

Big Red CNC Training



Outcomes

1. Understand the basics of operating the Big Red CNC machine safely.
2. Demonstrate turning on, jogging, referencing, and running a toolpath on the CNC machine safely.
3. Each student sets up and runs a pre-prepared project.

Assumptions

- You've watched the pre-class videos or already have a knowledge of the concepts in them.
- You want to get started using this machine.
- Your CNC parts are usually one off or low quantity.
- You accept that there may be faster and more robust methods for higher volume production on this machine. (Tool libraries and offsets, production jigs and fixtures, touch-off indicators, and other improved efficiency tools and methods.)

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 zero

- Facing down the machine, X is left to right, Y is front to back and Z is up and down.
- The front left corner of the machine is absolute X0, Y0.
- Usually the front left corner of your workpieces is set as X0, Y0 during the project setup process. There may be times when you use the center.
- Z0 should be the top of the material to be cut until you get more experience.
- Other setups are possible but these are advanced uses.
- Be sure you have the vacuum on when registering the Z at the top of your material.

X0 Y0



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. Double check that the end mill height is high enough to clear any clamps or fixture pieces. Just because you set the Z height to 4" up doesn't mean the end mill is 4" up from the table until you've referenced as the Z during setup.
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.
7. Be sure your tools are securely mounted in in the collets and spindle.



Machine Parts



Hardware

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

Parts 1

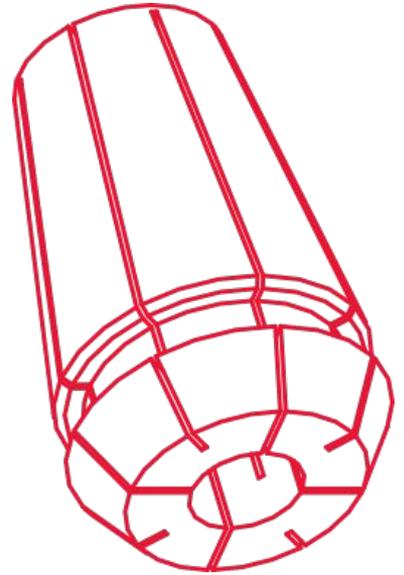
- The power switch for the CNC machine is on the wall to the right of the PC. Turn on the grey power strip.
- The vacuum switch is on the yellow box to the right of the grey power strip. You only need to have the vacuum on when you are cutting. Please check and empty the vacuum bag as necessary.
- Limit 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

- 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.
- The collet 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 collet.
- Mach 4 is the program that you will open on the computer to move the machine, load and run your GCode.

Collet & Collar

- Use the correct collet size to match the shank of your bit.
- Screw the collar and collet onto the spindle finger tight first using the wrenches, tighten no more than 1/8th additional turn.
- In no event should you be straining to tighten the collar and collet.
- 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.
- Don't let the shank stick out of the top of the collet.

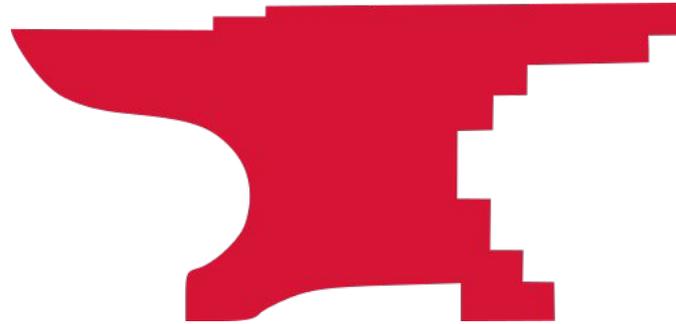


Collet & Collar

- 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.
- 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.

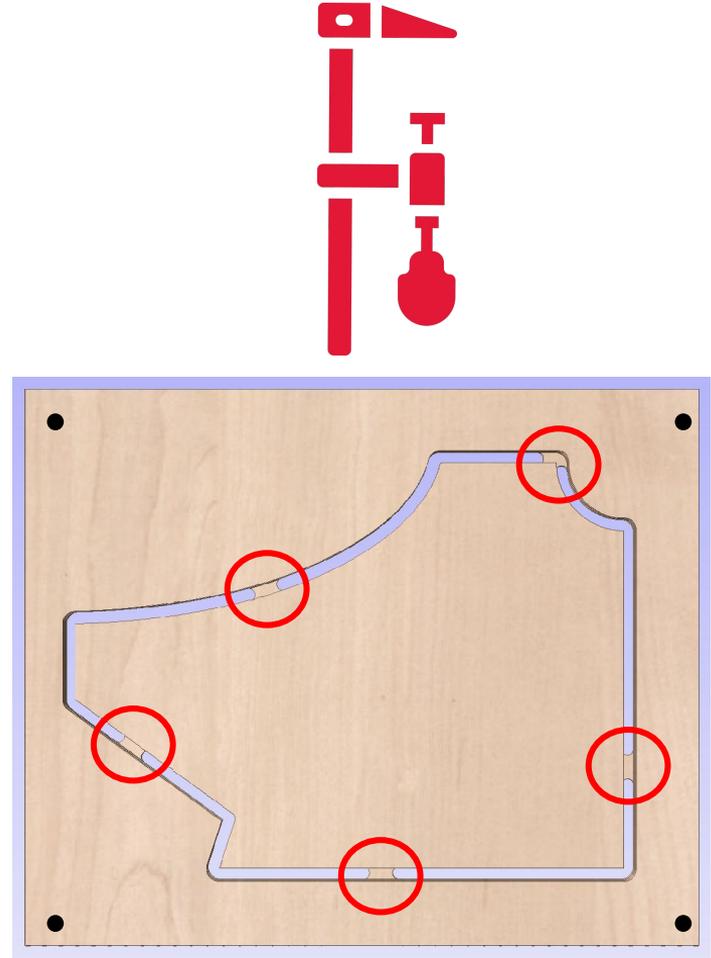


Fixturing - Securing Material



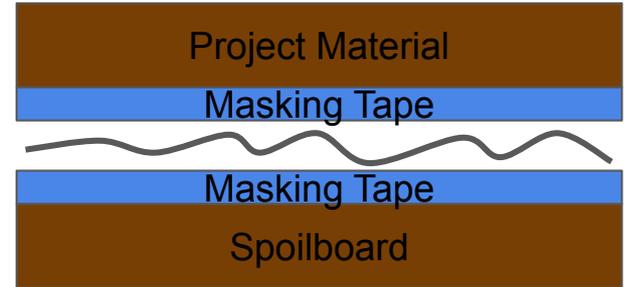
Fixturing

- Clamp your workpiece down directly to the bed if you are not cutting all the way through.
- Clamp a spoilboard down and screw the workpiece to the spoilboard if you're cutting through but not cutting pieces out (slots).
- Clamp a spoilboard, screw the workpiece to the spoilboard and use tabs to hold your piece in place if cutting all the way through.

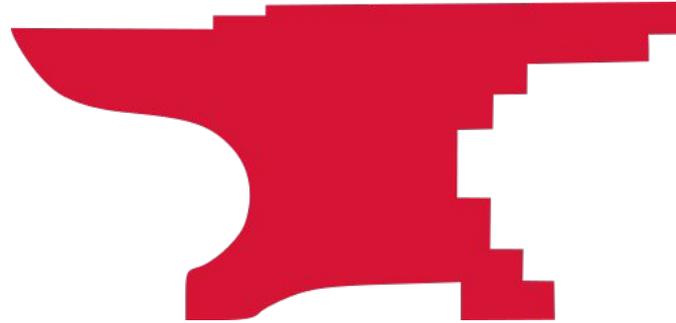


Fixturing

- Clamp a spoilboard and use double sided tape or the tape and spray glue/super glue method if you're cutting all the way through. The tape and glue method works when cut-out pieces are small or when you want to avoid sanding the tabs off.
- Use double sided tape instead of screws or glue.



Toolpath Preparation

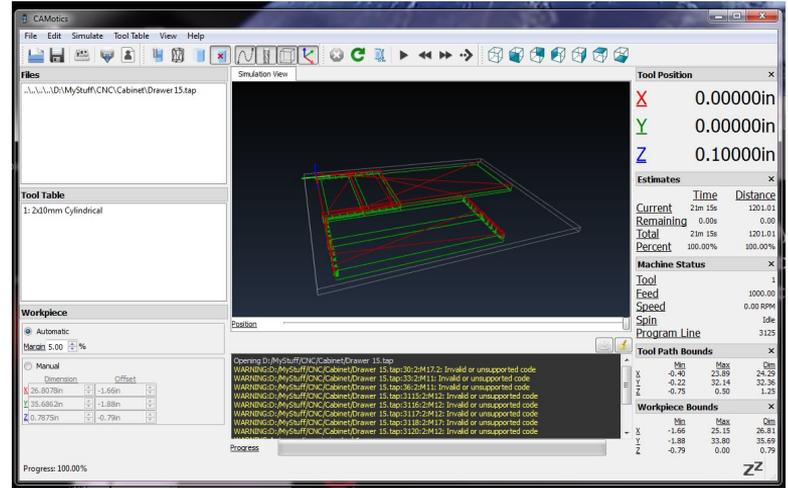


Toolpath Prep

This class assumes that you have already saved your Gcode and have used a simulator to *'absolutely confirm'* that your code is solid and only cuts where you want it to.

Gcode Simulators:

- CAMotics
- NC Viewer
- Simulator in Fusion 360
- Simulator in VCarve



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!

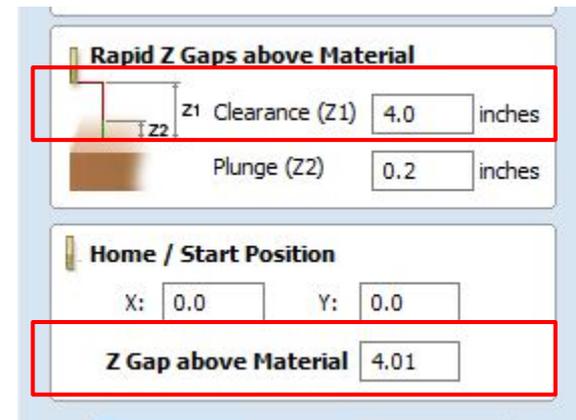
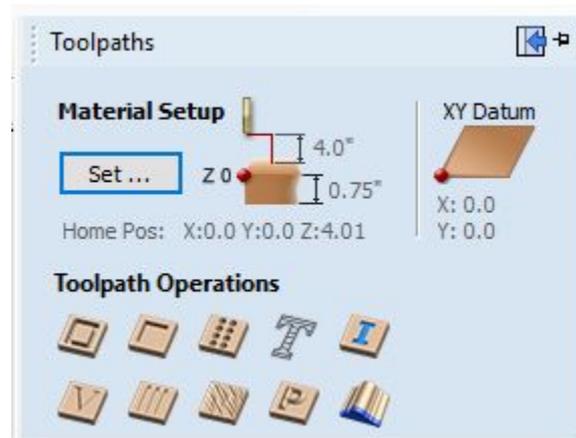
The screenshot shows the 'Job Setup' dialog box with the following settings:

- Job Type:** Single Sided (selected)
- Job Size:** Width (X): 18.0 inches, Height (Y): 10.0 inches, Thickness (Z): 0.75 inches
- Units:** inches (selected)
- Z Zero Position:** Material Surface (selected)
- 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]

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.

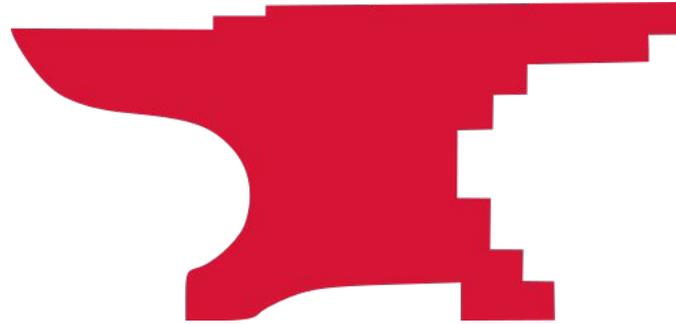


Cut Order

- Be aware of cut order.
- Try to leave smaller pieces attached to bigger pieces as long as possible.
- Use tabs to hold your pieces together during cutting. An oscillating cutting tool or jigsaw are useful to cut them off afterwards.
- Use additional hold down devices.



Machine Setup

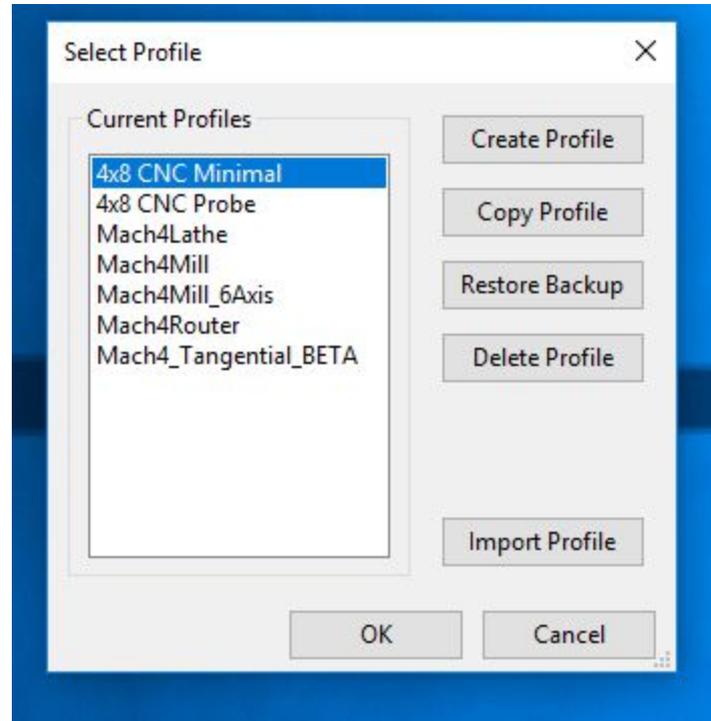


Important!!

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. Double check that the end mill height is high enough to clear any clamps or fixture pieces. Just because you set the Z height to 4" up doesn't mean the end mill is 4" up from the table until you've referenced as the Z during setup.
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.
7. Be sure your tools are securely mounted in in the collets and spindle.



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

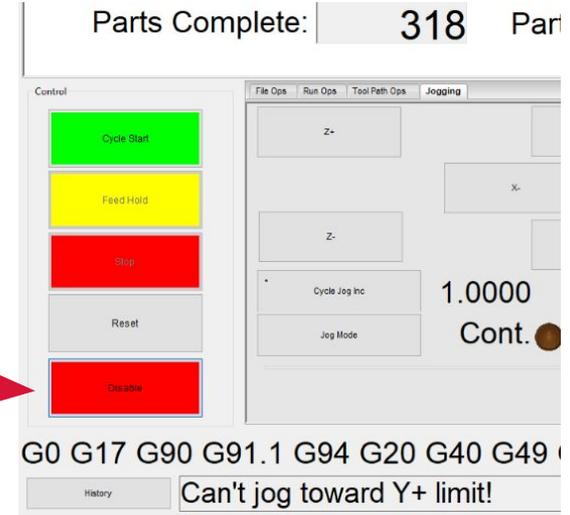
History PMDX: Mach halted due to Y-- LIMIT switch active Profile: 4x8 CNC Minimal Feed Rate: 0.00

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.



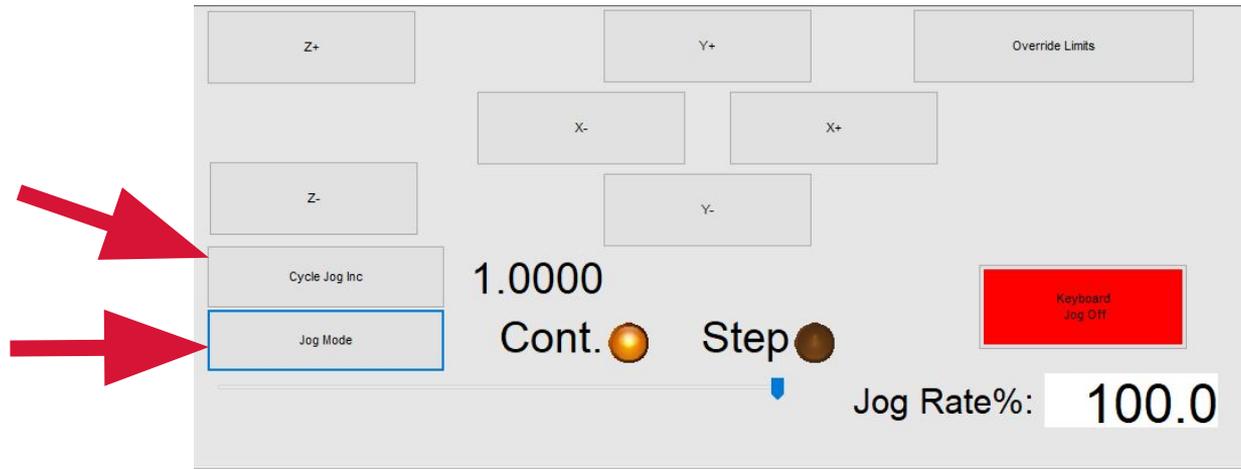
Jogging

The screenshot displays the Mach4 software interface. At the top, the menu bar includes File, View, Configure, Diagnostic, Wizard, Operator, and Help. Below the menu bar, there are tabs for Program Run, MCK, Tool Path, and Machine Diagnostics. The main interface is divided into several sections:

- Axis Control:** Shows the current position of the X, Y, Z, and A axes. The X-axis is at 1.4307, Y-axis at 26.4198, Z-axis at 3.8640, and A-axis at 0.0000. A red arrow points to the 'Jogging' tab in the Control section.
- Control:** Contains buttons for Jogging, Feed Hold, Stop, Reset, and Cycle. The Jogging section includes buttons for X+, X-, Y+, Y-, Z+, and Z- movement, along with Jog Rate% set to 100.0.
- Tool Information:** Shows Tool: 13, Tool Chng. button, Diameter: 0.0000, Height: 0.0000, and Tool Height button.
- Feed Rate:** Shows RRO%, FRO%, and SRO% indicators.
- Spindle:** Shows Spindle CW button and 0 RPM.
- Parts Complete:** 318, Parts Req.: 0, Total Parts: 0.
- Bottom Status Bar:** Shows G0 G17 G90 G91.1 G94 G20 G40 G49 G80 G98 G50 G67 G97 G54 G64 G69 G15 G40.1, Profile: 4x8 CNC Minimal, and 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.

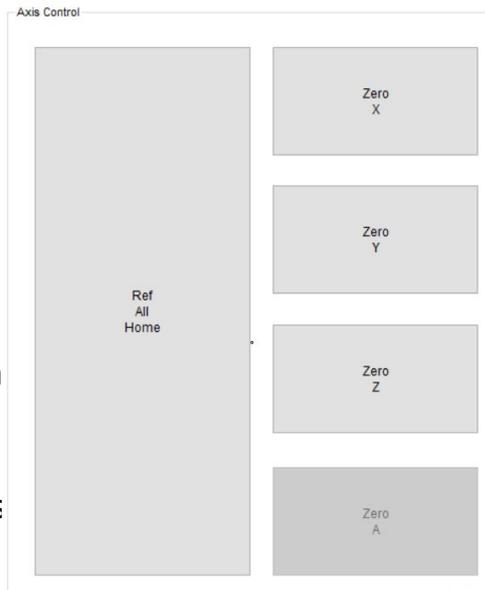
Jogging



- You'll use the plus and minus buttons to move the machine around prior to starting your GCode.
- 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.
- 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.
- 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.
- 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.
- 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.



Setting X0, Y0, Z0

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 X0 Y0

- 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.
- It doesn't matter if you set the X or Y first.
- 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
- Once both X and Y are set, move on to Z.



Setting X0 Y0

Mach 4 - Hobby
File View Configure Diagnostic Wizard Operator Help

Program Run MDI Tool Path Machine Diagnostics

G Code

Axis 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 Jng Follow

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

Control

Cycle Start Feed Hold Stop Reset Trapping

File Ops Run Ops Tool Path Ops Jogging

Edit G Code Set Next Line

Recent File Run From Here

Line: **0**

Close G Code Load G Code

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 Z0

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 end mill 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.

Setting Z0

Mach 4 - Hobby
File View Configure Diagnostic Wizard Operator Help

Program Run MDI Tool Path Machine Diagnostics

G Code

Axis 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 Jpg 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 Trailing

Edit G Code Set Next Line

Recent File Run From Here

Line: 0

Close G Code Load G Code

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

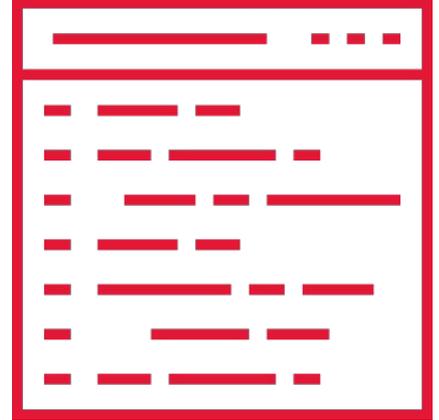
Sliders

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.

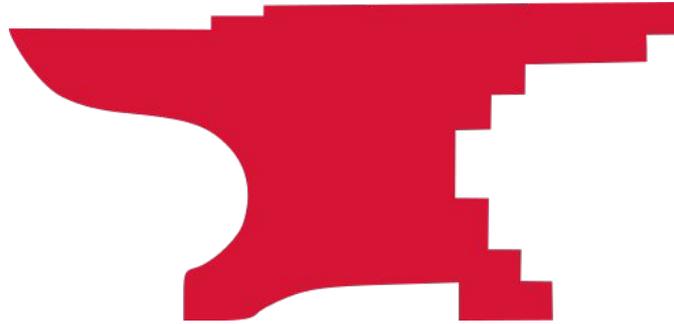


Load GCode

- Load GCode
- Review the header
- Review the image of the toolpaths
- 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.
- 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.



Running A Project



Step By Step

There is a laminated step by step checklist in the Big Red CNC room. If you want a copy of your own, it is on the Makersmiths Wiki.

If you find anything that should be changed, please mention it and the checklist will be updated.

<http://wiki.makersmiths.org/pages/viewpage.action?pageId=12386369&src=contextnavpagetreemode>

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!!!

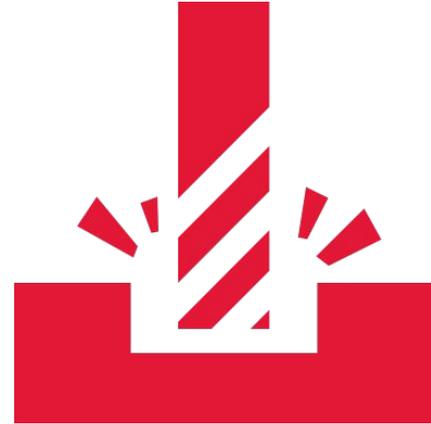
When you're comfortable that everything has been setup correctly, hit Cycle Start.

The screenshot displays the Mach4 software interface. At the top, the menu bar includes File, View, Configure, Diagnostics, Wizard, Operator, and Help. The main window is divided into several sections:

- Q Code:** A large empty text area for G-code.
- Axis Control:** A panel with four digital readouts (DROs) for X, Y, Z, and A axes. The X-axis is at 13.5310, Y-axis at 43.7331, Z-axis at 2.5840, and A-axis at 0.0000. Below these are buttons for 'Goto Zero', 'To Go', 'Machine Coordinates', and 'Soft Limits'.
- Tool Path:** A 2D graphical view of the tool path on a blue background.
- Current File:** A text field for the current program file.
- Control Panel:** A row of buttons including 'Load Macros', 'Load Last Wc', 'Limit Out', 'Ramp ToolPath', 'Display Mode', and 'Jog Follow'.
- Production Statistics:** A row of three input fields: 'Parts Complete: 318', 'Parts Req.: 0', and 'Total Parts: 0'.
- Control Buttons:** A vertical stack of buttons: 'Cycle Start' (highlighted with a red box), 'Cycle Stop', 'Feed Hold', 'Reset', and 'Dry Run'.
- File Operations:** Buttons for 'File Open', 'Run Ops', 'Tool Path Ops', 'Jogging', 'Edit G Code', 'Set Feed Line', 'Recent File', 'Run From Here', 'Clear G Code', and 'Load G Code'.
- Line:** A text field showing 'Line: 0'.
- Tool Information:** Fields for 'Tool: 13', 'Tool Chng.' (button), 'Diameter: 0.0000', 'Height: 0.0000', and 'Tool Height' (button). Below are 'Remember', 'Return', and 'Touch' buttons.
- Feed Rates:** Three vertical sliders for 'RRO%', 'FRO%', and 'SRO%'. A 'Spindle CW' button and a '0 RPM' display are also present.
- Bottom Status Bar:** Shows 'G0 G17 G90 G91.1 G94 G20 G40 G49 G80 G98 G50 G67 G97 G54 G64 G69 G15 G40.1', 'PMDX: Mach halted due to Y-- LIMIT switch active', 'Profile: 4x8 CNC Minimal', and 'Feed Rate: 0.00'.

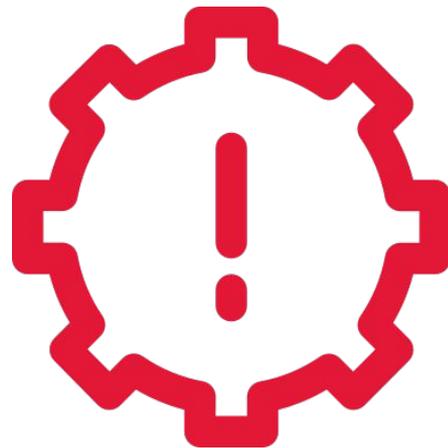
During Job

- Stay with the machine.
- Be ready to hit the Emergency Stop button if necessary.
- 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.
- NOTE: Speed slider does not automatically reset to 100% between tool changes or toolpaths.
- Once you have a little bit of experience, you can change the feeds and speeds of the cut.



Going Wrong

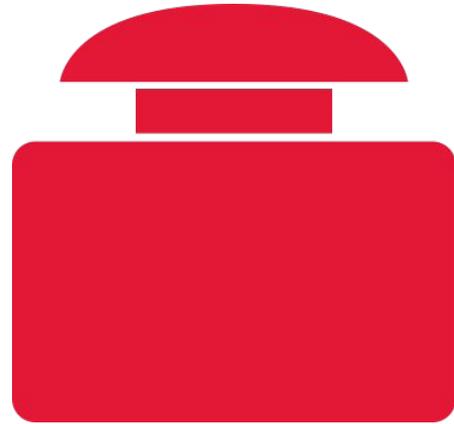
- Hit a clamp, screw, or other object you didn't mean to cut - EStop the machine, check your end mill and replace if necessary, re-reference the machine, start over
- Windows freezes/updates - the machine will stop if Windows freezes. You may have to start your cut over entirely.
- Cut depth is too deep - 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
- 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.
- Toolpath not cutting in the right place - abort the project
- Parts 'jump' off the work surface - abort the project
- You hit the limit switches and the machine stops.



Recovery

Recovering from Estop or Abort

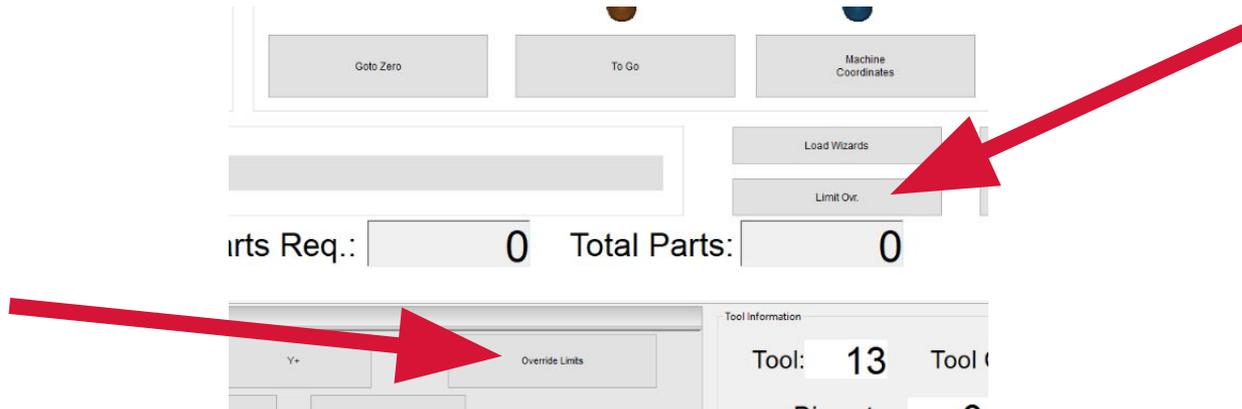
1. Turn the Estop switch to release if applicable
2. Press the green flashing "Enable" button
3. If the machine skipped - tried to move X or Y and was stopped because the bit was jammed or didn't move it's Z position correctly (you'll hear a difference) then you will need to re-reference the machine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



Recovery

Hitting a Limit Switch

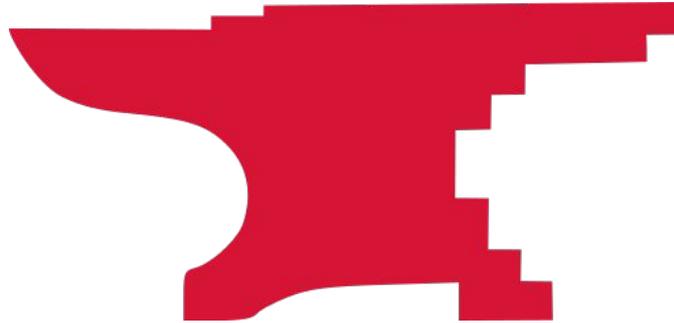
1. The machine is set up with limit switches which automatically stop the machine moving X or Y too far. If you hit a limit switch while jogging the machine or during an operation, the machine will stop.
2. You will need to press the Override Limits or Limit Ovr. button, move the machine off the limit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



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 end mills back on the shelves and pegboard

Other Info and Tips



Materials

Do and Do Not Cut List

DO cut:

Wood (soft and hard) MDF, chipboard, sign foam, melamine, lexan, acrylic, polycarbonate. (Special end mills needed for some materials.)

Cut with Care:

Aluminum up to ¼' thick.

DO NOT CUT

Anything else

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.
- Record your X0 Y0 position before hitting zeroing for your workpiece. This helps you get back to the same starting position in the event of an abort or Estop.
- Practice on a piece you don't care about first.
- A good fixture will help with repeatability and consistency.
- Consider using a registration mark to confirm X Y Zero

Advanced Techniques

Fixturing

Tool Offsets

Tool Compensation

Tool Libraries

G55 – G59 Offsets

Gcode subroutines

Manual Gcode

Conditional Gcode statements

Threading

3D

2 Sided work

Check Rides

Taking this class doesn't sign you off to run the machine alone immediately.

A "check ride" is required for final sign off.

Coordinate with one of the CNC class teachers for one of them to be present when you run your first project.

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. Double check that the end mill height is high enough to clear any clamps or fixture pieces. Just because you set the Z height to 4" up doesn't mean the end mill is 4" up from the table until you've referenced as the Z during setup.
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.
7. Be sure your tools are securely mounted in in the collets and spindle.

