

# 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 Machinery4
Machinery Geometry
Bits Geometry
Flutes6
Material6
Tip Geometry & Applications6
Feeds and Speeds7
Calculating Feed and Speeds7
G code
Mini Mill Machine Operations Procedures8
Safety8
Solidworks $\rightarrow$ G Code9
SolidWorks G code $\rightarrow$ Milling
Conclusion23
Appendix I G Code24
Appendix II Chip load Table & SFM25
Appendix III Types of bits
Appendix IV Types of Material and Coating29
Appendix V Trial and Errors

## 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.

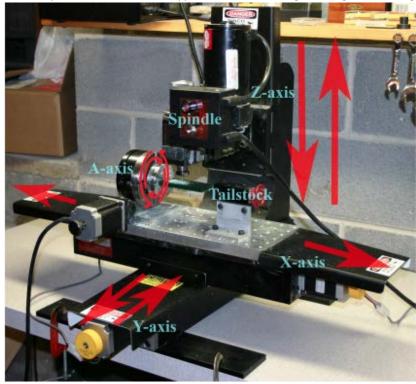


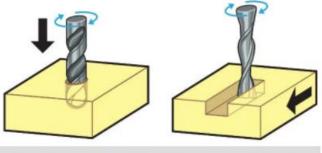
Figure 1: Micromill GX

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.

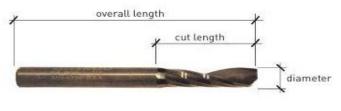
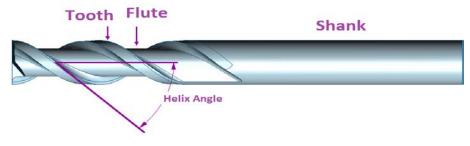


Figure 4: Cutting Tool

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.

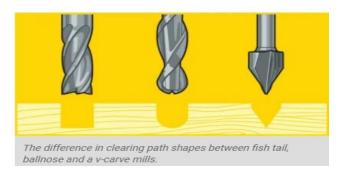


Figure 6: Cutting Geometry

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:

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 $\rightarrow$ 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.

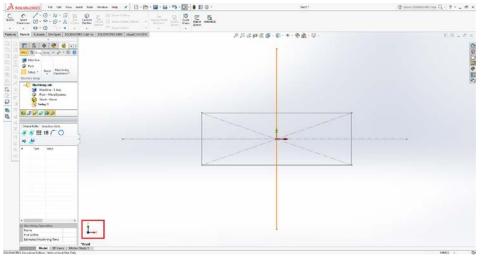


Figure 7: Coordinate system

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.

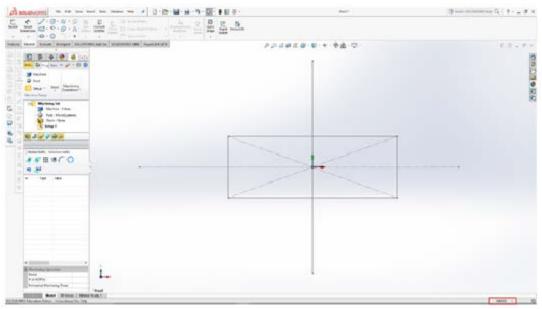


Figure 8: Changing Units

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

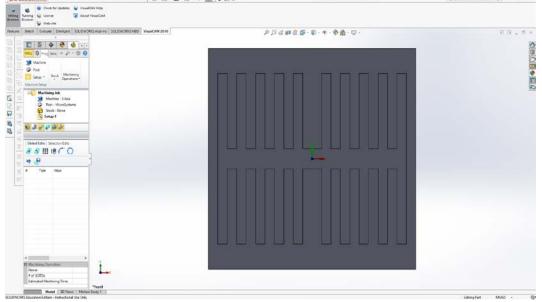


Figure 9: Visual Cam Milling Tab

- 3. In the milling tab select Machine  $\rightarrow$  Align to  $\rightarrow$  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.

Machine Tool Setup
Machine Tool Coordinate System Machine Tool Definition
Manual Definition     Load From File
Machine Type Number of Axes 3 Axis ~
General Parameters
Tool Change Pt: X 0 🔹 Y 0 🔹 Z 0 🔹 😽
Output all coordinates in local Setup Coordinate System
Translational Limits
Max: X 5000 + Y 5000 + Z 5000 + R
4th Axis (Primary Axis) Parameters
Rotary Center: X 0 🗘 Y 0 🗘 Z 0 🗘 🎝
Rotary Axis:   +X ·· +Y ·· +Z ·· Z
Rotary Axis Angle Limit 🗸 🗸
Rotary Axis Limits (In Degrees) Min 0 🔺 Max 360 🔺
5th Axis (Secondary Axis ) Parameters
Rotary Center: X 0 + Y 0 + Z 0 + K
Rotary Axis: O +X O -X O +Y O +Z O -Z
Rotary Axis Angle Limit 🗸
Rotary Axis Limits (In Degrees) Min 0 🗘 Max 360 🗘
Gage Length (For Machines with a Rotary Head) : 10
OK Cancel Help

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.

Set Post-Processor Options	$\times$
Set Post-Processor Options	
Select Post Processor Current Post Processor: MicroSystems Clarent Post Processor files are located: -Soft Corporation/VisualCAM 2016 for SolidWorks\Posts\Mill	
Program to send posted file to	
Posted File Naming Conventions When a Machining Operation(s) is selected for Posting, use Part File Name+Machining Operation Name When a Setup is selected for Posting, use Part File Name+Setup Name V	
When posting all in file, use Part File Name	
Posted File Extension: .nc   Add New  Show Output Dialog when Post-Processing	
0K Cancel Help	

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

Box Stock	$\times$
Stock Geometry	
0	
0	
Height (H)	
Width (W)	
<sup>9</sup> Length (L)	
Corner Coordinates	
Xc •1.35 ♀ Yc •3.00128 ♀ Zc 0 ♀ &	
Dimensions	
L 2.7 • W 6.00256 • H 0.55	
Copy Model Bounding Box	
Ignore Wireframe Geometry in Bounds Computation	
OK Cancel Help	_
Caricei Nep	

Figure 12: Stock Dimensions

7. Select Align  $\rightarrow$  set Z alignment to bottom and XY Alignment to center and hit ok.

Align Part and/or Stock	Geometry		×
Align Stock about Part			
		1	
ZAlignment			
() Тор	<ul> <li>Center</li> </ul>	Bottom	
-XY Alignment			
O North West	🔿 North	O North East	
⊖ Mid-West	Center	⊖ Mid-East	
O South West	🔾 South	O South East	
Ignore Wireframe	Geometry in F	'art Bounds Computati	on
	ОК	Cancel H	elp

Figure 13: Aligning Part to Stock

8. Select Materials under the stock tab  $\rightarrow$  select the material and hit ok. Note: In this case the selected stock is polycarbonate plastic.

Select Stock Material	×
Choose Stock Material	
Material File	
FeedsSpeedsDataINCH.xml ~	
Material	
PLASTIC - POLYCARBONATE 🗸 🗸	
Material Texture	
Source folder for material files:	
Soft Corporation VisualCAM 2016 for SolidWorks Waterials	
OK Cancel Help	

Figure 14: Selecting Stock Materials

 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.

Work Zero	×
Set Work Zero	
*	
<ul> <li>Pick          <ul> <li>Set to Stock Box</li> <li>Set to Part Box</li> </ul> </li> <li>Zero Face</li> </ul>	
Highest Z O Mid Z O Lowest Z	
Zero Position	
○ North West ○ North ○ North East ○ West ○ Center ○ East	
South West     South     South East	
x -25 ↓ y -25 ↓ z 10 ↓ k	
Ignore Wireframe Geometry in Part Bounds Computation	
grote wrienanie deoniedy in Part Bounds Computation	
Output Work Offset	
Work Offset Register Number: 54	
Generate Cancel Save Help	)
Figure 15: Zero Coordinates	

10. To make channels in the stock select the Program tag  $\rightarrow$  2-axix  $\rightarrow$  pocketing and this window should pop up.

Cut Parameters	Cut l	evels	Entry/	Exit	Advanced	Cut Paramet	ers Sorting
Control Geome	etry	Tool		Fee	ds & Speeds	Clea	arance Plane
Part Regions	Avoid R	_	Start Poi	nts			
<		griegion		>	FPar	t/Containme	nt Region
🔼 Move	e U p	🔽 Mo	ive Dow	n			
🗙 Remo	ve All	🗙 Rem	ove Acti	ve			
😽 Sele	ect Curve	/Edge R	egions				
k} s≀	elect Flat	Area Reg	gions				
R Sel	ect Pre-D	efined R	egions				

Figure 16: Defining Cutting Path

11. Select the curve / Edge Regions  $\rightarrow$  select the outer edges of the hole as shown below.

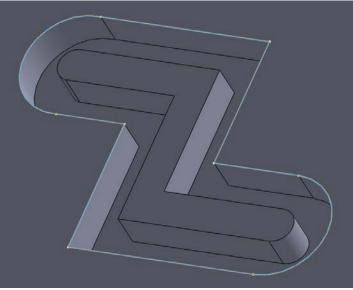


Figure 17: Picking the Cutting Edge

12. Select Cut Parameters tab  $\rightarrow$  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.

Cut Parameters       Cut Levels       Entry/Exit       Advanced Cut Parameters       Sorting         Global Parameters       Tolerance:       0.001       Image: Compensation:       Region       Toolpath         Stock:       0.0025       Image: Compensation:       AUTO/NONE       Image: Compensation:       Auto/None         Cut Pattern       Image: Cut Diffset Spiral       Linear       Circular Spiral       Radial         Cut Direction       Image: Compensation:       Image: Cut Direction       Image: Cut Direction       Image: Cut Direction         Climb (Down Cut)       Conventional (Up Cut)       Image: Cut Direction       Image: Cut Direction       Image: Cut Direction         Start Point       Image: Cut Direction       Image: Cut Direction       Image: Cut Direction       Image: Cut Direction         Start Point       Image: Cut Direction       Image: Cut Direction       Image: Cut Direction       Image: Cut Direction         Start Point       Image: Cut Direction       Image: Cut Direction       Image: Cut Direction       Image: Cut Direction         Start Point       Image: Cut Direction       Image: Cut Direction       Image: Cut Direction       Image: Cut Direction         Image: Cut Direction       Image: Cut Direction       Image: Cut Direction       Image: Cut Direction       Image: Cut Direction </th <th>Control Geome</th> <th>ry Tool</th> <th>Feed</th> <th>ls &amp; Speeds</th> <th>Clearance F</th> <th>lane</th>	Control Geome	ry Tool	Feed	ls & Speeds	Clearance F	lane
Tolerance:       0.001         Stock:       0.0025         Compensation:       AUTO/NONE         Cut Pattern       Image: Compensation:         Image: Diffset Offset Spiral       Linear         Cut Direction       Circular Spiral         Cut Direction       Climb (Down Cut)         Conventional (Up Cut)       Image: Compensation:         Image: Distance       Image: Compensation:         Stepover Distance       Image: Compensation:         Image: Distance       Image: Cleanup Pass	Cut Parameters	Cut Levels B	Entry/Exit	Advanced Cut	Parameters	Sorting
Tolerance:       0.001         Stock:       0.0025         Compensation:       AUTO/NONE         Cut Pattern       Image: Stock         Image: Olifset Olifset Spiral       Linear         Cut Pattern       Circular Spiral         Image: Olimb (Down Cut)       Conventional (Up Cut)         Image: Onventional (Up Cut)       Image: Olimb (Down Cut)         Image: Olimb (Down Cut)       Image: Olimb (Down Cut)         Image:	Global Param	neters		Region		
Stock: 0.0025 Compensation: AUTD/NDNE Cut Pattern Olffset Offset Spiral Olinear Olicular Spiral Radial Cut Direction Olimb (Down Cut) Onventional (Up Cut) Mixed Start Point Inside Outside Stepover Distance Stepover Distance Stepover Distance Olicular Spiral Cut Direction Olicular Spiral Cut Direction Olicular Spiral Olicular Spira	Tolerand				<u>_</u> ↑	
Compensation:       AUTO/NONE         Cut Pattern <ul> <li>Offset</li> <li>Offset Spiral</li> <li>Linear</li> <li>Circular Spiral</li> <li>Radial</li> <li>Cut Direction</li> <li>Climb (Down Cut)</li> <li>Conventional (Up Cut)</li> <li>Mixed</li> <li>Start Point</li> <li>Outside</li> <li>Start Point</li> <li>Start Point</li> <li>Dutside</li> <li>Start Point</li> <li>Distance</li> <li>Cut Start Cenup Loops</li> <li>None</li> <li>Inside</li> <li>Outside</li> <li>Comer Cleanup Loops</li> <li>Cleanup Pass</li> <li>Cleanup Pass</li> </ul>	Stoc	k: 0.0025		the -		
Cut Pattern O Difset ○ Difset Spiral ○ Linear ○ Circular Spiral ○ Radial Cut Direction Climb [Down Cut] O Conventional (Up Cut) Mixed Start Point Start Point Start Point Start Point Start Point Distance X Tool Dia. Distance Correr Cleanup Loops None ○ Inside ○ Dutside Cleanup Pass	Compensatio		~	8		
<ul> <li>Offset Offset Spiral OLinear OCircular Spiral ORadial</li> <li>Cut Direction</li> <li>Climb (Down Cut)</li> <li>Conventional (Up Cut)</li> <li>Mixed</li> <li>Start Point</li> <li>Inside OLutside</li> <li>Stepover Distance</li> <li>% Tool Dia.</li> <li>Distance</li> <li>Distance</li> <li>Correr Cleanup Loops</li> <li>None Olnside OLutside</li> <li>Cleanup Pass</li> </ul>		ACTOMOTE	Ť	->   ← St	ock	
<ul> <li>Offset Offset Spiral OLinear OLincular Spiral Radial</li> <li>Cut Direction</li> <li>Climb (Down Cut)</li> <li>Conventional (Up Cut)</li> <li>Mixed</li> <li>Start Point</li> <li>Inside OLutside</li> <li>Stepover Distance</li> <li>X Tool Dia.</li> <li>Distance</li> <li>Distance</li> <li>Correr Cleanup Loops</li> <li>None Olnside OLutside</li> <li>Cleanup Pass</li> </ul>						
Cut Direction Climb (Down Cut) Conventional (Up Cut) Mixed Start Point Inside Dutside Stepover Distance X Tool Dia. Distance Correr Cleanup Loops None None Inside Outside Climb (Down Cut) Conventional (Up Cut) Cut) Cut Cut Cut Cut Cut Cut Cut Cut Cut Cut Cut Cut Cut Cut Cut	Cut Pattern					
○ Climb (Down Cut)       ○ Conventional (Up Cut)         ⓒ Mixed       ○ Inside         Start Point       ○ Inside         ⓒ Inside       ○ Iutside         Stepover Distance       ○ Iutside         ○ Distance       0.125         ○ None       ○ Inside         ○ None       ○ Inside	Offset	Offset Spiral	◯ Linear	🔿 Circular S	piral 🔿 Radi	ial
○ Climb (Down Cut)       ○ Conventional (Up Cut)         ③ Mixed       ○ Inside         Start Point       ○ Inside         ③ Inside       ○ Intside         Stepover Distance       ○ Inside         ③ X Tool Dia.       ○ Inside         ○ Distance       0.125         ○ None       ○ Inside         ③ None       ○ Inside						
Conventional (Up Cut) Mixed Start Point Inside Dutside Stepover Distance ③ % Tool Dia. Distance 0.125 Correr Cleanup Loops None ○ Inside ○ Dutside Cleanup Pass	Cut Direc	tion				
Mixed   Start Point   Inside   Outside     Stepover Distance   X Tool Dia.   Distance   0.125     He Stepover Distance     Corner Cleanup Loops     None     Inside     Outside     Cleanup Pass	🔿 Climb	(Down Cut)				
Start Point         ● Inside       O Dutside         Stepover Distance         ● % Tool Dia.         ○ Distance         0.125         ● Kone         ● None         ● Inside         ● Outside	⊖ Conve	entional (Up Cut)				
© Inside ○ Dutside Stepover Distance © % Tool Dia. ○ Distance 0.125 Corner Cleanup Loops © None ○ Inside ○ Dutside □ Cleanup Pass	Mixed					
Stepover Distance						
% Tool Dia.     Distance     0.125     Ormer Cleanup Loops     Onne ○ Inside ○ Dutside	<li>Inside</li>	<ul> <li>Outside</li> </ul>	e			
O Distance 0.125 ↔ H• Stepover Distance Ormer Cleanup Loops	Stepover	Distance		+	-	
O Distance 0.125 ↔ + + Stepover Distance Corner Cleanup Loops Cleanup Pass	© ° Tor	J Dia 25		nine 🧹		
Corner Cleanup Loops ● None ○ Inside ○ Dutside □ Cleanup Pass	● ~ 100			HH		
None     Inside     Outside     Cleanup Pass	ODista	0.123		-> + Stepo	ver Distance	
None      Inside      Dutside     Cleanup Pass	🔿 Distar					
Always Keep Tool Down		eanup Loops				
	Corner Cl		Outside	Cleanup Pass		
	Corner Cl		Outside	Cleanup Pass		
	Corner Cl		Outside	Cleanup Pass		
	Corner Cl		Outside	Cleanup Pass		

Figure 18: Defining Tolerances

13. Select the Tools tab  $\rightarrow$  select the type of tool for the job  $\rightarrow$  enter in the dimensions for the tool  $\rightarrow$  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 <a href="http://pmtnow.com/end-mill/Micron">http://pmtnow.com/end-mill/Micron</a>.

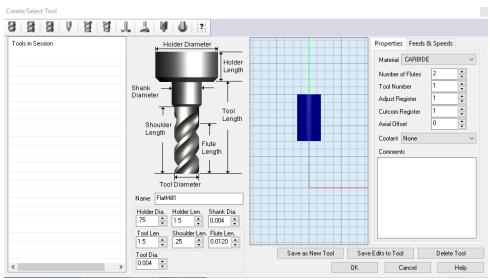


Figure 19: Defining Cutting Tool

14. Select cut levels  $\rightarrow$  Make sure location of cut geometry is from the top  $\rightarrow$  ensure cut depth control is specify on part and set the cutting level.

2 1/2 Axis Pocketing			×
Control Geometry Tool Cut Parameters Cut Levels Entry/I	Feeds & Speeds Exit Advanced Cut	Clearance Parameters	Plane Sorting
Location of Cut Geometry At Top At Bottom Pick Top Cut Depth Control Total Cut Depth: Bough Depth: 0.007874 0 0 0 0 0 0 0 0 0 0 0 0 0	Rough Depth Total Depth Finish Depth	Geometry at T Foot Teptt Finish Depth	h h/Cut
Rough Depth/Cut: 0.0078741 ▼	Finish Depth/Cut		
Clear Island Tops	Use 3D Model to E	etect Depth)	
Cut Levels Ordering		5 <u>6</u> 7	
Generate	Cancel	Gave	Help

Figure 20: Cutting Levels

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

Cut Parameters Cut Le	vels En	try/Exit	Advanced Cul	Parameters	Sorting
Control Geometry	Tool		s & Speeds	Clearance	
Clearance Plane Definitio Automatic Part Max Z + Dist Stock Max Z + Dist Absolute Z Value	0.25		F	s Stock M	ax Z Part Max Z
Cut Transfer Method Skim Skim Clearance (C) © Clearance Plane	0	1	Clearand	ce Plane	
			Part M	ax Z	

Figure 21: Clearance Plane

- 17. Once all parameters are set hit generate
- 18. To see the tool path under simulation  $\rightarrow$  select play Note: It should show you your tool path and how much material is taking off per step.

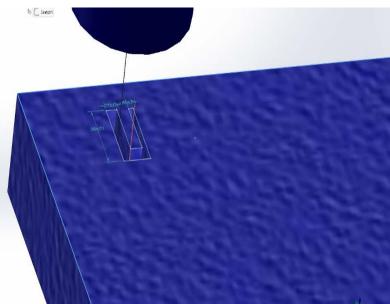


Figure 22: Cutting Simulation

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 <a href="https://nraynaud.github.io/webgcode/">https://nraynaud.github.io/webgcode/</a> or <a href="https://nraynaud.github.g

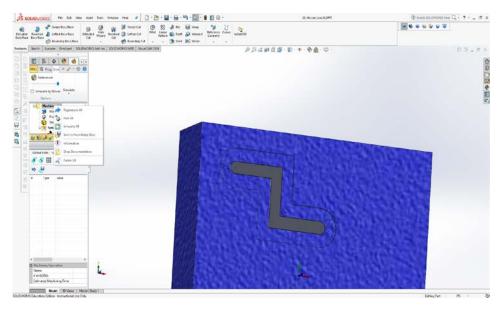


Figure 23: Creating Pose Process

#### SolidWorks G code $\rightarrow$ 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

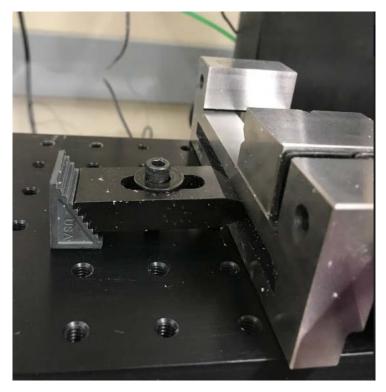


Figure 25: Vice Grip

3 On the computer screen launch the Mach3 CNC Program, the following screen will appear.

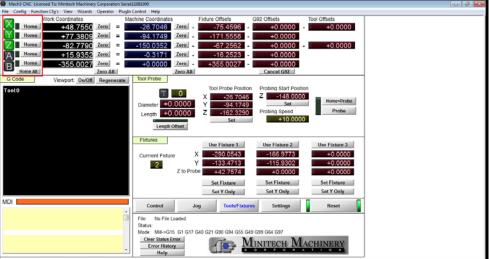


Figure 26: Homing Position

- 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.

Mach3 CNC Licensed To: Minitech Machinery Corporation	n Seriak11081000	
File Config Function Cfg's View Wizards Operator	Plugin Control Help	
Work Coordinates	Machine Coordinates Fixture Offsets G92 Offsets Tool Offsets	
Home +292.1962 Zero =	+0.0000 Zero292.1962 - +0.0000 - +0.0000	
Home +132.3638 Zero =	+0.0000 Zero132.3638 - +0.0000	
Z Home +202,1340 Zero =	+0.0000 Zero202.1340 - +0.0000 - +0.0000	
A Home +0.0000 Zero =	+0.0000 Zero - +0.0000 - +0.0000	
B Home +0.0000 Zero =		
Home All		
G Code Viewport: On/Off Regener	rate Jog Control Jog Buttons	
Toel:0	Continuous         Y+         Z+         A+         B+           Step         Jog Feedrate         Jog Feedrate         Jog DRO's         X         +0.0000         Go To         Go To XY         Z         A-         B-           100%         50%         10%         X         +0.0000         Go To         Go To XY         Z         +0.0000         Go To Safe Z.B.XY         A         HO         Go To Safe Z.B.XY         A         HO         HO	
MDI.	Control Jog Tools/Fixtures Settings Reset	
	File: No File Loaded. Status: ReConfiguration Estop	
	Mode: Mill->G15 G80 G17 G40 G21 G90 G94 G54 G49 G99 G64 G97	
	Chear Status Error Error History Help	

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.

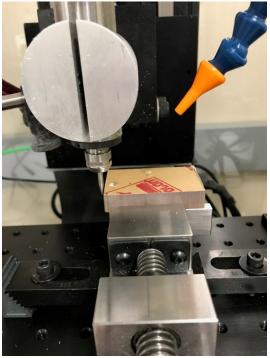


Figure 28: Zeroing Out the X and Y

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.

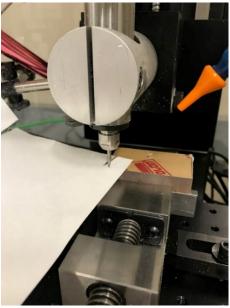


Figure 29: Zeroing Out the Z

- 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  $\rightarrow$  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.

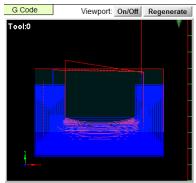


Figure 30: G Code Cutting Path

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.

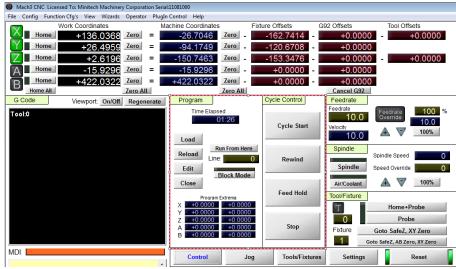


Figure 31: Controlling the G Code

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.

MDI:	
(2 1/2 Axis Pocketing) N2(Tool Diameter = 0.004 Length = 1.5 )	-
N3G54	_
N4G20T1M6 N5 S10000M3	
N6G90G0X2.1755Y0.5487	
N7G43Z0.25H1	
N8 G1Z0.035 F29.334	

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.

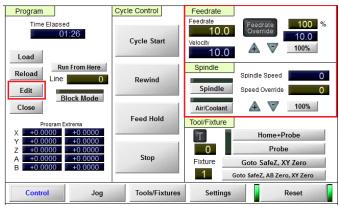


Figure 33: Changing Spindle and Feed Speeds

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.

MDI:	
N3G54	
N4G20T1M6	
N5 S10000M3	
N6G90G0X2.1755Y0.5487	
N7G43Z0.25H1	
N8 G1Z0.035 F29.334	
N9 Z0.01 F14.667	
N10 Y0.5755Z0.0053 F11.	-

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-Cod	e 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 <a href="http://www.onsrud.com/xdoc/FeedSpeeds">http://www.onsrud.com/xdoc/FeedSpeeds</a>

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"
				**Recomme	nded 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"

## **Chip Load Chart**

#### **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	Speed S.F.M. Under	Materials	Speed S.F.M. Over				ed (Inc Id Mill I		· ·			Recommendation for
Soft Grades	32 HRC	Hard Grades	32 HRC	1/8"	3/16'	1/4"	3/8"	1/2"	5/8"	3/4"	1"	Coated Tools
		NONFERR	OUS MATERIA	LS								
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
<b>Copper</b> 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
Magnesium De-CAt, Extruded	1000 min	Nickel Silver, Beryllium Copper, Oxygen-Free Copper	700-1000	.001	.002	.002	.003	.004	.006	.008	.009	TIAIN add 60% on SFM
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	Speed S.F.M. Under	Materials	Speed S.F.M. Over					h/Toot Diamet			<b>Recommendation for</b>	
Soft Grades	32 HRC	Hard Grades	32 HRC	1	3/16"	1/4"	3/8"	1/2"	5/8"	3/4"	1"	Coated Tools
				:	STEEL	s						
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	<u>On Steels &gt;45 HRC on Roughing use Coolant</u> <u>on Finishing Run Dry</u> 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	<b>Speed</b> S.F.M. Under	Materials	Feed (Inch/Tooth) End Mill Diameter								<b>Recommendation for</b>	
Soft Grades	32 HRC	Hard Grades			HRC 1/8" 3/16" 1/4" 3/8" 1/2" 5/8" 3/4" 1"		Coated Tools					
	CAST IRON											
Ductile Cast Iron	100-400	Ductile Cast Iron	80-140	.0005	.001	.0015	.002	.002	.003	.004	.005	<u>Always use coolant</u> 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	Speed S.F.M. Under	Materials	Speed S.F.M. Over				d Inch/ Mill Di		<b>Recommendation for</b>			
Soft Grades	32 HRC	Hard Grades	32 HRC	1/8"	3/16"	1/4"	3/8"	1/2"	5/8"	3/4"	1"	Coated Tools
				TITAN	IUM							
<b>Titanium Alloys</b> 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	<u>Always use coolant</u> TiCN add 40% on SFM TIAIN add 60%-100% n SFM

Materials	Speed S.F.M. Under	Materials	Speed S.F.M. Over				d (Inch/ Mill Dia		<b>Recommendation for</b>			
Soft Grades	32 HRC	Hard Grades	32 HRC	1/8"	3/16"	1/4"	3/8"	1/2"	5/8"	3/4"	1"	Coated Tools
			ST	AINLES	SS STEE	L						
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	
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	TiCN add 40% on SFM TIAIN add 60%-100% on SFM
Martensitic 403, 410, 416	100-250	420, 430F, 440C, 446	100-175	.0001	.0002	.0005	.0005	.001	.001	.003	.004	

Materials	Speed S.F.M. Under	Materials	Speed S.F.M. Over				ed (Incł d Mill D					Recommendation for
Soft Grades	32 HRC	Hard Grades	32 HRC	1/8"	3/16"	1/4"	3/8"	1/2"	5/8"	3/4"	1"	Coated Tools
			HIGH TE	MP ALL	OYS							
<b>Cobalt Base</b> 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	
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	TIAIN Coating is required add 50%-80% in SFM
<b>Iron Base</b> 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