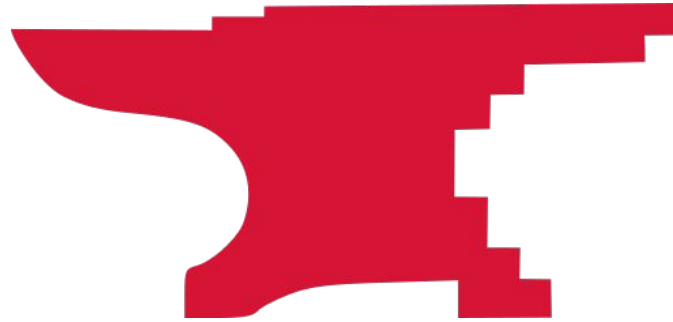


Big Red CNC Training



Outcomes

At the end of the course, students should:

1. Understand the basics of operating the Big Red CNC machine safely.
2. Understand the basics of feeds, speeds, cut depths and endmill selection.
- *3. Demonstrate turning on, jogging, referencing, and running a toolpath on the CNC machine safely.

Priorities

1. Don't hurt yourself.
Risk = Low, Impact = Medium
2. Don't break the machine.
Risk = Medium, Impact = High
3. Don't hurt your end mill or project
Risk = Medium, Impact = Medium

X, Y, Z

On Big Red, facing the machine:

X = left to right

Y = front to back

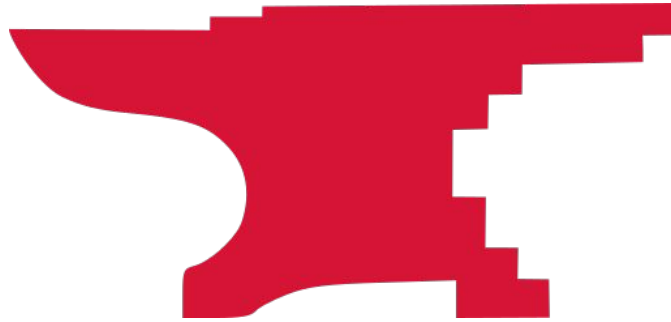
Z = up and down

Important!!!

1. Stay with the machine while it's cutting and be ready to stop the machine if something goes wrong.
2. Always have the Emergency Stop button in hand. Be careful not to drop the Emergency Stop button because it can trigger when it hits the floor.
3. Before jogging the machine, double check that the endmill height is high enough to clear anything on the table.
4. Make sure your toolpaths aren't going to accidentally cut something you don't want cut (clamps!). Leave clearance on your paths, set your Z clearance height and starting position height correctly.
5. ALWAYS use a spoil board if you are going to cut through your chosen material. If you cut into the MDF machine bed, you have to purchase a new bed, and you have to sit there while it recuts the holes for the clamps.
6. Do not put anything on the rails. Be careful of anything you put on the bed that you don't want cut or knocked over.



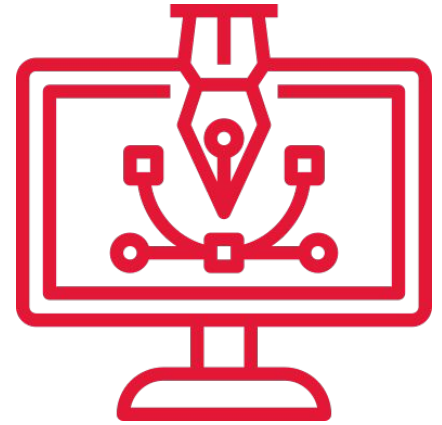
Four Components



1. CAD
2. CAM
3. GCode
4. Controller

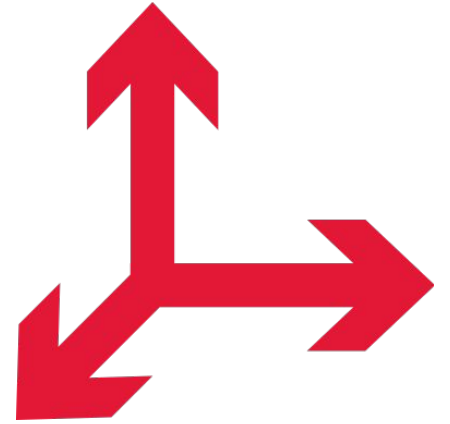
CAD

CAD is Computer Aided Design - it's the software that you use to make/draw/design the thing you want to cut. If you're doing 2.5 cutting, you can draw in Adobe Illustrator, Corel Draw, Inkscape, VCarve and others. If you're doing 2.5d or 3d projects, you can use more complex programs like Fusion360 or other CAD software packages.



CAM

CAM is Computer Aided Manufacturing - it's the software you use to generate the code that tells the machine how fast to spin the end mill, how fast to move the machine and where to move. For Big Red, that is typically VCarve or Fusion360.



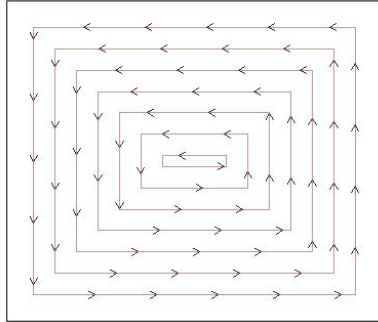
GCode

GCode is the computer language that tells the machine where to start, how fast to spin the spindle, where to move, how fast, how high to move between passes etc.



GCode

GCode is Basically A Text Based Programming Language With Its Own Syntax:



G0 Z4.0 = Raise the spindle Z to 4.0 (Safe Height)
S12000, M03 = Start spinning at 12,000 RPM
G04 P4000 = Dwell/Pause for 4 seconds
G0 X0, Y0 = Rapid Move to X0 Y0
G0 X10, Y10 = Rapid Move to X10 Y10
G0 Z0.25 = Rapid Move to Z 0.25
G1 Z-0.125 = Controlled Move to Z -0.125
G1 X10.2 = Move spindle to X10.2 (in the material!)
G1 Y10.2 = Move spindle to Y10.2

GCode

GCode Has Header Parameters That Impact The Rest of the Job

G20 = Set machine to use inches

G21 = Use Millimeters

G90 = Use Absolute Mode

G91 = Use Incremental Mode

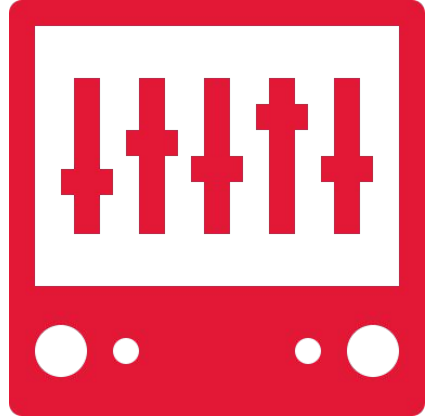
S12000 = Until its changed, spindle speed will be 12,000 every time the spindle is turned on

F100 = Move at 100" per minute anytime G1 is used

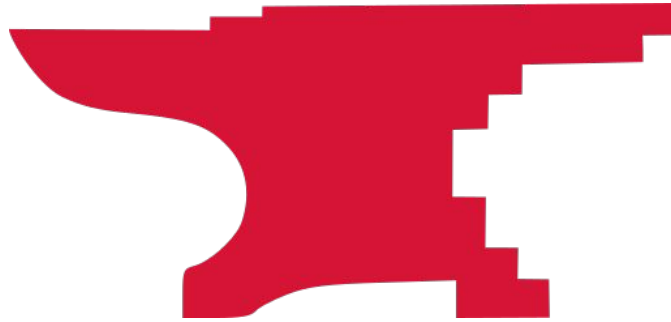


Controller

Controller software runs your GCode. On Big Red this is Mach 4. Pronterface is another which is used on the smaller CNC machine. WinCNC is the program on the EBF machine in Purcellville. Each have their quirks you need to be aware of.



Machine Parts



Hardware

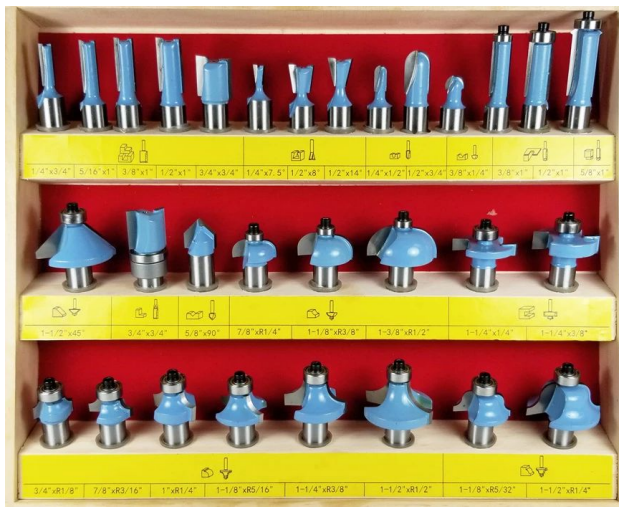
1. Machine Power On/Off
2. Vacuum, Switch and Bag
3. Limit Switches
4. Spindle, End Mills and Collette
5. Computer Software - Mach 4

Parts 1

1. The power switch for the CNC machine is on the wall to the right of the PC. Turn on the grey power strip.
2. The vacuum switch is on the yellow box to the right of the grey power strip. You only need to have the vacuum on when you are cutting. Please check and empty the vacuum bag as necessary.
3. Limit switches are the blue plastic pieces on the ends of the rails and the gantry that stop the machine moving too far in any direction. Note there are no limit switches on the Z axis.

Parts 2

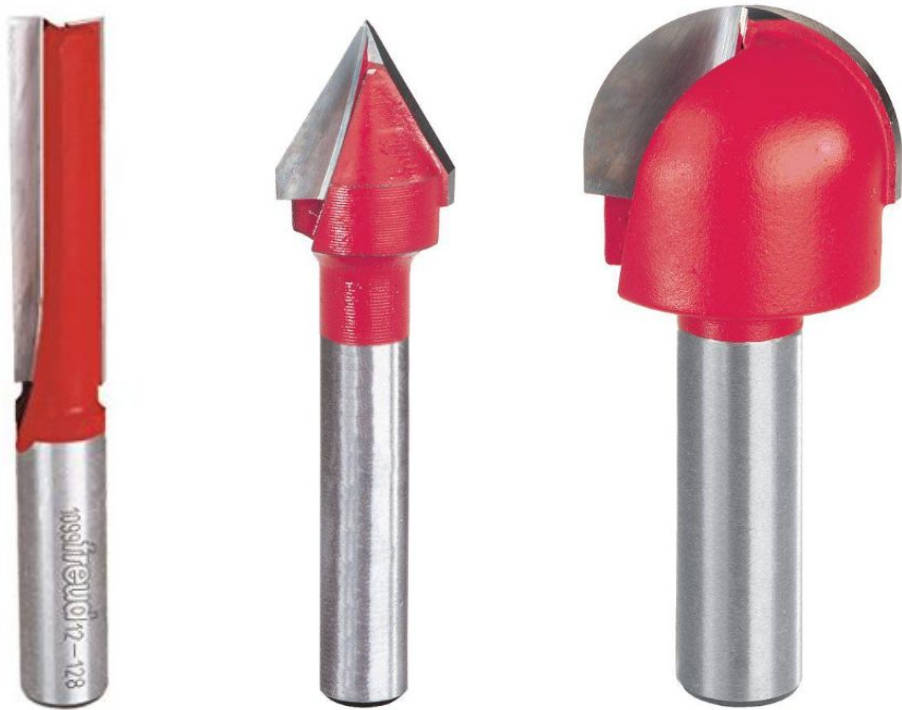
1. The spindle is the part of the CNC machine that spins the endmill. This is not something you will need to manually control or turn on. Spindle speed, on and off are controlled by the GCode you create and run.
2. The collette is the piece that holds your endmill in place. Colletes are inserted into the nut that attaches to the spindle. Always use the right size collette.
3. Mach 4 is the program that you will open on the computer to move the machine, load and run your GCode.



Router bits and endmills come in all shapes and sizes.

There are different coatings, numbers of flutes, cut direction and more.





You can use regular router bits in a CNC machine.

Depending on the brand, the cutting surfaces may be carbide.



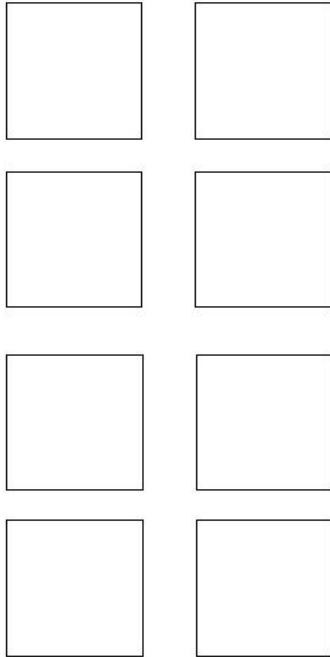
Don't use router bits with bearings on the CNC machines.



Upcut - pulls material up, helps clear chips but can fray the top of your material

Downcut - flatter bottoms, prevents workpiece from lifting, doesn't clear chips as well

Impact of Endmill and Toolpath Types



Square 7

Endmill: .5 ballnose
Path: profile on line
Depth: 0.125 inch
Passes: 1

Square 5

Endmill: .125 straight
Path: profile inside line
Depth: 0.125inch
Passes: 3

Square 3

Endmill: .25 straight
Path: profile inside
Depth: 0.125 inch
Passes: 2

Square 1

Endmill: 60 degree .5 V bit
Path: profile on line
Depth: 0.0675 inch
Passes: 1

Square 8

Endmill: .5 ballnose
Path: pocket
Depth: 0.25 inch
Passes: 2

Square 6

Endmill: .25 straight
Path: profile outside line
Depth: 0.250 inch
Passes: 5

Square 4

Endmill: .25 straight
Path: pocket
Depth: 0.125
Passes: 2

Square 2

Endmill: .25 straight
Path: profile outside line
Depth: 0.0675 inch
Passes: 1

X, Y, Z zero

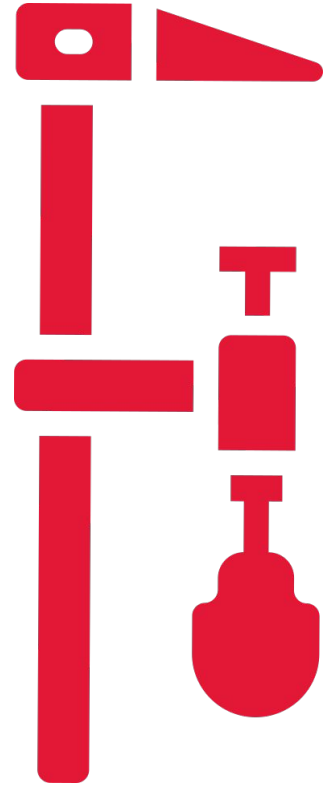
Where is X, Y, Z 0?

1. On Big Red, facing the machine:
 - a. X = left to right
 - b. Y = front to back
 - c. Z = up and down
2. The machine has an absolute X0 and Y0.
3. A project's X, Y, Z zero is manually set for each project.
4. Usually the bottom left corner is X0, Y0. There may be times when you use the center.
5. Z0 should be the top of the material to be cut until you get more experience.
6. Other setups are possible but these are advanced uses.

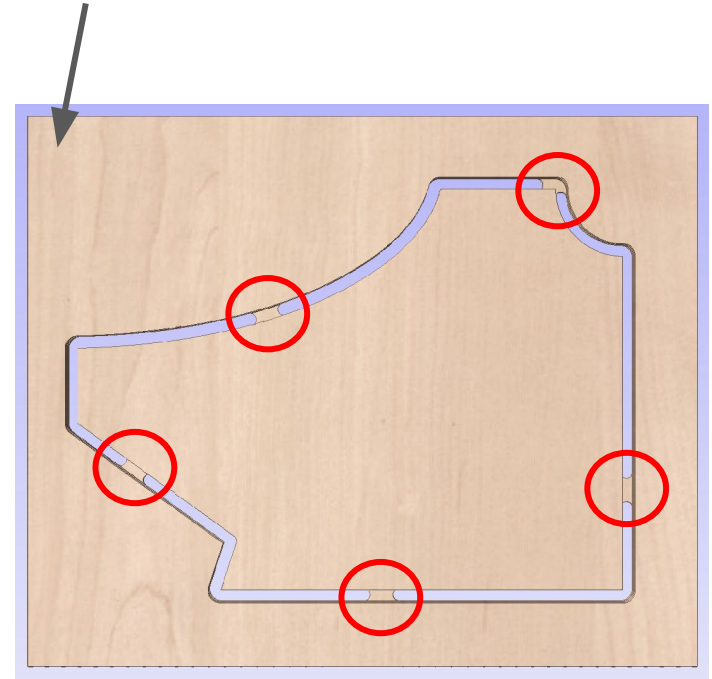
Securing Material



1. Clamp your workpiece down directly to the bed if you are not cutting all the way through.
2. Clamp a spoilboard down and screw the workpiece to the spoilboard if you're cutting through but not cutting pieces out (slots).



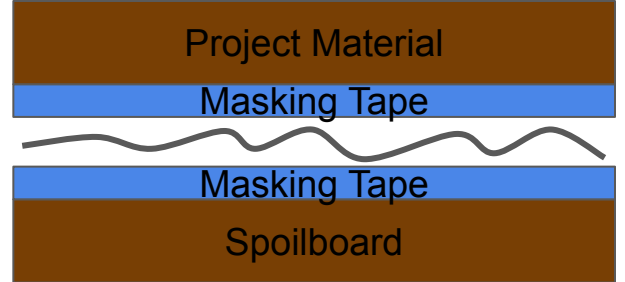
3. Clamp a spoilboard, screw the workpiece to the spoilboard and use tabs to hold your piece in place if cutting all the way through.



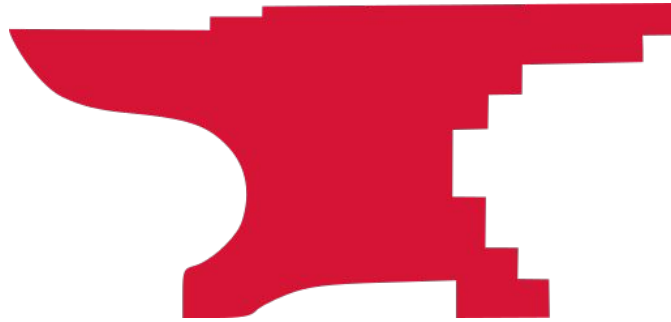
4. Clamp a spoilboard and use double sided tape or the tape and spray glue/superglue method if you're cutting all the way through.

5. Use double sided tape instead of screws or glue.

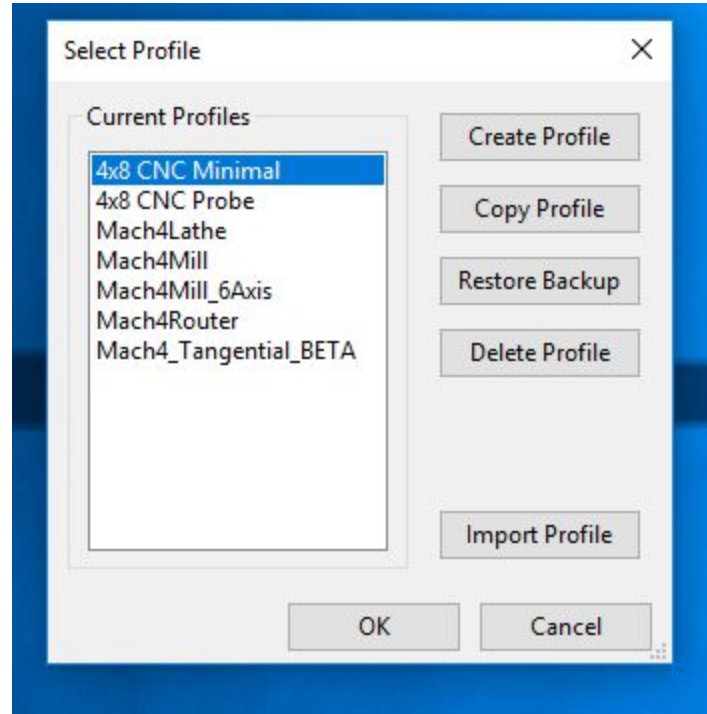
The tape and glue method works when cut-out pieces are small or when you want to avoid sanding the tabs off.



Using the Machine



Mach 4



Interface

Mach 4 - Hobby
File View Configure Diagnostic Wizard Operator Help

Program Run MDI Tool Path Machine Diagnostics

G Code

Axe Control

Ref All Home

Zero X **13.5310**

Zero Y **43.7331**

Zero Z **2.5840**

Zero A **0.0000**

Go to Zero To Go Machine Coordinates Soft Limits

Tool Path

Current File

Load Wizards Load Last Wiz

Limit On

Regen ToolPath Display Mode Jog Follow

Parts Complete: **318** Parts Req.: **0** Total Parts: **0**

Control

File Ops Run Ops Tool Path Ops Jogging

Cycle Start Feed Hold Stop Reset Drains

Edit G Code Set Next Line

Recent File Run From Here

Close G Code Load G Code

Line: **0**

Tool Information

Tool: **13** Tool Chng.

Diameter: **0.0000**

Height: **0.0000**

Tool Height

Remember Return Touch

Feed Rate

RRO% FRO% SRO%

Spindle

Spindle CW

0 RPM

G0 G17 G90 G91.1 G94 G20 G40 G49 G80 G98 G50 G67 G97 G54 G64 G69 G15 G40.1

History PMDX: Mach halted due to Y-- LIMIT switch active Profile: 4x8 CNC Minimal Feed Rate: 0.00

Z Clearance

Before you move the machine in any way, make sure your end mill is raised high enough not to hit anything on the table.

Your CAM software will likely have a setting for the safe clearance height and a Home / Start Position height that the endmill moves to between cuts and at the start and end of the GCode.

Enable

If the this button is flashing Green and says "Enable", you need to click it to enable control of the machine.



Parts Complete: 318 Part

Control

File Ops Run Ops Tool Path Ops Jogging

Cycle Start

Feed Hold

Stop

Reset

Enable

Z+

X-

Z-

Cycle Jog Inc 1.0000

Jog Mode Cont.

G0 G17 G90 G91.1 G94 G20 G40 G49

History Can't jog toward Y+ limit!

Detailed description: This image shows a screenshot of a CNC control interface. At the top, it displays 'Parts Complete: 318 Part'. Below this is a 'Control' panel with several buttons: 'Cycle Start' (green), 'Feed Hold' (yellow), 'Stop' (red), 'Reset' (grey), and 'Enable' (red). A red arrow points to the 'Enable' button. To the right of the control panel is a 'Jogging' panel with buttons for 'Z+', 'X-', 'Z-', 'Cycle Jog Inc', and 'Jog Mode'. A digital readout shows '1.0000' and 'Cont.' with a jog wheel. At the bottom, a status bar shows G-code 'G0 G17 G90 G91.1 G94 G20 G40 G49' and a message 'Can't jog toward Y+ limit!'.

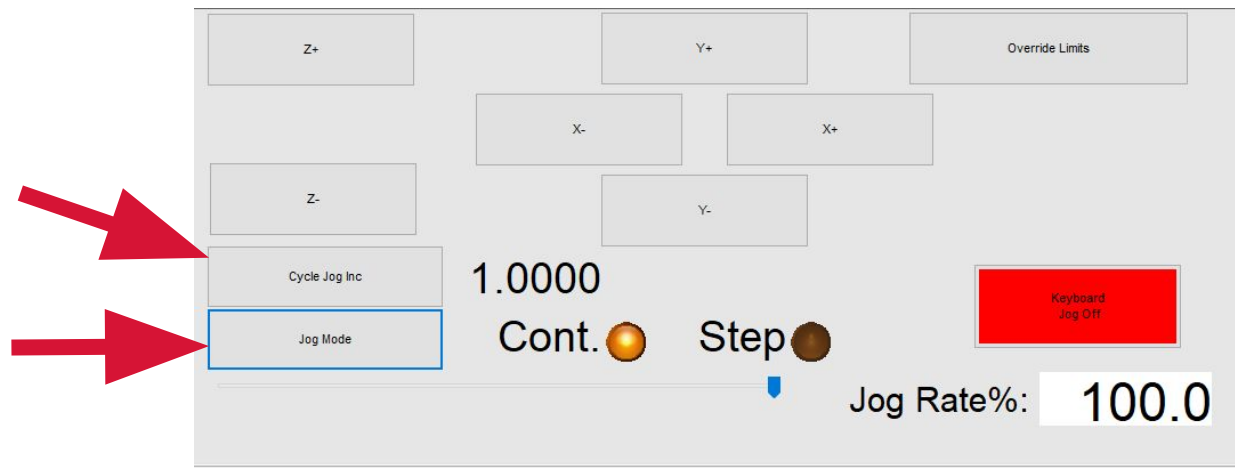
Jogging

The screenshot shows the Mach4 software interface with the following elements:

- Axis Control:** Displays Zero X (1.4307), Zero Y (26.4198), Zero Z (3.8640), and Zero A (0.0000).
- Control Panel:** Includes buttons for Jogging, Cycle Jog Inc, Jog Mode, Jog Rate% (100.0), and Jog Rate (1.0000).
- Tool Information:** Shows Tool: 13, Diameter: 0.0000, Height: 0.0000, and Spindle RPM: 0.
- Feed Rate:** Displays RRO%, FRO%, and SRO%.
- Status Bar:** Shows "Can't jog toward Y+ limit!" and "Profile: 4x8 CNC Minimal Feed Rate: 0.00".

Jogging the machine is moving the machine. In Mach 4, you can jog the machine incrementally, continuously, or by writing GCode manually. Click on the "Jogging" tab indicated by the arrow to reach the jogging buttons.

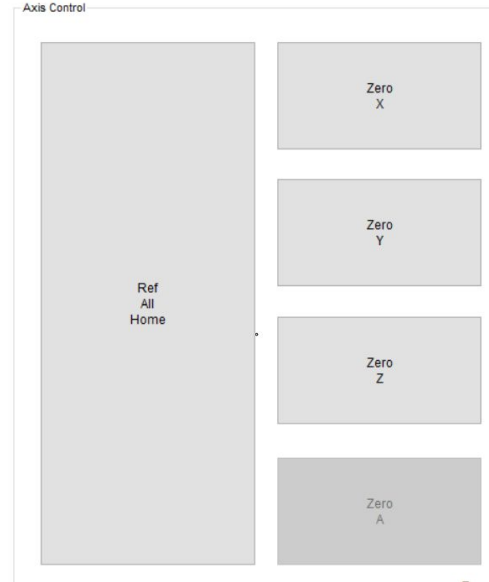
Jogging



1. You'll use the plus and minus buttons to move the machine around prior to starting your GCode.
2. When you first start using the machine, use 'Step' mode. This is changed by clicking the "Jog Mode" button until the circle next to "Step" is lit up. This is safer and less likely that you'll drive the endmill into your workpiece or that you'll hit the limit switches at each end of the machine.
3. Use the "Cycle Jog Inc" button to change how far the machine will move each time you press on of the X, Y and Z plus and minus buttons. 1.000 equals 1 inch and so forth.
4. The X, Y and Z plus and minus buttons are laid out on the page relative to the machine. If you click the X+ the machine moves right.
5. When moving the Z height, be careful you don't move it too high or the spindle carrier will come out. When moving it downwards towards your workpiece, make sure to change the step so that you don't drive the endmill into the workpiece, risking breaking it.
6. Before you do anything else, jog the machine around a little bit to make sure everything is moving correctly.

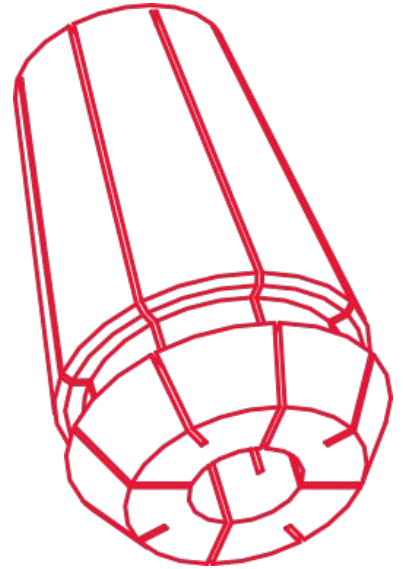
Referencing

1. Referencing the machine tells it where the absolute edges of the machine's movement are
2. Whenever you start a new project on the CNC, you want to first reference the machine in case someone else has racked it.
3. Check Z Clearance before hitting Ref anything.
4. Make sure there is nothing on the bed or the rails that can get hit.
5. Click "Ref All Home". Note that this will not Ref Z because this machine does not have limit switches on Z and the machine knows that.
6. In the event that the machine 'skips' when moving during a cut, the machine no longer correctly knows where the absolute X0 Y0 point is so you will need to re-reference the machine before you start again.



Collet & Collar

1. Use the correct collet size to match the shank of your bit.
2. Screw the collar and collet onto the spindle finger tight first using the wrenches, tighten no more than 1/8th additional turn.
3. In no event should you be straining to tighten the collar and collet.
4. Try to make your endmill stick out the collet as little as you can, bearing in mind that the endmill has to stick out at least as far as the deepest cut you're making. You can't have the endmill stick out 1/2 inch and try to cut 3/4 inch into something. The less stickout you have, the less the endmill deflects/bends during the cut.
5. Don't let the shank stick out of the top of the collet.



Collet & Collar

1. Try to make your endmill stick out the collet as little as you can, bearing in mind that the endmill has to stick out at least as far as the deepest cut you're making. You can't have the endmill stick out 1/2 inch and try to cut 3/4 inch into something. The less stickout you have, the less the endmill deflects/bends during the cut.
2. Don't let the shank stick out of the top of the collet. The first picture is fine, the second picture is too far through the collet.



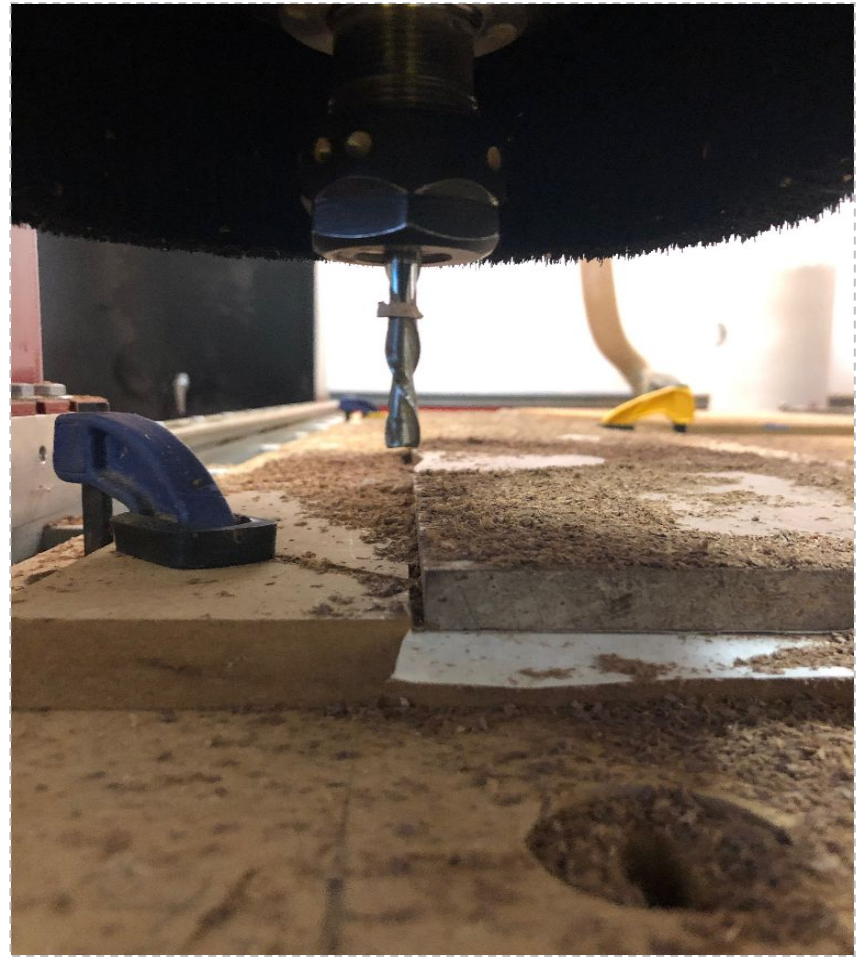
Setting 0,0,0

When you setup your CAM, you set where X0,Y0,Z0 is. The common practice is to set X and Y zero to the bottom left corner of your piece in both the CAM and in your machine setup.

Other setups are possible but outside the scope of this class.

Setting 0,0,0

1. Absent using a probe, you set X and Y zero by sight. Jog the machine incrementally until the center of your endmill lines up with the edges of your piece. You may want to move the endmill down towards the work surface so you aren't trying to estimate while the endmill is too far from the cutting surface.
2. It doesn't matter if you set the X or Y first.
3. Once you are happy that the endmill is in the correct position, click the Zero X or Zero Y button in Mach 4. These buttons are large and in the middle of the screen
4. Once both X and Y are set, move on to Z.



Setting 0,0,0

The screenshot displays the Mach4 software interface. The 'Axis Control' panel is highlighted with a red box, showing buttons for 'Zero X', 'Zero Y', 'Zero Z', and 'Zero A'. To the right, four digital readouts (DROs) show the current coordinates: X=13.5310, Y=43.7331, Z=2.5840, and A=0.0000. The 'G Code' window is empty. The 'Tool Path' window shows a blue coordinate system. The 'Parts Complete' counter is at 318, 'Parts Req.' is at 0, and 'Total Parts' is at 0. The 'Control' panel includes buttons for 'Cycle Start', 'Feed Hold', 'Stop', 'Reset', and 'Trailing'. The 'Tool Information' panel shows 'Tool: 13', 'Diameter: 0.0000', and 'Height: 0.0000'. The 'Feed Rate' panel shows 'RRO%', 'FRO%', and 'SRO%' sliders, and a 'Spindle CW' button with '0 RPM'. The status bar at the bottom shows 'G0 G17 G90 G91.1 G94 G20 G40 G49 G80 G98 G50 G67 G97 G54 G64 G69 G15 G40.1' and a message: 'PMDX: Mach halted due to Y-- LIMIT switch active'. The 'Profile' is set to '4x8 CNC Minimal' and 'Feed Rate' is '0.00'.

Mach4 - Hobby
File View Configure Diagnostic Wizard Operator Help

Program Run MDI Tool Path Machine Diagnostics

G Code

Axis Control

Ref All Home

Zero X
Zero Y
Zero Z
Zero A

13.5310
43.7331
2.5840
0.0000

Tool Path

Current File

Load Wizards Load Last Wiz
Limit On

Regen ToolPath Display Mode Jng Follow

Parts Complete: 318 Parts Req.: 0 Total Parts: 0

Control

File Ops Run Ops Tool Path Ops Jogging

Edit G Code Set Next Line
Recent File Run From Here
Close G Code
Load G Code

Line: 0

Tool Information

Tool: 13 Tool Chng.
Diameter: 0.0000
Height: 0.0000
Tool Height

Remember Return
Touch

Feed Rate

RRO% FRO% SRO%

Spindle CW
0 RPM

G0 G17 G90 G91.1 G94 G20 G40 G49 G80 G98 G50 G67 G97 G54 G64 G69 G15 G40.1

History PMDX: Mach halted due to Y-- LIMIT switch active Profile: 4x8 CNC Minimal Feed Rate: 0.00

Setting 0,0,0

1. Until you are trained to use the probe, or if your project can't be used with the probe, jog the endmill over the highest section of the surface you are going to cut. Most of the time you will be cutting a flat board, but there are always exceptions.
2. Make sure the jog mode is set to incremental, NOT CONTINUOUS when setting Z to lessen the chance that you'll ram the endmill into the workpiece.
3. SLOWLY and at a sensible increment, jog the endmill down towards your work surface until you are just above the highest part of the work surface as outlined above.
4. Put a piece of paper between the endmill and the work surface. Move the paper back and forth slowly under the endmill while lowering the endmill in small increments until the endmill just bites the paper and makes it hard to move.
5. Click the Zero Z button.
6. Raise the Z Height back up to a safe clearance height.

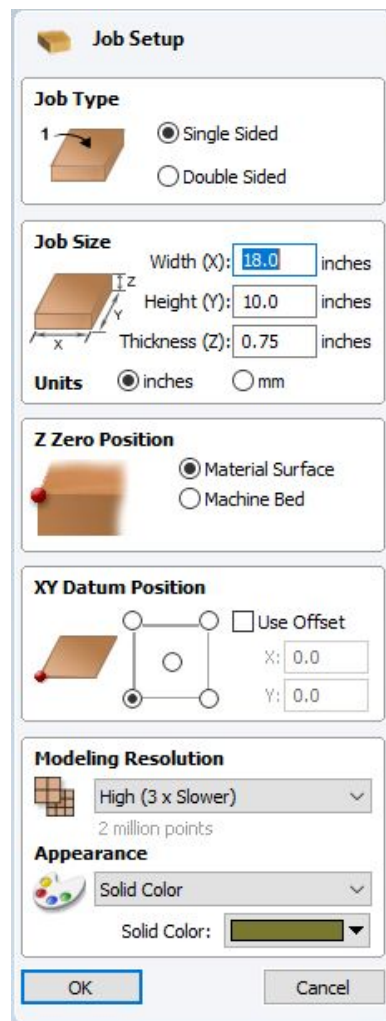
Loading GCode

This class assumes that you have already saved your GCode to run on the machine.

Toolpath Prep

If you are using vCarve, this class assumes that your Job Type is Single Sided, that your Job Size has been set to the size of your workpiece, that the Z Zero Position is set to the Material Surface and that the XY Datum Position is set in the bottom left corner.

In short, there are a lot of things to get right!



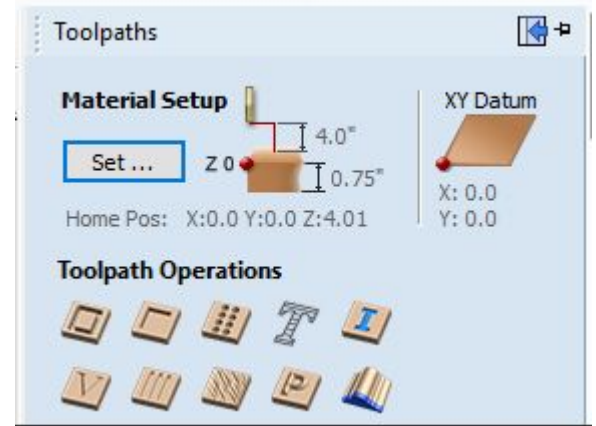
The screenshot shows the 'Job Setup' dialog box with the following settings:

- Job Type:** Single Sided (selected), Double Sided (unselected).
- Job Size:** Width (X): 18.0 inches, Height (Y): 10.0 inches, Thickness (Z): 0.75 inches.
- Units:** inches (selected), mm (unselected).
- Z Zero Position:** Material Surface (selected), Machine Bed (unselected).
- XY Datum Position:** Use Offset (unchecked), X: 0.0, Y: 0.0.
- Modeling Resolution:** High (3 x Slower) (selected), 2 million points.
- Appearance:** Solid Color (selected), Solid Color: [Green color swatch].

Buttons: OK, Cancel.

Z Clearance

Your CAM software will likely have a setting for the safe clearance height and a Home / Start Position height that the endmill moves to between cuts and at the start and end of the GCode. In Vcarve this is set by clicking "Set..." in the Toolpaths menu, then setting the Clearance (Z1) rate to avoid your clamps and the Home / Start Position Z Gap Above Material to a safe clearance height. When you're starting off, use a value of 3-4 inches.



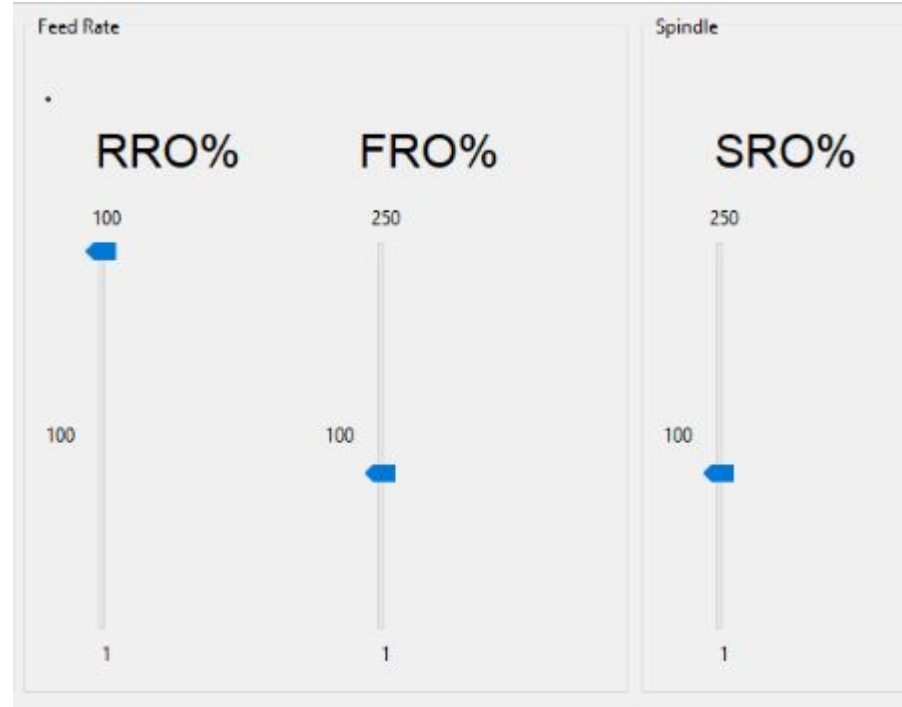
Load GCode

1. Load GCode
2. Review the header
3. Review the image of the toolpaths
4. Move the machine to the edges of your image to make sure that you are always on your workpiece and not hitting any clamps or screws.
5. Make sure to select the top line of your GCode before hitting GO or else the machine will try to start your project partway through without the endmill spinning.



Sliders

Mach4 has sliders that allow you to change the feed and speed rate of the machine while its cutting. Make sure these are set to 100 per the picture before hitting Cycle Start.



Before GO

1. Did you Ref X and Ref Y the machine when you started?
2. Are the X, Y, Z zero set in the right place for the toolpath you're about to cut?
3. Are the feed and speed sliders set to 100%?
4. Is the right endmill in the machine?
5. Is there anything on the rails or bed that could get hit?
6. Is your material secured down?
7. Is the emergency stop button ready?
8. Did you turn on dust collection?
9. Is the first line of the GCode highlighted?



GO!!!

When you're comfortable that everything has been setup correctly, hit Cycle Start

Mach4 - Hobby

File View Configure Diagnostics Wizard Operator Help

Program Run | MDI | ToolPath | Machine Diagnostics

Q Code

Axis Control

Zone X 13.5310

Zone Y 43.7331

Zone Z 2.5840

Zone X 0.0000

Tool Path

Current File

Parts Complete: 318 Parts Req.: 0 Total Parts: 0

Control

File Open Run Ops Tool Path Ops Joggling

Edit G Code Set Feed Line

Recent File Run From Here

Line: 0

Clear G Code Load G Code

Tool Information

Tool: 13 Tool Chng.

Diameter: 0.0000

Height: 0.0000

Tool Height

Ramp Retard Return Touch

Feed Rate

RRO% FRO% SRO%

Spindle CW

0 RPM

G0 G17 G90 G91.1 G94 G20 G40 G49 G80 G98 G50 G67 G97 G54 G64 G69 G15 G40.1

History PMDX: Mach halted due to Y-- LIMIT switch active Profile: 4x8 CNC Minimal Feed Rate: 0.00

During Job

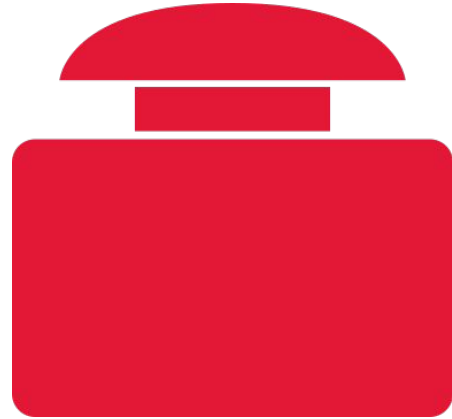
1. Stay with the machine.
2. Be ready to hit the Emergency Stop button if necessary.
3. Listen to the sound of the machine - you'll get used to the sound it should make depending on the material you are cutting. Too high pitched a "scream" or too low a pitched "grind" means there is something wrong with your feeds, speeds, or depth of cut.



Recovery

Recovering from Estop

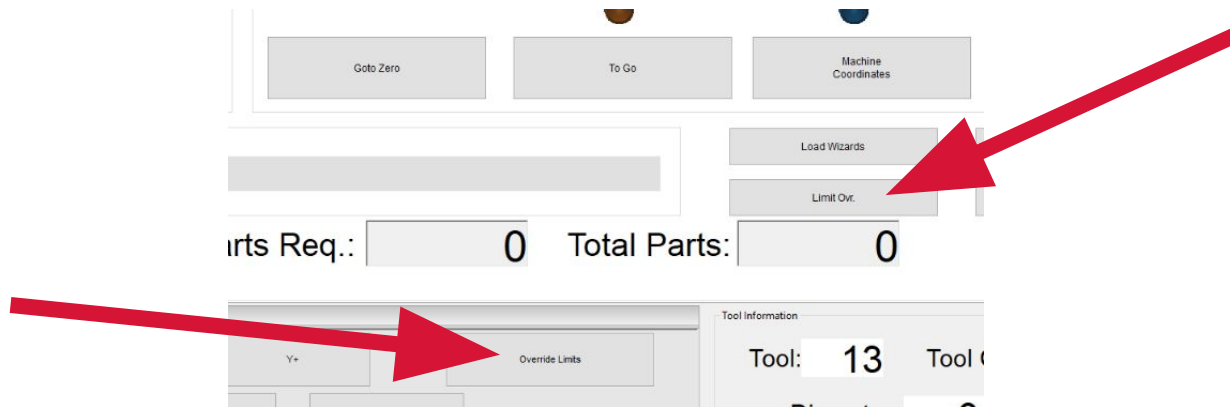
1. Turn the Estop switch to release
2. Press the green flashing "Enable" button
3. If the machine skipped - tried to move X or Y and was stopped because the bit was jammed or didn't move it's Z position correctly (you'll hear a difference) then you will need to re-reference the machine and re X,Y,Z zero before you start again.



Recovery

Hitting a Limit Switch

1. The machine is set up with limit switches which automatically stop the machine moving X or Y too far. If you hit a limit switch while jogging the machine or during an operation, the machine will stop.
2. You will need to press the Override Limits or Limit Ovr. button, move the machine off the limit switch, then press the button again to remove the override.



Going Wrong

1. Hit a clamp, screw, or other object you didn't mean to cut - EStop the machine, check your end mill and replace if necessary, re-reference the machine, start over
2. Windows freezes/updates - the machine will stop if Windows freezes. You may have to start your cut over entirely.
3. Cut depth is too deep - either overall or on a first pass - Estop machine, change the depth of cut in your CAM program, reload the Gcode in Mach 4, don't reset X,Y,Z zero, run GCode
4. Plastic material melts, solidifies and grabs the bit - the goal of cutting is make chips, cutting plastics and polycarbonate usually requires slowing down the speed (8000 rpm) and reducing the depth of cut.
5. Toolpath not cutting in the right place
6. Parts 'jump' off the work surface
7. You hit the limit switches



After Job

1. Wait till the spindle stops spinning before doing anything.
2. Jog the machine out of the way of your material
3. Remove your material
4. Vacuum top, under and around machine
5. Empty dust collection bag if full
6. Put all tools and endmills back on the shelves and pegboard

Feeds and Speeds



Art vs Science

- Setting machine “Feeds and Speeds” is as much art as it is science.



Feeds

How fast the machine moves:

- Too fast and you get deflection of your endmill, reduction in end mill life, racking of the machine, excessive pulling on your workpiece
- Too slow and you can burn your piece, melt it, or just take too long to cut your piece



Speeds

How fast the spindle spins:

- Too fast and you can burn or melt your work or put too much strain on the machine if spinning a heavy/wide end mill
- Too slow and the endmill isn't cutting away enough material as it moves, pulling at the workpiece, racking the machine, adding deflection



Depth

How much material is removed at a time:

- As you get started, err on the conservative side and use shallow cut depths
- The only downside to shallow cut depths is the process takes longer
- Rule of thumb - each cut should be no deeper than half the diameter of your endmill

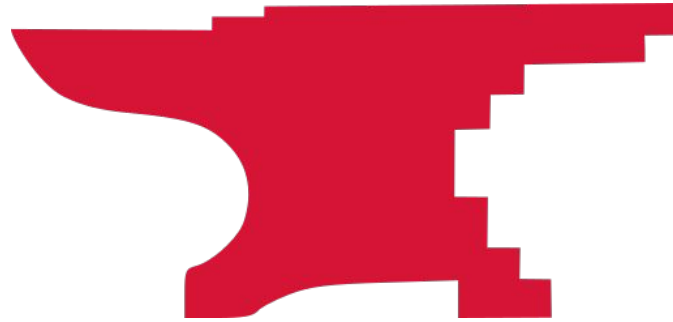


1/8-1/2 Endmill

- If you are using one of the $\frac{1}{4}$ or $\frac{1}{2}$ fluted end mills and cutting a wood product, use 12,000 as the spindle speed and 100 inches per minute as the feed speed. $\frac{1}{4}$ end mill = .125 AT MOST depth per cut. $\frac{1}{2}$ end mill = .250 AT MOST depth per cut.



Wrap Up



Materials

Do and Do Not Cut List

- DO cut
 - Wood, MDF, chipboard, sign foam, Corian
 - Melamine, lexan, acrylic, polycarbonate, aluminum
- DO NOT cut list
 - Metals other than aluminum

Other Tips

If something goes wrong, try not to move your piece or change X,Y,Z zero.

You can do repeatable cuts by putting your material in exactly the same place each time with a jig or fixture.

Practice on a piece you don't care about first.

Important!!!

1. Stay with the machine while it's cutting and be ready to stop the machine if something goes wrong.
2. Always have the Emergency Stop button in hand. Be careful not to drop the Emergency Stop button because it can trigger when it hits the floor.
3. Before jogging the machine, double check that the endmill height is high enough to clear anything on the table.
4. Make sure your toolpaths aren't going to accidentally cut something you don't want cut (clamps!). Leave clearance on your paths, set your Z clearance height and starting position height correctly.
5. ALWAYS use a spoil board if you are going to cut through your chosen material. If you cut into the MDF machine bed, you have to purchase a new bed, and you have to sit there while it recuts the holes for the clamps.
6. Do not put anything on the rails. Be careful of anything you put on the bed that you don't want cut or knocked over.



Check Rides

I'm out of town next week

Will run a test with a friend before I leave

Coordinate with me through Slack (Wednesdays and Mondays starting 7 April for 1-2 hours)

Meet at Leesburg

We'll go through a machine setup and do a small cut

Bring a project (in VCarve or .svg) you have in mind, and we can look at it