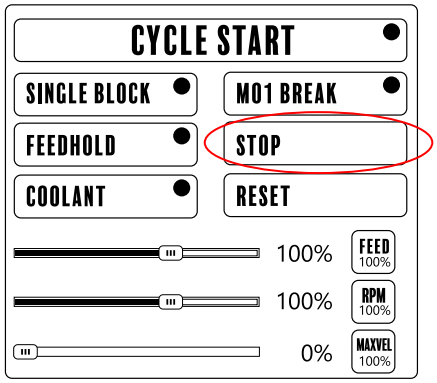
POWERING OFF PATHPILOT + PCNC MILL

5. Double check whether the tool position is correct by referencing the X, Y and Z Work Offset DROs. For example, if the tool looks like it has stopped ½" above your workpiece, the Z DRO should read 0.5000.



PathPilot Interface

6. If the tool position aligns with the DRO values, gradually increase the **MaxVel 100%** slider to resume the cycle.
7. Keep a close eye on the machine throughout the cut, and be ready to **Stop.** Only use the **E-Stop** button for emergencies and when powering down the machine as all the offsets will be lost once pressed.



ALWAYS be ready to click stop once the spindle is on!

PathPilot Interface

8. Once the cut is complete, push the **E-Stop** button, open the machine envelope and remove your milled workpiece from the vise.

Step 3:

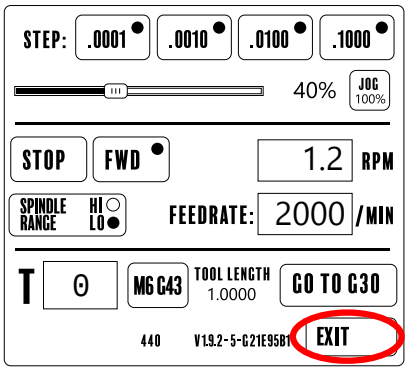
Mill your project

POWERING OFF PATHPILOT + PCNC MILL

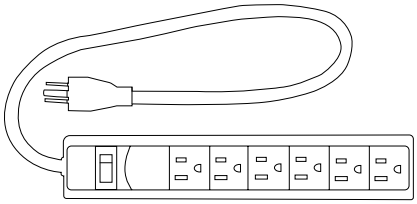
Please execute the four steps listed below, in order.

1. Push the red **E-stop** button in

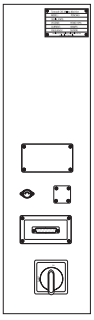
2. **Click EXIT** on the screen (located on the bottom right corner); when prompted click OK to power off



3. Turn the PathPilot Controller power strip OFF



4. Turn the Main Disconnect switch OFF (located on the right side of the machine)



[1] Tormach PCNC 440 Operator Manual. Tormach Inc., Waunakee, WI., United States of America, 2017.