MINITECH-GX MICROMILL

ABSTRACT
The purpose of this document is to give an overview of how to turn a Solid works drawing to G code and how to operate the micro mill to print the G code patterns. This document will also go over some terminology used in milling, calculating speed of your tool and basic safety procedures for the Micro mill. The Appendix will include trials and errors of different speeds and feeds for different size bits.

By: Bobby Ngo

Contents

Introduction
CNC Machinery

Machining in general is a way to transform a stock piece of material such as plastic to a finished product by means of a controlled material removal process. The CNC machine interprets the design as instructions for cutting patterns. Automated cuts improve both the speed and the accuracy with which prototype parts can be created - especially when the material is critical.

Machinery Geometry

The mini mill's three table motions are classified as X, Y, and Z axis as indicated in the figure below. These axes are motored controlled and the axis orientation is standard to most industrial operations. The mini mill can perform three basic types of work; milling, drilling and boring.
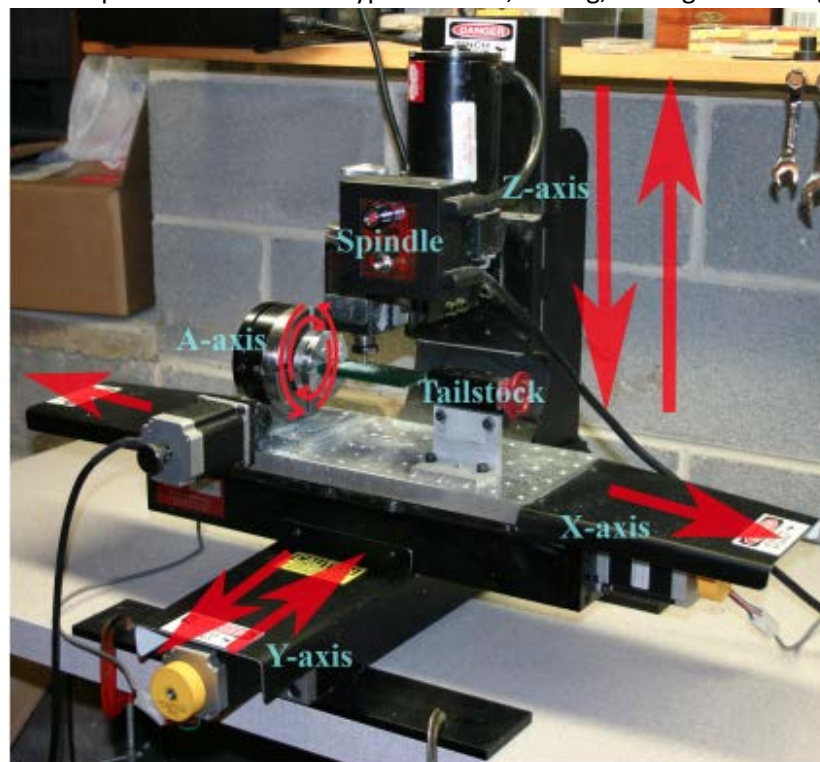

Figure 1: Micromill GX

On the milling tool, there a few components that needs to be define. The first component is the spindle. The spindle is a rotating axis of the machine that is spun by a separate motor next to the machine and is used to rotate the cutting tool. The cutting tool held to the spindle by using a collet, shown in the figure below. Most of the time the collet should be attached to the spindle already. Prior to using the machine, ensure that the collet is the appropriate size for the cutting tool. With the appropriate bit inside the collet; the collet should be able to be tighten. The next component is the vice grips. The vice grip is used to hold down the stock. If the stock is not properly held down, damages to the machine and injuries to the user may occur. It is important to use machining parallels to ensure proper flatness of the stock and also gives the user a few extra inches to mill without hitting the vice grip. Machining parallels are shown in the figure below.
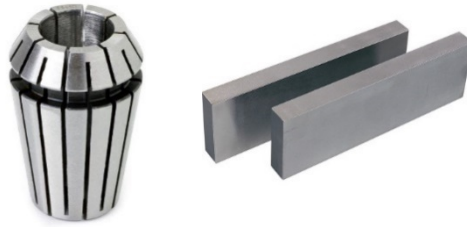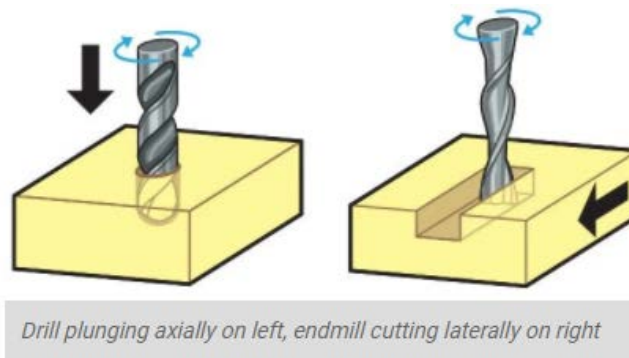
Figure 2: Collet and machining parallels

Machining is a subtractive process that uses rotational cutting tools called "end mills" to remove material. An end mill is similar in appearance to a drill bit but it is far more versatile. However, in practice the terms "bit" and "end mill" are often used interchangeably.

Drill plunging axially on left, endmill cutting laterally on right

Figure 3: Drilling vs Milling

Here is the key difference between the end mill and the drill bit. Drill bits are designed to plunge directly into the material, cutting axially and creating a cylindrical hole (shown above on the left). Usually these drill bits will have a sharp pointed end. While end mills are typically used for horizontal carving and laterally cutting, (shown above on the right) in some applications the endmill can be used for drilling and cutting, but it is good practices to use the drilling bit to drill and the end mill for cutting.

Bits Geometry

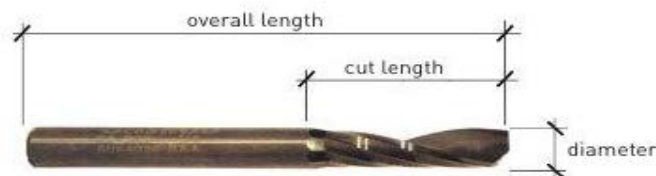Cutting tools come in various ranges of sizes, materials, and cutting geometry.

Figure 4: Cutting Tool

Each cutting tool has an overall length, cutting length and the diameter as shown in the figure above. The overall length is the length of the entire bit, the cutting length is the length of cutting end, and the diameter is the diameter of the cutting side. These dimensions will be important to determine the cutting speeds.
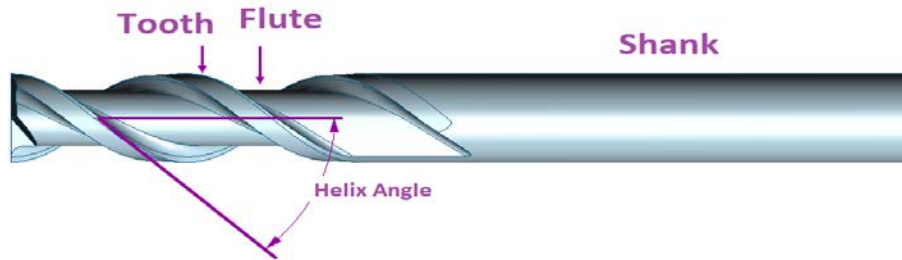
Flutes

Figure 5: Cutting Tool Flute

Flutes are the helical grooves that wraps around the side of the end mill, shown in the figure above. Each flute has a tooth with a sharp cutting edge. As the bit is plunge into the material, the flute will whip pieces of material as it cuts. The whip pieces is known as chips. The fewer the flute, the more chips it will eject with each tool rotation but rougher cuts. While the fewer flutes remove less material but smoother surface finish. The overall cutting depth should never exceed the length of the cutting length. If cutting deeper than the cutting length, the top of the cutting length will be blocked and the chips would not be able to clear. These build up can cause excessive heating and reduces the tool life.

Material

End mills are made either out of cobalt steel alloys known as high-speed steel (HSS), tungsten carbide (carbide) or cobalt steel. Carbide tools can usually be run at speeds 2 to 2.5 times faster than HSS tools. When using carbide tools ensure that your machine tool is rigid with a solid spindle and that the holder has little or no run out. Due to the brittle nature of carbide and the speeds at which carbide tools are typically run, rigidity is critical to prevent tool breakage. For finishing applications carbide is much more advantageous over HSS or cobalt. Carbide also provides better rigidity than HSS, which enables the end mill to provide a higher degree of dimensional accuracy and superior surface finishes. Carbide provides the user with the ability to run faster with less tool wear. However, the user should be aware that carbide's hardness comes at the expense of its toughness. Carbide is brittle and tends to chip when conditions are not ideal, rather than wear. Heavy feed rates are more suitable for HSS and cobalt tools. To see the difference in the material refer to the appendix.

Tip Geometry & Applications

Each end mill tip is designed for a specific purpose. Some common cutting shapes are ball nose, fishtail, surface planning, carving, and straight. Ball nose mills produce a rounded pass, while fishtail cutters will produce a flat surface and V bits will produce a V shaped pass. The figure below shows the difference in geometry and the different style of cut it produces. Keep in mind, when the cutting diameter gets smaller it will be very hard to tell what the cutting geometry is. Ensure to check the bit under a microscope and check the box it came in to define the proper cutting geometry.
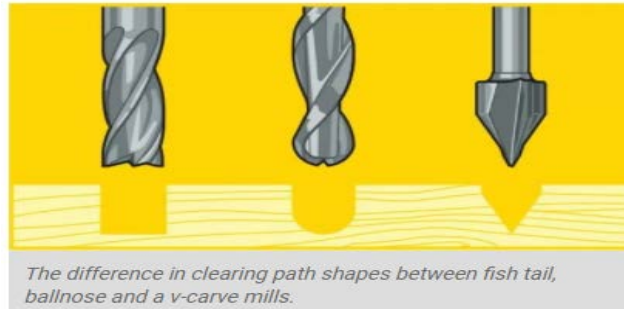
Figure 6: Cutting Geometry

Milling Tools are either center cutting or non-center cutting. Center cutting mills can plunge straight into material, while non-centering cutting tools cannot. Notice that the cutting edges of the center cutting end mill continue to the center of the tool. The center of the other has a small hole at the center. Non-centering cutting end mill requires a pilot hole, ramping, or helical motion to plunge into material. The way you can tell the differences is the teeth of some cutters do not go all the way to the center of the end face. Refer to the appendix for more cutting geometry.

Feeds and Speeds

The speed at which the cutter moves across the material is called the feed rate. The rate of bit rotation is known as speed. The speed is dictated by how fast the spindle turns the cutting tool. Both the feed rate and speed will vary based on the material being cut. A general rule of thumb is that the tool should to move though the material as fast as possible. The longer the tool rotates in one area, more heat will generate. The heat generated can cause the material to burn or radically decreases the life of the cutting tool. If the spindle speed is too fast in combination with slow feed rate, it can cause burning or melting of the material. If the spindle speed is to slow in combination with fast feed rate can result in dulling of the cutting edge, deflection of the end mill and possibility breaking the end mill.

Calculating Feed and Speeds

When milling, drilling, or creating a toolpath for a CNC machine the feed rate must be determine manually or using an online calculator. In order to calculate the speeds and feed, the following dimensions are needed; Stock material, tool type, tool material, tool dimension, and machines max RPM. The following link will do the calculations automatically with the define dimensions. http://micro100.hsmadvisor.com/?shell_id=247

Spindle Speed RPM:

$$RPM = 12 * \frac{SFM}{\pi * cutter\ diameter}$$

The surface speed (SFM) can be looked up on a table in the appendix
Feed Rate:

$$Feed\ Rate = Chip\ load * \#\ of\ flutes * RPM$$

RPM = Spindle RPM
Chip loads can be found on a chip load table. These numbers can also be found in the appendix.

G code

G code is a common name for the most widely used numerical control programming language. It is used mainly in computer aided manufacturing to control automated machine tools. The G code is the instruction given provided by people to create something. To see some common G code refers to the appendix.

Mini Mill Machine Operations Procedures

Safety

The most important lesson to learn is that of safety. Caution and safety must be considered at all times when using the Mill, as well as any other machine tool. User should learn and follow all safety rules any time they operate the Mill.

Safety Rules:

1. Always wear safety glasses – the operation of any power tool can result in flying foreign debris.
2. Never run a CNC Machine unattended - CNC machines are potentially dangerous. Untrained people must not operate the CNC machine.
3. Keep Guard in place- Never remove the safety shield. Use it for protection against flying debris.
4. Remove adjusting keys and wrenches- Form a habit of checking to see that keys and adjusting wrenches are removed from the spindle and machine prior to turning the machine on.
5. Keep work area clean – cluttered areas and benches invite accidents.
6. Do not force tool & Use the right tools – do not force tool or attachment to do a job it was not designed for.
7. Wear proper appeal- Avoid loose clothing or jewelry that could get caught in the moving parts of the machine.
8. Secure Work – use clamps or a vice to hold work. Do not use your hands to hold work down.
9. Do not touch machine when it is on – Avoid touching any parts of the machine while it is operating. If the proper settings are made, and the work is fastened properly, there is no need to touch the machine.
10. Avoid accidental starts – make sure spindle switch is off before plugging in the power cord.

Solidworks → G Code

Create your drawing in the front plane. The X Y and Z coordinates will match the coordinates of the milling machine, as shown below. You want the Z-axis to be pointing towards you, Y-axis pointing up, and X axis pointing to the left.
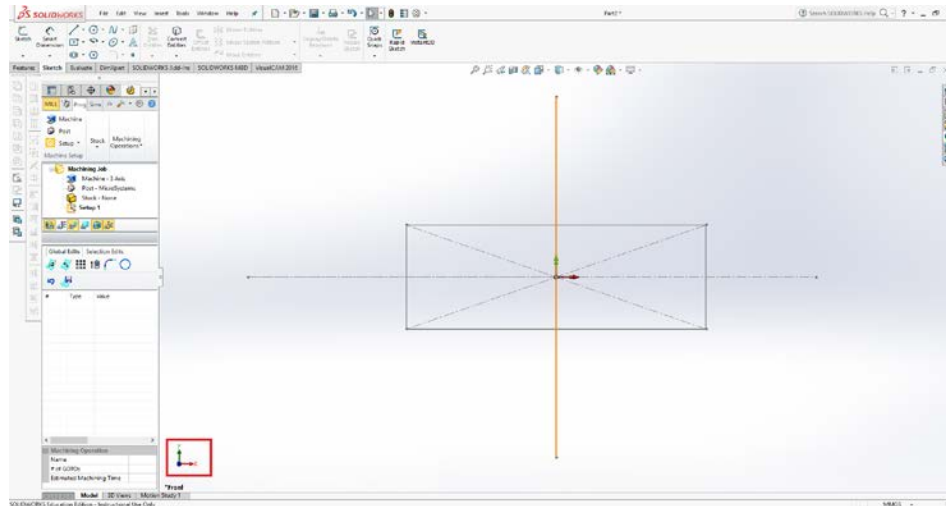

Figure 7: Coordinate system

1. To change the units use the drop down menu on the bottom right hand side. This is indicated in the little red box below.
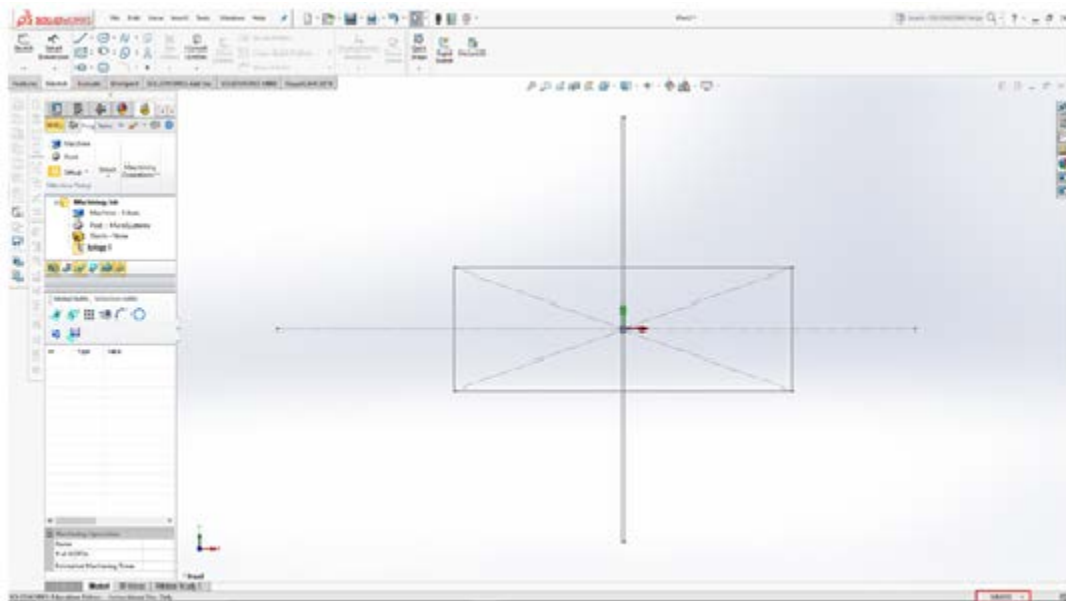

Figure 8: Changing Units

2. Once you have created your part, go to the visual cam tab and click the milling browser.
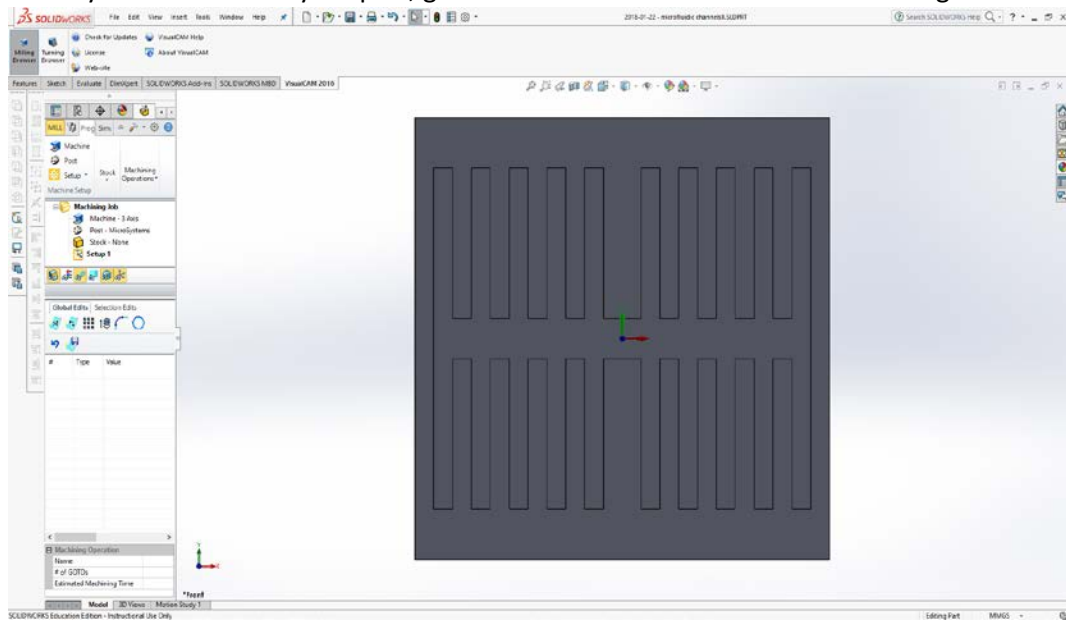

Figure 9: Visual Cam Milling Tab

3. In the milling tab select Machine → Align to → world coordinate system. This will align the work coordinate to your drawing coordinates.
4. Next move to the Machine tool definition tab → set number of axis to 3 → Set min and max for the three coordinates and hit OK. Note: do not use these coordinates below.
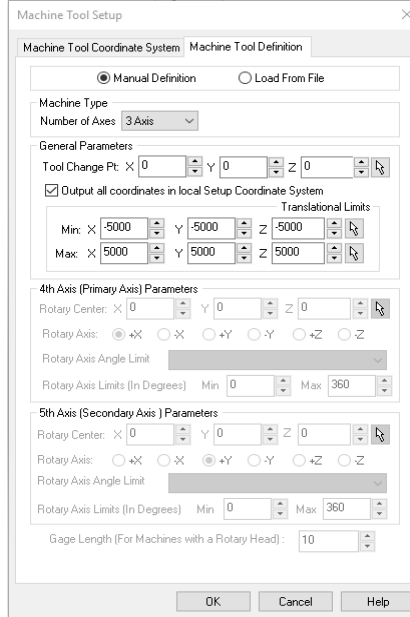

Figure 10: Milling Machine Dimensions

5. Now select the Post tab→ Set Current Post Processor to the settings shown below and hit ok when finish.
   NOTE: The post processor is what is used to create the G code. There are numerous post process but for this tool, it is Mach 3 post process.
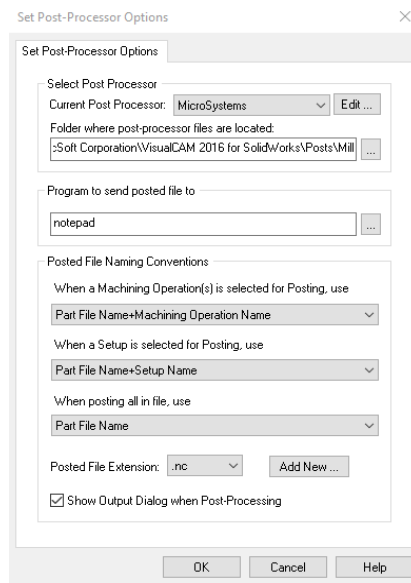
Figure 11: Post Process

6. Select the Stock drop down menu → select box stock →Define the geometry of the stock and hit ok
   Note: measure your stock piece and enter those dimensions here

Figure 12: Stock Dimensions

7. Select Align → set Z alignment to bottom and XY Alignment to center and hit ok.
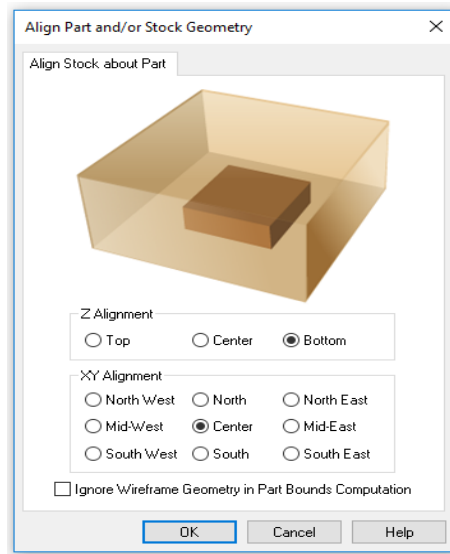
Figure 13: Aligning Part to Stock

8. Select Materials under the stock tab → select the material and hit ok. Note: In this case the selected stock is polycarbonate plastic.
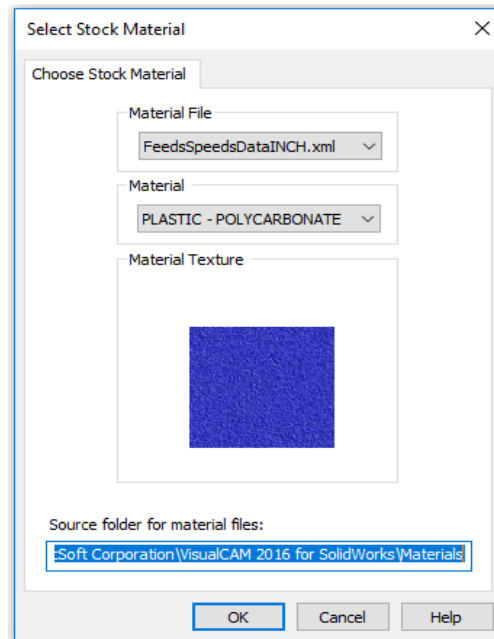
Figure 14: Selecting Stock Materials

9. Under Materials Operation → select work zero to the setting shown below and save
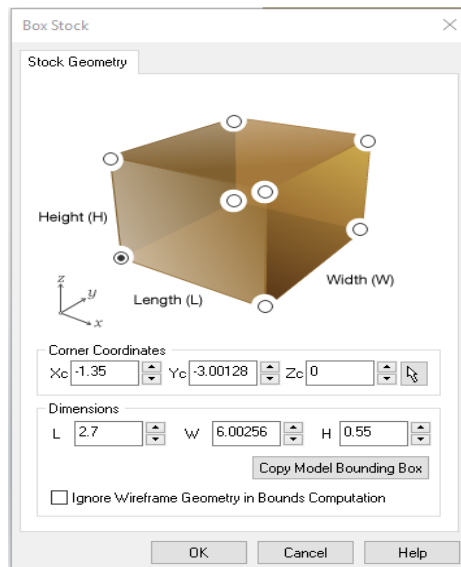   Note: This is where to set up your zero point on the machine. In this case it is set to the bottom right corner of the stock as indicated by the ball.
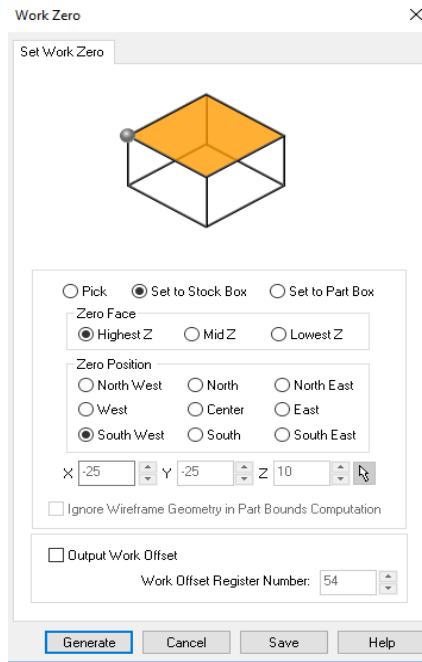
Figure 15: Zero Coordinates

10. To make channels in the stock select the Program tag → 2-axix →pocketing and this window should pop up.
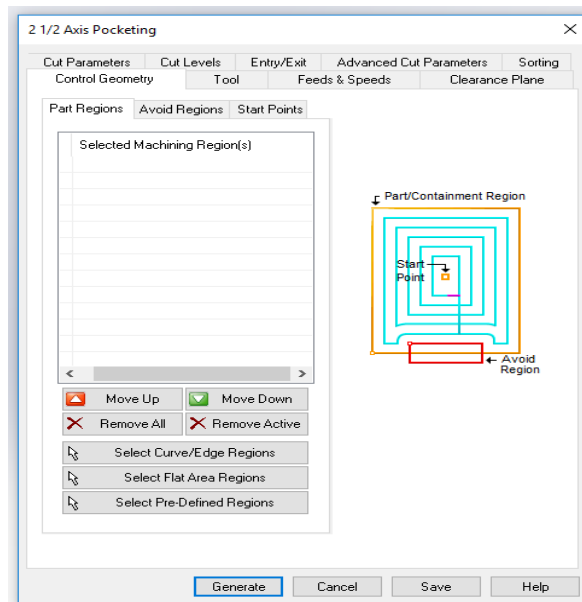
Figure 16: Defining Cutting Path

11. Select the curve / Edge Regions → select the outer edges of the hole as shown below.
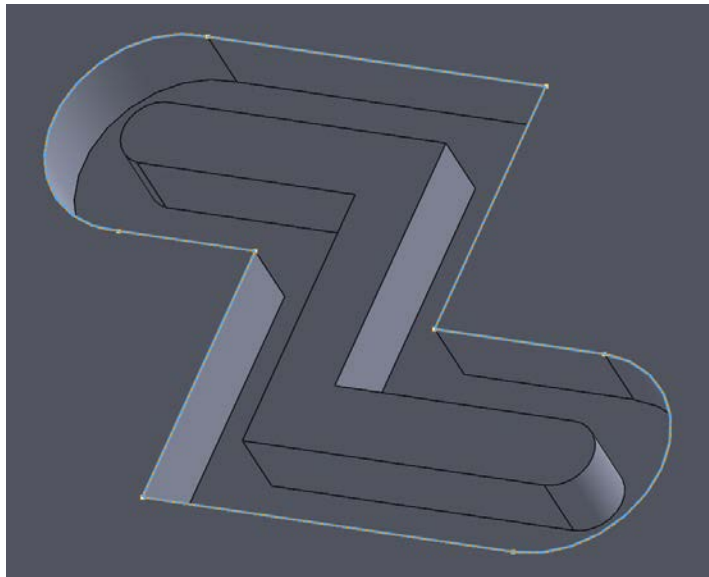
Figure 17: Picking the Cutting Edge

12. Select Cut Parameters tab → set the parameters to your design. Note: Most of the time the program will fail to run due to these parameters. Adjust these values as needed.
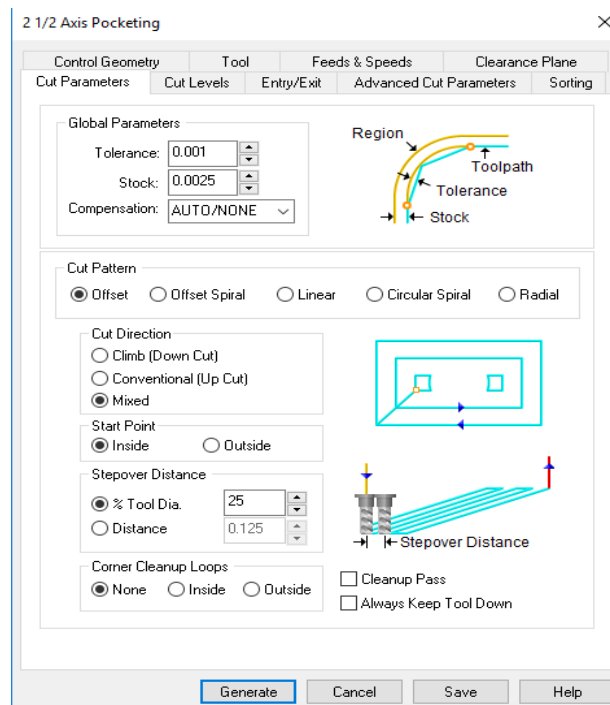
Figure 18: Defining Tolerances

13. Select the Tools tab → select the type of tool for the job → enter in the dimensions for the tool → save as new tool and hit Ok
    Ensure: to take proper measurements of the tool before entering it into the data below.
    Note: these dimensions are for a PMT 100 micron square endmill and they could be found on http://pmtnow.com/end-mill/Micron.
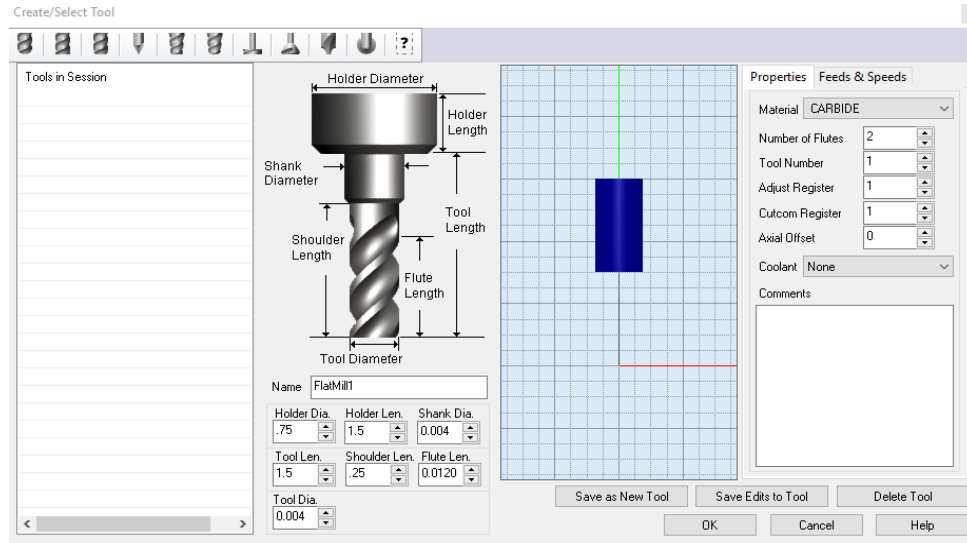
Figure 19: Defining Cutting Tool

14. Select cut levels → Make sure location of cut geometry is from the top → ensure cut depth control is specify on part and set the cutting level.
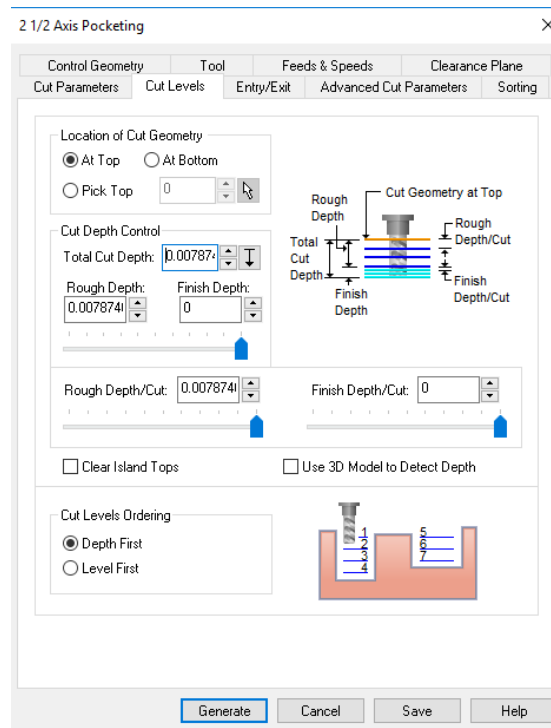
Figure 20: Cutting Levels

15. Under Feeds and Speeds tab → define the speed of the spindle.
    Note: for smaller bits use RPMS of 30,000-50,000
16. Under the clearance plane→ the plane should be set automatically to 0.25 inches above the stock.
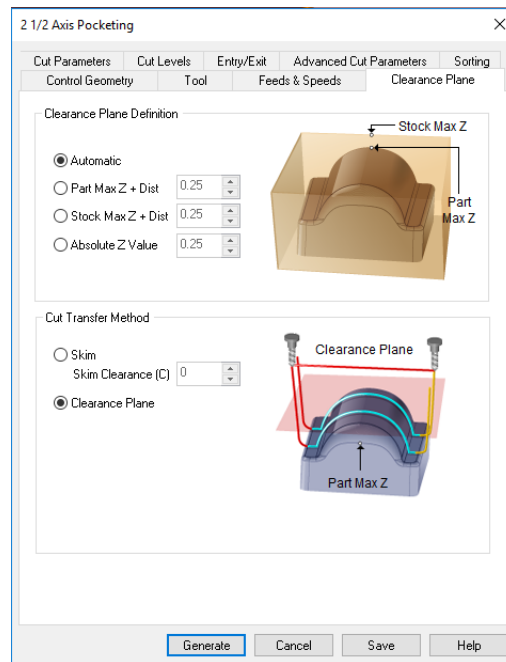
Figure 21: Clearance Plane

17. Once all parameters are set hit generate
18. To see the tool path under simulation → select play
    Note: It should show you your tool path and how much material is taking off per step.
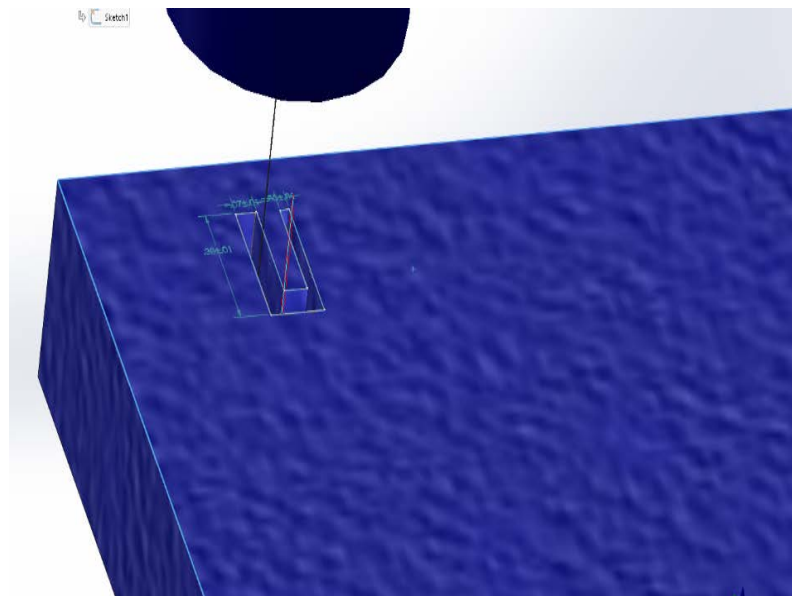
Figure 22: Cutting Simulation

19. To ensure proper tool path after the simulation go to the tree window → right click → select post process → this will produce the G code. Note: to ensure that the path is correct run the G code through an online path similar such as https://nraynaud.github.io/webgcode/ or https://ncviewer.com/
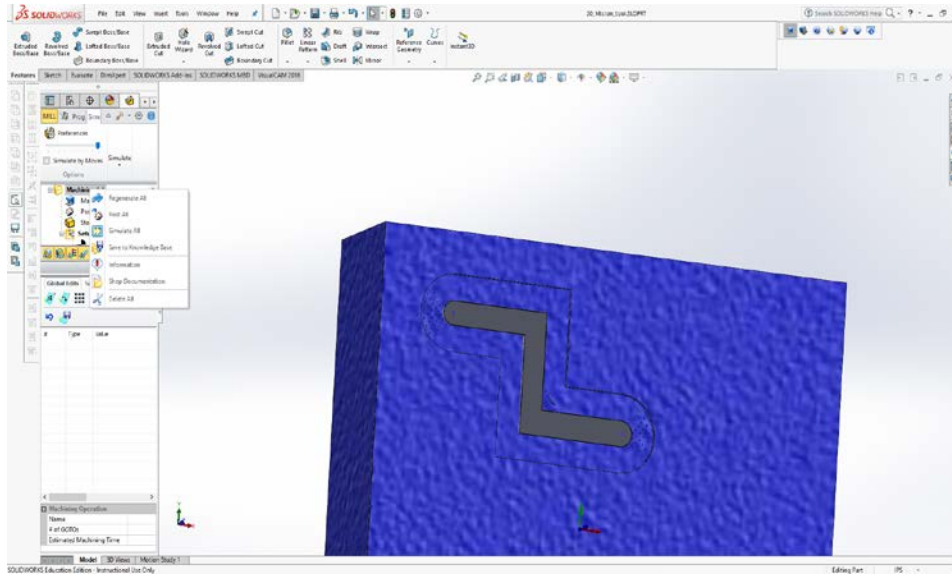
Figure 23: Creating Pose Process

SolidWorks G code → Milling

1 Before starting any job on the mill makes sure that the tool is cleaned and then turn on the spindle and the tool.

Figure 24: Spindle and MiniTech Tool

2   Ensure to clamp down the stock. In this scenario we will be using vice grips and parallels
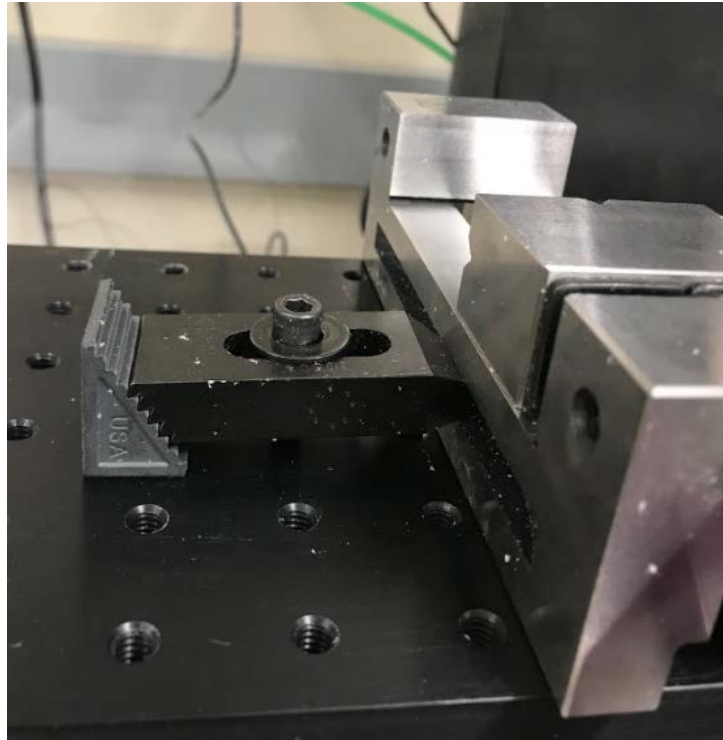
Figure 25: Vice Grip

3   On the computer screen launch the Mach3 CNC Program, the following screen will appear.
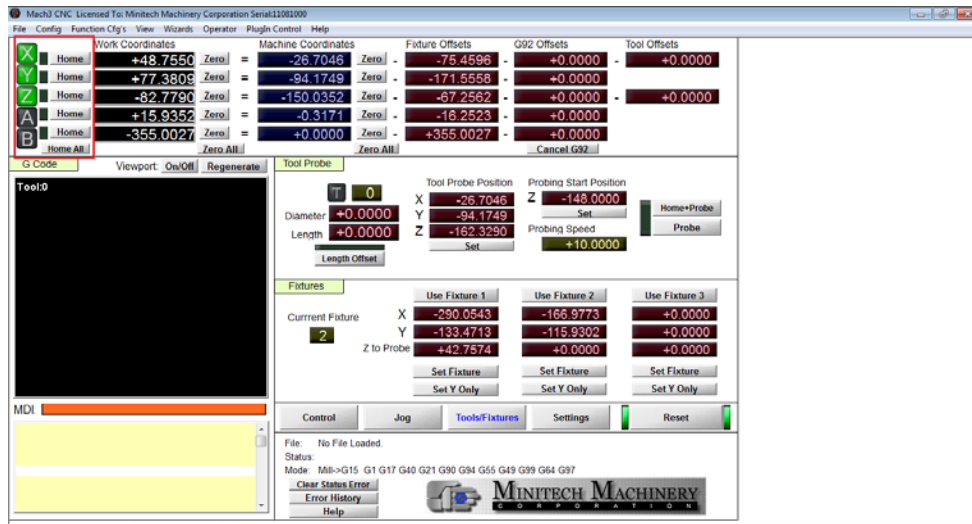
Figure 26: Homing Position

4   Once the program is loaded hit the reset on the bottom right hand corner and home all. When the machine is properly homed the X, Y, and Z will light up green, as shown in the figure above.

5   Next you will need to identify the work zero coordinates for the X, Y, and Z positions. To do this open Jog tab; the following screen should appear.
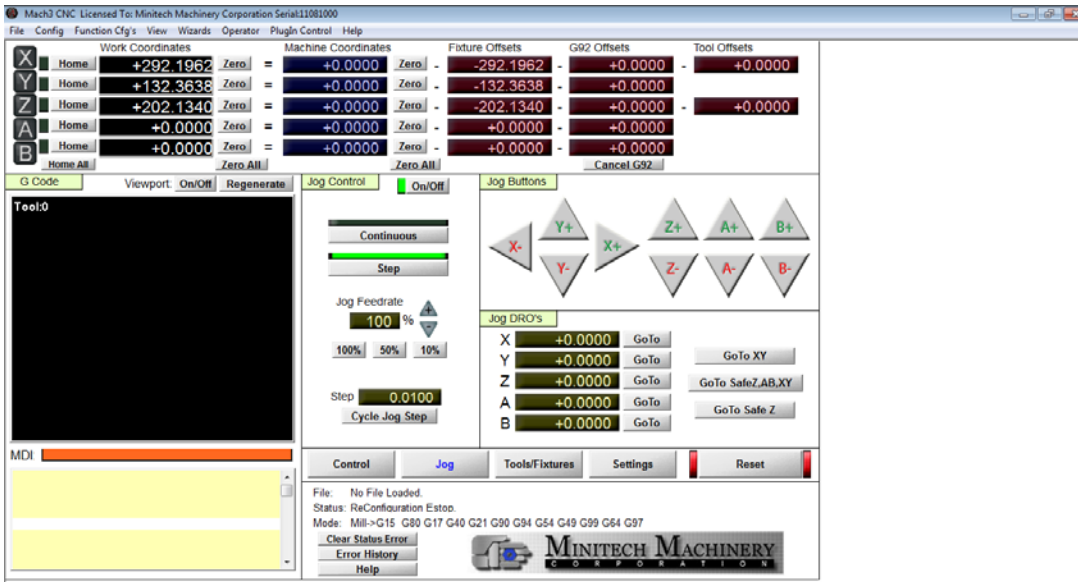
Figure 27: Jogging the Motor in Steps

To continuously move the X, Y, and Z motor click the Continuous bar until it glows green. To adjust the increments of the motor adjust the steps. In the figure above the motor is set to move in Steps of 001mm per step.

6   Before moving zeroing out the X and Y axis make sure that the Z axis is out of the way. Jog the motor until you reach the desire work zero coordinates. In references to the figure below the work zero coordinates is the bottom left corner of the stock.
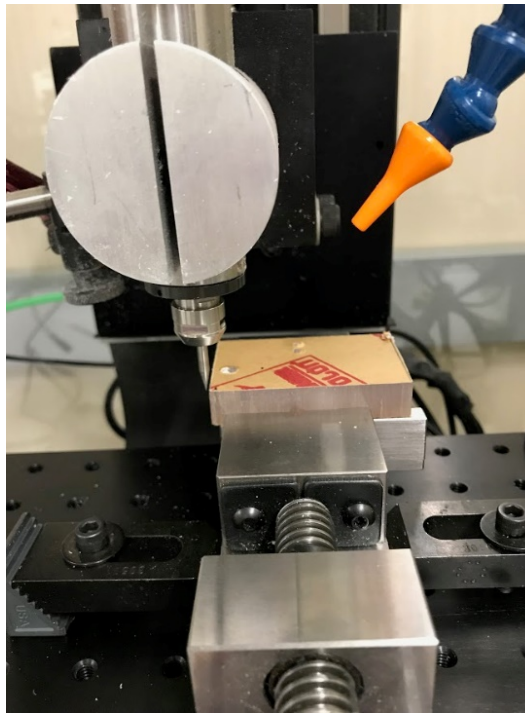
Figure 28: Zeroing Out the X and Y

7   For the Z-axis there are various ways to zero this out. First method is the paper method as shown in the figure below. You place a paper in between the stock and bit. Move the bit down slowly until the bit pins the paper against the stock. Once that's done hit zero on the Z work coordinates. This is shown in the figure below.
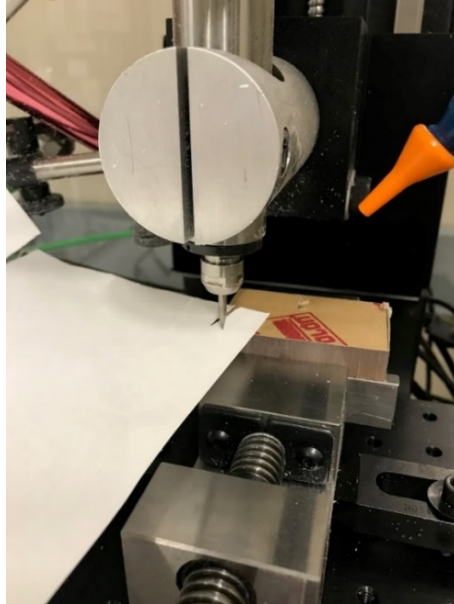
Figure 29: Zeroing Out the Z

8   The second method you will need the camera. To start place the bit into the holder but do not tighten the bit. Slowly move bit until you see the bit is pushed in by the stock. This will tell you that the bit is touching the stock and then tighten the collar. To ensure that it is still zero out. Turn on the spindle and move in increments of about 10 microns. Once you see small chips or cuts zero out the Z work coordinate and hit set fixture under fixture 1. This will prompt the machine that this is its starting point. The machine will now probe the bit to ensure maximum cutting precision.

9   To load the G code simple go to files → load G code. You should be able to see the code loaded in the yellow region under the G code tab.
Note: To see the cutting pattern click viewport on and generate, the desire cutting should appear just like the figure below.
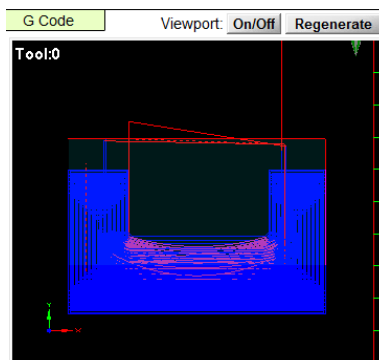
Figure 30: G Code Cutting Path

10   When running the G code it is a good practice to run it in block mode. Block mode allows you to run the code line per line manually, to check the machine calibrations. The first few lines are to set the safety mode of the machine. It allows the user to have the emergency stop option and tool change. To turn on block mode go to Control and hit block mode under the program tab as shown in the figure below.

Figure 31: Controlling the G Code

11   In block mode click cycle start to run the line of code. On N3 this will prompt the machine to move the bit to fixture 1 in a safe Z-axis. If this doesn't work you can move it manually to your work zero by hitting Go to Safe X, Y Z zero.
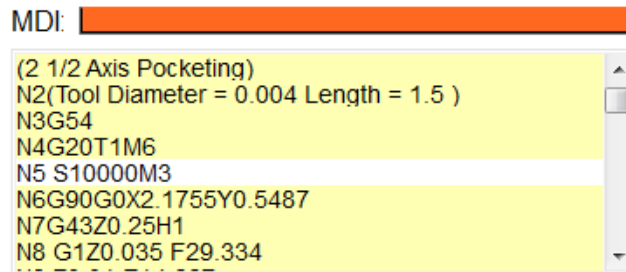
Figure 32: G Code Spindle Speed

12 On line N5 this code prompts the speed of the spindle. If this is not the proper speed you can change it in the code by simply hitting edit or manually set the speed on the left hand side. Both methods are shown below.
Note: for small bits it is recommended to run the spindle speed from 15000 – 50000 RPM and feed rate of 25-50 base on bit size.
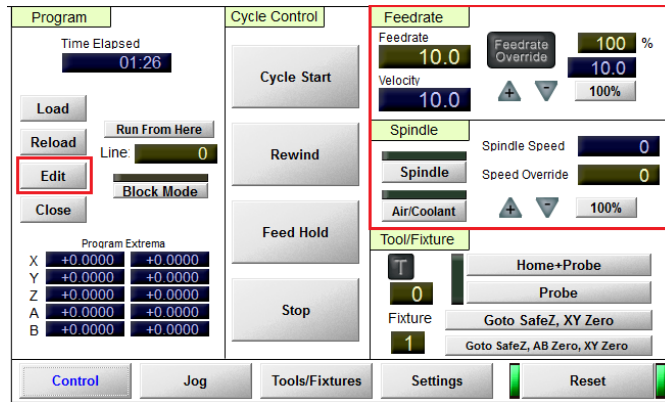
Figure 33: Changing Spindle and Feed Speeds

13 The next set of codes is the work zero to your cutting location. When running the next few lines of code it will move from the zero position to the cutting location. Check the bit location to ensure this is where cut needs to be. The following line N7 will move the bit 0.25 inches above the work surface. At this point if the bit is in the wrong position, stop the tool and recalibrate using the following steps above.
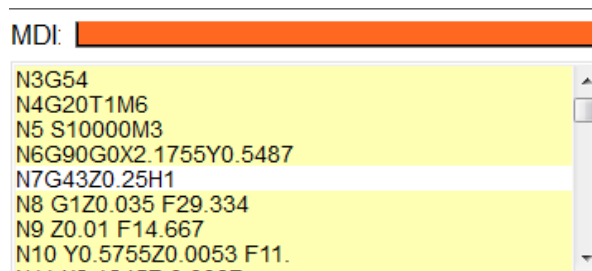
Figure 34: Safety Plane

14   Once everything is correct N8 code will move the bit into the stock and start cutting. At this point it is optional to take the tool out of block mode and have it run through the rest of the code. To do this simply click on block mode until the yellow light goes out and click cycle start.

Conclusion

This concludes the tutorial on how to use the CNC micromill. As you use the micromill keep in mind that the feeds and speed will very base on the bit and the stock you are using to cut. The feeds and speeds calculator will give you a rough estimate of what to use but through trial and error, you will see that the speeds and feeds can be adjusted to create smoother cuts. In the appendix there is a trial and error of different sizes bits ran at different feeds and speeds.

Appendix I G Code

Most CNC machine will use traditional Cartesian coordinates. In G- code the letter X controls the horizontal position or X-Axis of the machine, Y controls the vertical position, and the Z controls the depth or z axis. The distances travel is determined by the numbers next to the X, Y, Z letters. M codes stands for miscellaneous code. M06 is tool change and T is the tool slot.

G-Code words			
G0	Rapid Linear Motion	G59.2	Select Coordinate System 8
G1	Linear Motion at Feed Rate	G59.3	Select Coordinate System 9
G2	Arc at Feed Rate	G80	Cancel Modal Motion
G3	Arc at Feed Rate	G81	Canned Cycles – drilling
G4	Dwell	G82	Canned Cycles – drilling with dwell
G10	Set Coordinate System Data	G83	Canned Cycles – peck drilling
G17	X-Y Plane Selection	G85	Canned Cycles – boring,no dwell, feed out
G18	Z-X Plane Selection	G86	Canned Cycles – boring, spindle stop, rapid out
G19	Y-Z Plane Selection	G88	Canned Cycles – boring, spindle stop, manual out
G20	Length Unit inches	G89	Canned Cycles – boring, dwell, feed out
G21	Length Unit milimeters	G90	Set Distance Mode Absolute
G28	Return to Home	G91	Set Distance Mode Incremental
G30	Return to Home	G92	Coordinate System Offsets
G53	Move in Absolut Coordinates	G92.1	Coordinate System Offsets
G54	Select Coordinate System 1	G92.2	Coordinate System Offsets
G55	Select Coordinate System 2	G92.3	Coordinate System Offsets
G56	Select Coordinate System 3	G93	Set Feed Rate Mode units/minutes
G57	Select Coordinate System 4	G94	Set Feed Rate Mode inverse time
G58	Select Coordinate System 5	G98	Set Canned Cycle Return Level
G59	Select Coordinate System 6	G99	Set Canned Cycle Return Level
G59.1	Select Coordinate System 7		

Appendix II Chip load Table & SFM

If Chip load data can't be found on the chart below refer to this web page
http://www.onsrud.com/xdoc/FeedSpeeds

Chip Load Chart

Tool Diameter	Hard Wood	Softwood Plywood	MDF/Particle Board	High Pressure Laminate **	Phenolic **
1/8"	.003" - .005"	.004" - .006"	.004" - .007"	.003" - .005"	.004" - .005"
1/4"	.009" - .011"	.011" - .013"	.013" - .016"	.009" - .012"	.011" - .012"
3/8"	.015" - .018"	.017" - .020"	.020" - .023"	.015" - .018"	.017" - .018"
1/2" & up	.019" - .021"	.021" - .023"	.025" - .027"	.023" - .025"	.024" - .026"

**Recommended RPM 9-10,000

Tool Diameter	Hard Plastic	Soft Plastic	Solid Surface	Acrylic	Aluminum
1/8"	.002" - .004"	.003" - .006"	.002" - .004"	.003" - .005"	.003" - .004"
1/4"	.006" - .009"	.007" - .010"	.006" - .009"	.008" - .010"	.005" - .007"
3/8"	.008" - .010"	.010" - .012"	.008" - .010"	.010" - .012"	.006" - .008"
1/2" & up	.010" - .012"	.012" - .016"	.010" - .012"	.012" - .015"	.008" - .010"

Other Valuable Formulas:

Feed Rate = RPM x number of flutes x chip load

RPM = feed rate/(number of flutes x chipload)

Metric Conversion: Divide inches per minute by 39.374 (ex. 300 inches per minute divided by 39.374 = 7.62 meters per minute)

RPM Selection - the general operating RPM for tooling contained in this catalog is between 10,000 and 20,000 revolutions per minute. Usually the higher the RPM, the better surface finish becomes. However, the higher the RPM, the higher the friction generated between the tool and the work piece. This friction is what creates the mechanical wear on the cutting edge. Your goal is to select the lowest RPM possible for each application.

SPEEDS & FEEDS

SOLID CARBIDE HIGH PERFORMANCE END MILLS

Materials Soft Grades	Speed S.F.M. Under 32 HRC	Materials Hard Grades	Speed S.F.M. Over 32 HRC	Feed (Inch/Tooth) End Mill Diameter								Recommendation for Coated Tools
				1/8"	3/16"	1/4"	3/8"	1/2"	5/8"	3/4"	1"	
NONFERROUS MATERIALS												
Aluminum + Aluminum Alloys 2024-T4/T6, 2014, 6061-T6/T651, 7075-T6	800-2000	440, 356, 380, C61300	500-1000	.001	.002	.002	.003	.004	.005	.006	.007	TiCN add 40% on SFM
Copper Yellow Brass, High Lead Brass, Red Brass	800-1500	Navel Brass, High Silicon Bronze, A-17, C-17200	800-1000	.001	.001	.002	.0025	.003	.004	.004	.005	
Copper Alloys Alum/Bronze, Low Silicon Bronze	800-1000	Nickel Silver, Beryllium Copper, Oxygen-Free Copper	700-1000	.001	.001	.002	.0025	.003	.004	.004	.005	TiCN add 40% on SFM TIAIN add 60% on SFM
Magnesium De-CAt, Extruded	1000 min	Nickel Silver, Beryllium Copper, Oxygen-Free Copper	700-1000	.001	.002	.002	.003	.004	.006	.008	.009	
Plastics, Acrylics, Phenolics Polysulfone	200-600	Polycarbonate	200-500	.001	.002	.003	.004	.006	.008	.010	.015	
Carbon, Graphites	200-400			.004	.004	.006	.008	.010	.010	.015	.020	

Materials Soft Grades	Speed S.F.M. Under 32 HRC	Materials Hard Grades	Speed S.F.M. Over 32 HRC	Feed (Inch/Tooth) End Mill Diameter								Recommendation for Coated Tools
				1/8"	3/16"	1/4"	3/8"	1/2"	5/8"	3/4"	1"	
STEELS												
High Steel Strength 4340, 6150, 52100, H-11, H-13	50-250	4340M, EDT-150, HP9-430, 300M, D6-Ac, 11-10	80-180	.0005	.0005	.001	.001	.001	.0015	.002	.004	On Steels >45 HRC on Roughing use Coolant on Finishing Run Dry TiCN add 40% on SFM TIAIN add 60%-100% n SFM
High Alloy Steels A-2/6/10, P-3/10, 01, 02, 06	100-300	Hy-Tugg, Stressproof, AmorPlate	80-180	.0005	.0005	.001	.001	.001	.0015	.002	.004	
Medium Alloy Steels 200, 250, 300	150-350	Hy-Tugg, Stressproof, AmorPlate	80-180	.0005	.0005	.001	.002	.0025	.0035	.004	.005	TiCN add 40% on SFM
Low Alloy Steels Maraging 10XX, 11XX, 13XX	100-400	23XX, 31XX	100-200	.0005	.001	.0015	.002	.003	.004	.005	.006	

Materials Soft Grades	Speed S.F.M. Under 32 HRC	Materials Hard Grades	Speed S.F.M. Over 32 HRC	Feed (Inch/Tooth) End Mill Diameter								Recommendation for Coated Tools
				1/8"	3/16"	1/4"	3/8"	1/2"	5/8"	3/4"	1"	
CAST IRON												
Ductile Cast Iron	100-400	Ductile Cast Iron	80-140	.0005	.001	.0015	.002	.002	.003	.004	.005	Always use coolant TiCN add 40% on SFM TIAIN add 60%-100% n SFM
Gray Cast Iron	100-400	Malleable, Chilled	90-160	.0005	.001	.002	.003	.004	.005	.006	.008	TiCN add 40% on SFM TIAIN add 60%-100% n SFM

Materials Soft Grades	Speed S.F.M. Under 32 HRC	Materials Hard Grades	Speed S.F.M. Over 32 HRC	Feed Inch/Tooth End Mill Diameter								Recommendation for Coated Tools
				1/8"	3/16"	1/4"	3/8"	1/2"	5/8"	3/4"	1"	
TITANIUM												
Titanium Alloys Commercially, Pure, 6AL-4V, Astm 1/2/3, 6 AL-25N-4Zr-2Mo-Si	50-250	5AL-2.5Sn-Eli 8Al-1 Mo-1V	90-160	.0005	.0005	.0005	.001	.001	.0015	.002	.004	Always use coolant TiCN add 40% on SFM TIAIN add 60%-100% n SFM

Materials Soft Grades	Speed S.F.M. Under 32 HRC	Materials Hard Grades	Speed S.F.M. Over 32 HRC	Feed (Inch/Tooth) End Mill Diameter								Recommendation for Coated Tools
				1/8"	3/16"	1/4"	3/8"	1/2"	5/8"	3/4"	1"	
STAINLESS STEEL												
Precipitation 13/8, 15/5, AM-350/355	80-250	17/4, 17/7, AF-71, Custom 450/636, 15/7 Mo, 21-6-9	90-150	.0005	.0005	.001	.001	.002	.002	.004	.006	TiCN add 40% on SFM TIAIN add 60%-100% on SFM
Austenic 200 Series, 302, 303, 304L, 316L	100-350	304, 310, 314, 316, 321, 330, 347, 348, 21-6-9	100-150	.0001	.0002	.0005	.001	.0015	.002	.003	.004	
Martensitic 403, 410, 416	100-250	420, 430F, 440C, 446	100-175	.0001	.0002	.0005	.0005	.001	.001	.003	.004	

Materials Soft Grades	Speed S.F.M. Under 32 HRC	Materials Hard Grades	Speed S.F.M. Over 32 HRC	Feed (Inch/Tooth) End Mill Diameter								Recommendation for Coated Tools
				1/8"	3/16"	1/4"	3/8"	1/2"	5/8"	3/4"	1"	
HIGH TEMP ALLOYS												
Cobalt Base Stellite, HS-21, HAYNES25/188, x-40, L-605	60-100	Air-Resist 13/213/215, Haynes 21/36, NASA CO-W-RE	40-80	.0004	.0005	.001	.001	.001	.001	.0015	.002	TIAIN Coating is required add 50%-80% in SFM
Nickel Base INCONEL 600/625, Nickel 200-270, Monel 400-405		Hastalloy-C/B/X, INCONEL 718/X/W, Waspalloy, Rene 41-95	50-90	.0005	.0005	.001	.001	.002	.0025	.003	.004	
Iron Base Incoly 600-802, Multimet N-155, Timkin 16-25-6	80-130	A-286, Haynes 556, Discoly, V57	60-120	.0005	.0005	.001	.0015	.002	.0025	.003	.004	

Appendix III Types of bits

Rough End Mill: The rough end mill is usually used for the initial cut. The rough end mill can be used at faster speeds but the cuts will be a lot rougher.

Finishing End Mill

The finishing end mill is a bit smaller than the roughing end mill and this is usually used at the end of the surface cut to give a smooth finish.

Ball End Mill: Ball End Mills will leave machine-rounded details in the stock.

Drill End Mill: This is known for drilling slots, side milling, beveling, and profiling

CDTooling Drill Mill Solid Carbide End Mills
2 and 4 Flute, 60, 90, 120, Degree ID 829

Fishtail / Square end mill: The square or fishtail end mill geometry is design to create square corners at the bottom of the channel

Appendix IV Types of Material and Coating

High Speed steel

Cobalt coated tools

Carbide

Appendix V Trial and Errors

The following are a few of my personal trial and errors for different size bits.

Bit size	Bit type	Spindle RPM	Feed Rate	Comments
100 micron	square	50,000	14.7	Stock melt completely on bit. Bit is unrecoverable
100 micron	square	20,000	14.7	Feed rate is to slow stock melts onto the tool but the bit can be recovered
100 micron	square	15,000	14.7	Feed rate is to slow stock melts onto the bit but the bit can be recovered
100 micron	square	15,000	24.7	Feed rate is to slow stock melts onto the bit but the bit can be recovered and pattern came out ok

100 micron bit with 50000 rpm and feed rate of 14.7