EBF CNC Workshop

Outcomes

1. Explain the basics of operating the EBF CNC machine safely.
2. Demonstrate turning on, jogging, referencing, and running a toolpath on the CNC machine safely.
3. Understand the basics of feeds, speeds, cut depths, tool, and end mill selection.

Work in a 2.5D environment.

The EBF is a Big Powerful 10 HP spindle machine capable of spindle speeds to 24,000 RPM, feed speeds to 600 IPM, and rapid moves at 1,200 IPM.

Assumptions

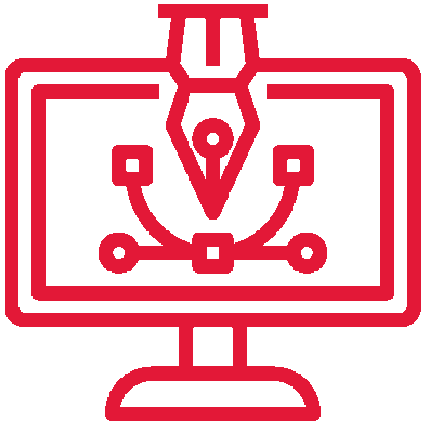
You want to get started using this machine.

Your CNC parts are usually 1 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.)

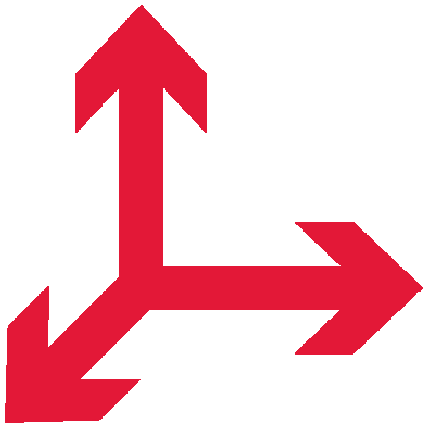
CAD

CAD is Computer Aided Design - it’s the software that you use to make/draw/design the thing you want to cut. At its most basic that could be a vector drawing program. If you're doing 2.5 cutting, you can draw in Adobe Illustrator, Corel Draw, Inkscape and others. You can even draw in Vcarve. If you're doing 2.5d or 3d projects, you can use more complex programs like Fusion360 or other CAD software packages.



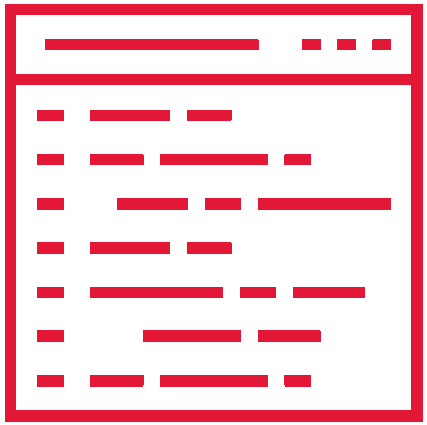
CAM

CAM is Computer Aided Manufacturing - is the software you use to generate the code that tells the machine what end mill you're using, how fast to spin, how fast to move and where to move. For most Makersmiths members that is VCarve or Fusion360. Note that you can do both CAD and CAM in VCarve and Fusion360, but you don't necessarily have to. Other CAM programs are CamBam, and jscut.org



GCode

GCode is the code 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. You save your GCode from your preferred CAM software.



Some GCodes

G0 = Rapid Motion

G1 = Controlled Motion Fxx where xx is Feed Rate

X#, Y# and/or Z# = Move machine (dependent on absolute or relative setting and rapid or controlled motion)

G28 = Go To Home

F= Feed Rate

S= Spindle Speed

M03 = Spindle on clockwise

M05 = Spindle stop

G04 = Dwell S/P (by itself requires 'return' to continue – recommended !)

Some GCodes

G92 X0 Y0 Z0 Sets a local zero spot for axis.

Tn n is number 1-8. Loads a cutting tool

T0 unloads any tool (empties spindle)

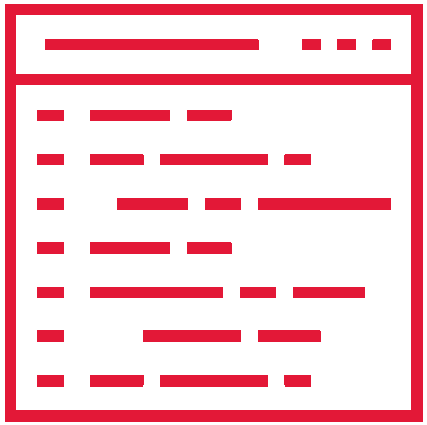
M11C6 - positioning pins up
M12C6 - positioning pins down

M11C2 - Dust hood down
M12C2 - Dust hood up

M11C4 - air blast cooling on
M12C4 - air blast cooling off

M11C8 - mist coolant on
M12C8 - mist coolant off

M11C7 - vacuum pump on
M12C7 - vacuum pump off



Some GCodes

G20 = all measurements in inches

G21 = all measurements in millimeters (But, EBF appears to default to centimeters !)

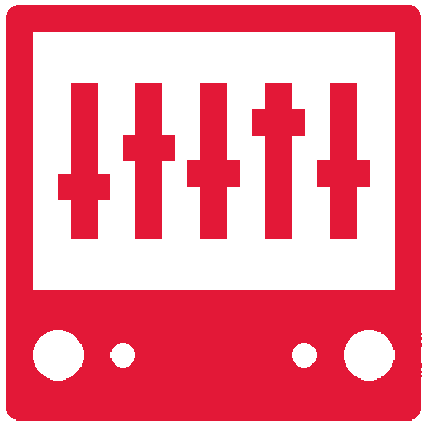
Macros: (sets of pre-coded instructions)

SETTOOL – uses big pushbutton (stored on left side of control cabinet) to touch off tools axis

SETMATERIAL – uses big pushbutton to set top of material (touch off Z axis).

Controller

After saving your GCode from your preferred software package, you execute/run that Gcode in WinCNC on the EBF CNC Machine. WinCNC is one example of a motion control software application. WinCNC is software that actually controls the EBF CNC machine. Pronterface is another which is used on smaller CNC machines. Others include LinuxCNC, Flashcut, PlanetCNC, and many others. Function of all is to convert a Gcode line into steps on stepper motors or other control switches.



Feeds

How fast the machine moves:

- Too fast and you get deflection of your endmill, reduction in endmill life, racking of the machine, excessive pulling on your workpiece, or a broken end mill
- 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:

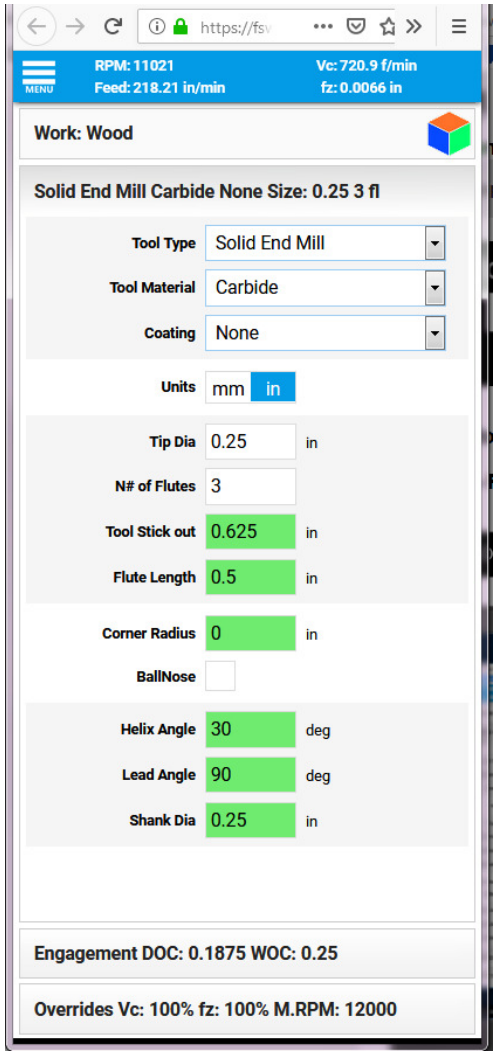
- Too fast and you can burn or melt your work or put too much strain on the machine if spinning a heavy/wide end mill
- Too slow and the end mill isn't cutting away enough material as it moves. Strangely enough, this wears out end mills very quickly.

Speeds

Speed (rotation and travel) is usually based on chip load – how much of a cut is made by each flute of the end mill and includes accommodation for chip evacuation from the cut.

Fancy formulas, or, a calculator, like,
https://zero-divide.net/?page=fswizard
(basic is free on-line, also a paid version.)
(G-Wizard is another from CNC Cookbook)

Be Conservative. Use a slower speed (half or less) until you see how the combination performs.



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 end mill

End mills

- End mills come in many shapes, sizes, cutting profiles, with different coatings, direction of flutes etc.
- For your starter projects you are likely to use spiral cut end mills of ⅛, ¼, and ½. V bits are also popular because they're often used for wooden sign making.
- Different bits are made for cutting different materials. Just because and end mill can cut something doesn't mean it should. Check what your end mill is supposed to be used for before using it on your project.

¼-½ Endmill

- If you are using one of the ¼ or ½ fluted end mills and cutting a wood product, use 12,000 as the spindle speed and 200 inches per minute as the feed speed.
- ¼ endmill = .125 AT MOST depth per cut.
- ½ endmill = .250 AT MOST depth per cut.

Toolpath Prep

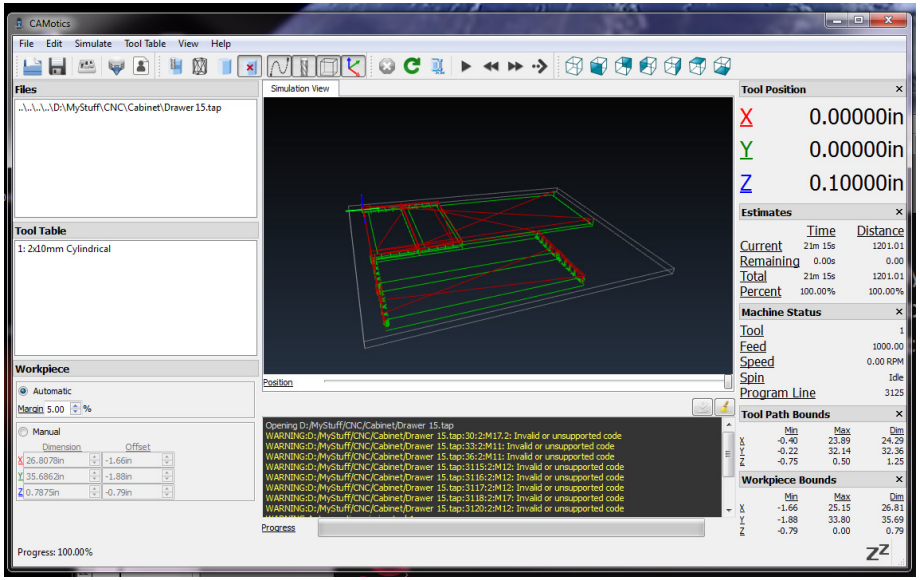
This class assumes that you have already saved your Gcode and have used a simulator to '*absolutely confirm*' that you code is solid and only cuts where you want it to.

Gcode Simulators:

CAMotics

NC Viewer

Simulator in Fusion 360



Cut Order

The Vacuum table holds sheets down really well, but not perfectly.

Be aware of cut order.

Try to leave smaller pieces attached to bigger pieces as long as possible.

Toolpath Prep

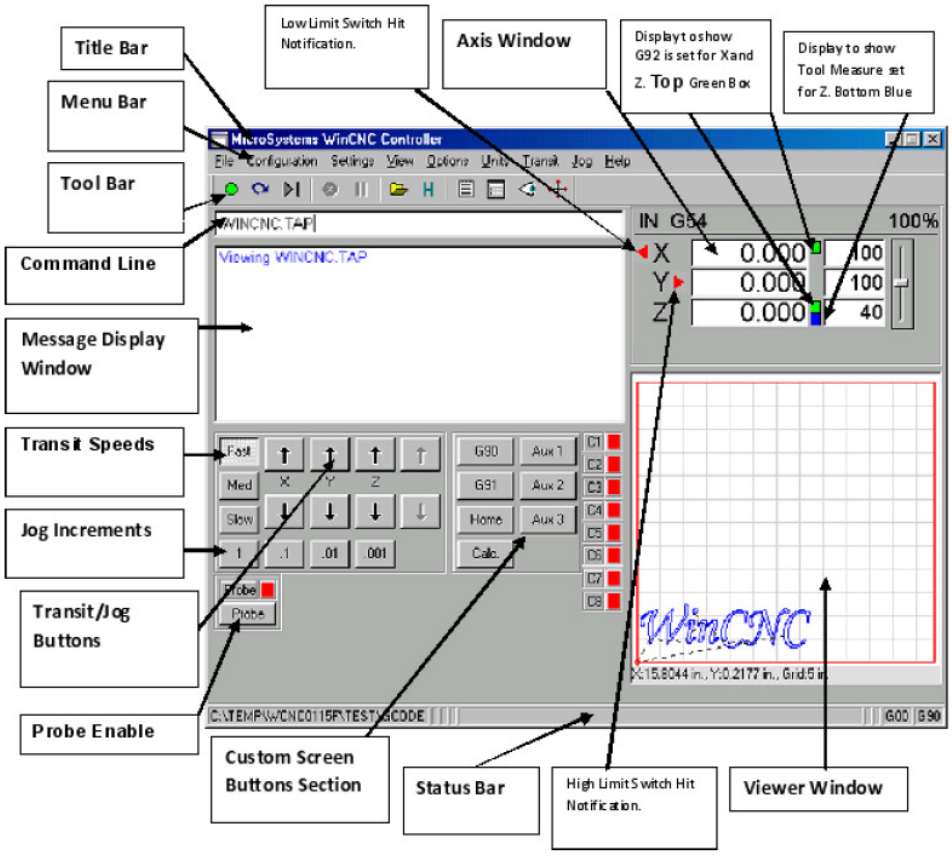
This class assumes that your Job Type is Single Sided, that your Job Size has been set to the size of your workpiece, that the Z Zero Position is set to the Material Surface, and that the XY Datum Position is set in the top left front corner. It also assumes that you have chosen the right end mills for each toolpath, your feeds and speeds settings are correct, and that your Z Clearance Height settings are correct.

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

Important!!!

You need to be with the machine while it's cutting and be ready to stop the machine if something goes wrong.
Always have quick access to the Emergency Stop button. WinCNC does not have an elegant restart or 'run from line' capability.
Double check that the endmill 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 endmill is 4" up from the table until you've referenced as the Z during setup.
Make sure your toolpaths aren't going to accidentally cut your clamps or fixture locating pieces.
ALWAYS use a spoil board if you are going to cut through your chosen material. If you cut into the Aluminum vacuum table machine bed it's going to ding your pocket book big time!
Be careful of anything you put on the bed that you don't want cut or knocked over.
Always raise the Z axis first (lest you break your end mill and get to wear the 'end mill of shame')
Be sure your tools are securely mounted in ISO30 tool holders with the correct ER32 collets, and that the tool configuration (stick out) is as per your tool set up.

WinCNC

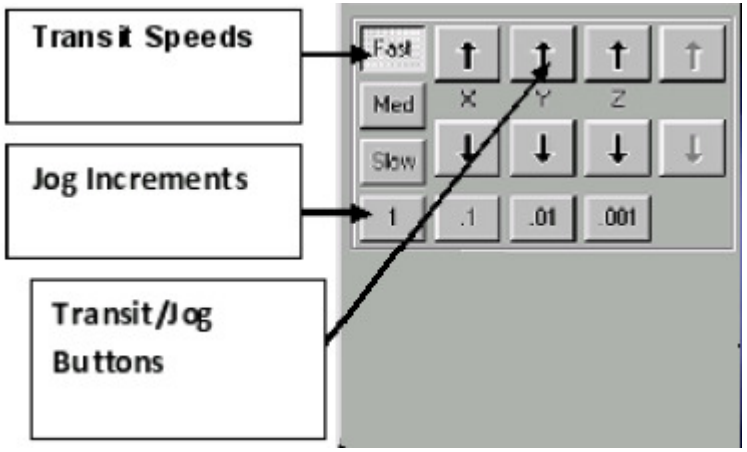

Title Bar
Menu Bar
Tool Bar
Command Line
Message Display Window
Transit Speeds
Jog Increments
Transit/Jog Buttons
Probe Enable
Low Limit Switch Hit Notification.
Axis Window
Display t oshow G92 is set for X and Z. Top Green Box
Display to show Tool Measure set for Z. Bottom Blue
Custom Screen Buttons Section
Status Bar
High Limit Switch Hit Notification.
Viewer Window
MicroSystems WinCNC Controller
File Configuration Settings View Options Units Transit Jog Help
WINCNC.TAP
Viewing WINCNC.TAP
IN G54
100%
X 0.000 100
Y 0.000 100
Z 0.000 40
Fast
Med
Slow
1
.1
.01
.001
G90
G91
Home
Calc.
Aux 1
Aux 2
Aux 3
Probe
Probe
C:\TEMP\WCNC0115P\TEST\GCODE
G00 G90

Referencing

Whenever you start a new project on the EBF, you want to first home the machine. This ensures that the machine always knows where the absolute X, Y and Z ends of the machine are.
Make sure there is nothing on the bed that can get hit.
Use the Home button to register the machine. This will raise the Z axis first, then home the X and Y. Note: Spindle will move to the front left corner of the machine.
This is the Absolute Coordinate X0 Y0 Z0
In the event that the machine 'skips' when moving during a cut, the machine will no longer correctly know where the absolute X0 Y0 point is and will abort. You will need to re-home the machine and set your local material zero before you start again.

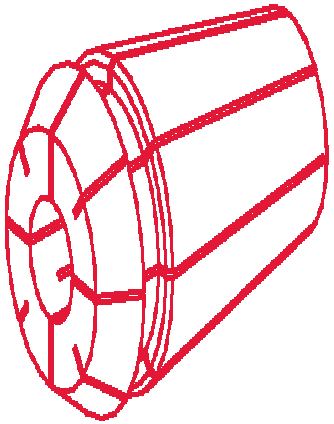
Jogging

You'll use the these buttons to move the spindle around prior to starting your GCode.
In Fast, Med, and Slow modes the spindle will move as long as you hold the button down.
In Jog Increments, spindle will move 1 unit for each button click.
When moving the Z height downwards towards your workpiece, make sure to change the step so that you don't drive the end mill into the workpiece, risking breaking it.



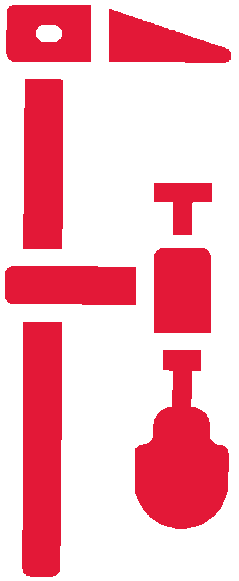
Collet & ISO30

1. The EBF uses ISO30 tool holders and ER32 collets.
2. Use the correct collet size to match the shank of your bit.
3. Use the tool holding fixture and wrench to loosen or tighten the nut on the tool holder. (Stored inside the front door of the control cabinet.)
4. Snap the collet into the collet nut first.
5. In no event should you be straining to tighten the collar and collet.
6. Try to make your end mill stick out the collet as little as you can, bearing in mind that the end mill has to stick out at least as far as the deepest cut you're making and in coordination with the tool length you set when your computer wrote your gcode. You can't have the end mill stick out 1/2 inch and try to cut 3/4 inch into something. And, leave a little space for chip evacuation. The less stickout you have, the less the end mill deflects/bends during the cut.
7. Note the bit 'stickout' for your tool table (if you are making one)
8. When placing an end mill in the collet, do not bottom the bit on the top of the holder. Leave about 1/16" of space (the tool moves up into the holder slightly as the nut is tightened).



Attachment

1. Do not screw, cut into or otherwise damage the machine bed.
2. The EBF has a 4 Zone Vacuum table.
3. The vacuum table has good holding force.
4. Vacuum a your own spoil board if you’re cutting through.
5. Use tabs for small pieces.



X, Y, Z Zero

1. Usually the front left material corner is X0, Y0. (Front left corner of the machine.) There may be times when you use the center.
2. Z0 should be the top of the material to be cut until you get more experience.
3. Other setups are possible but these are advanced uses.

Note: The corner of a pinned sheet/material is not X0 Y0

Z Clearance

Your CAM software will likely have a setting for the safe clearance height for the end mill between cuts.

Safe height above your material may not be a safe height above clamps or positioning blocks. Adjust accordingly.

Starting up

1. Check the oil in the Vacuum pump. (sight glass on back)
2. Make sure the dust collector bags (inside) are not full. Change if necessary.
3. Move the control cabinet out to clear the machine and the door.
4. Place correct bits in tool holders.
5. Put the tool holders into the correct positions (1-8, left to right)
6. 3/4” 5x10 coverage MDF base board should be on the table.
7. Place your 4x8 spoil board/fixture on top of this.
8. Prep data for your tool library (as needed).

Starting up

1. Make sure compressed air is supplied to the table (connection front lower left corner near floor – regulator and filter on wall – minimum 70 psi).
2. Make sure the Circuit Breaker on the Main Panel is on. (Circuit breaker feeds a transformer that supplies the correct power to the EBF)
3. Turn on the local power disconnect. (Lever up)
4. Rotary switch on back of the control cabinet should be set to 'Marche' (It stays in this position)
5. Power on the control cabinet (black switch on left side - up).
6. Open the cabinet and reset the computer (On door, Press button ⏻ on front center of computer case, press, hold 3 seconds, release). Close door. Machine takes a while to boot.

Starting up

1. Windows on this machine is in French. (Don't worry, the CNC control program is in English.) (We're trying to fix this.)
2. Your Gcode may be loaded from a flash drive. USB hub is on left side of machine.
3. Launch WinCNC
4. Home Machine
5. Move spindle towards back of machine, or wherever you need to move it to in order to position your spoil board/fixture.
6. Be sure fixtures and spoil boards are clear of pins.
7. Select 'Pins Up' button.
8. Position base board against side and front pins.
9. Position your spoil board/fixture as needed on the table.
10. Load your tool table as required.
11. Adjust dust hood if necessary (Allen head screws on sides)

Starting up

1. Position your raw material on your spoil board/fixture.
2. Turn on all 4 vacuum zone control switches (front of machine in oval cut out). (Adjust as necessary for alternate spoil boards)
3. Load tool as needed with Tn command (n is tool number 1-8)
4. Place Touch-Off button on side of machine bed.
5. Move tool over Touch-Off Button.
6. Execute 'Set Tool' (do this for each tool you are using)
7. Vacuum on
8. Move Spindle to material X0 and Y0 position.
9. Note down the X and Y positions (valuable if you need to restart)
10. Issue command G92 X0 Y0 to set work piece zero.
11. Move Touch-Off button to suitable location on material.
12. Move Tool over Touch-Off button.
13. Execute 'Set Material'

Starting up

1. Confirm Vacuum on (button on WinCNC)
2. Check security of raw material. (Table vacuum zones can be adjusted – please consult tool steward.)

3. Load Gcode (File, Open, yourfile) **WAIT !**
4. NOTE: Pressing Enter will start the cut !
5. Turn on Dust Collector (breaker switch on box just inside the wall to the right of the machine)

Before GO

1. Are the X, Y, Z zeroed/set for the toolpath you're about to cut?
2. Are the right tools in the machine and in the correct holders?
3. Is there anything on the rails or bed that could get hit?
4. Is your material secured down?
5. Is the emergency stop button ready?
6. Confirm there is air pressure.
7. Did you turn on dust collection?

GO

1. Press Enter

2. Cutting will begin.

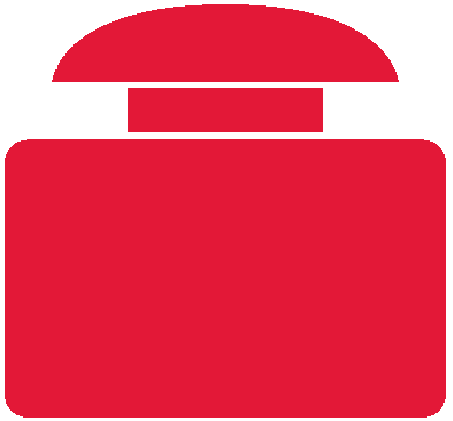
During Job

Stay with the machine.
Be ready to hit the Emergency Stop Button (Estop) 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. Speed slider may help.
NOTE: Speed slider does not automatically reset to 100%
The sound of air leaking means air pressure may have been lost. If so abort the job.
Once you have a little bit of experience, you can change the feeds and speeds of the cut.

Recovery

Recovering from Estop

Turn the Estop switch to release.
Home the machine.
Reset your Tools, X, Y, and Z material Zero's (if you noted these during setup, your life will be a lot easier).
Go through the Startup checklist
Restart your program.



Recovery

Recovering from Abort

Resolve the issue.

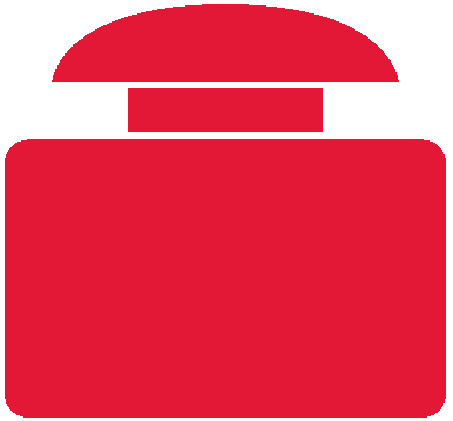
Depending on the problem, Home the machine, reset your Tools, X, Y, and Z material Zero's (if you noted these during setup, your life will be a lot easier), and

Abort during a G00 move usually confuses the axis and requires re-homing.

Go through the Startup checklist

Start your program, or

Restart your program (WinCNC has a 'run from line feature' – use with caution!)



Recovery

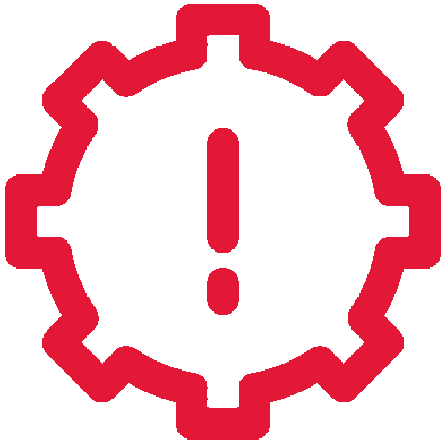
Hitting a Limit Switch

The machine is set up with limit switches which automatically stop the machine moving X, Y or Z too far. If you hit a limit switch while jogging the machine or during an operation, the machine will stop.

You will need to re-home the machine and start over.

Going Wrong

1. Hit a clamp, screw, or other object you didn't mean to cut - EStop the machine, check your endmill 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, re-home machine, set up X, Y, Z, reload the Gcode, go through start up checklist and restart project.
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 stops



After Job

Wait till the spindle stops spinning before doing anything.
Vacuum off
Dust collection off
Jog the machine out of the way of your material as needed
T0 removes any tool from the Spindle
Spindle to Parking
Lower Spindle Z
Remove your material
Vacuum top, under and around machine
Empty dust collection bag if full
Restore/adjust tools in tool holders and tool order as needed.

After Job

Disconnect air line
Turn off cabinet (black switch on side of cabinet down)
Turn off Power Disconnect
Turn of EBF Breaker in Breaker Box.
Clear any custom fixtures and make machine ready for next user

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, or leaving enough room for error

A good fixture will help with repeatability and consistency.

WinCNC and this machine do not have the most elegant recovery. Any Estop means re-homing the machine and starting over.

Consider using a registration mark to confirm X Y Zero

After Abort, you can attempt to restart your gcode from a particular line.

The screen slider can control feed speeds up to 200%. Note – it does not reset and has to be manually moved back to 100%.

When correctly set up, the machine is fast and smooth, a real production workhorse.

Advanced Class

Fixturing
Tool Offsets
Tool Compensation
Tool Libraries
G55 – G59 Offsets
Gcode subroutines
Manual Gcode
Conditional Gcode statements
Threading
3D
2 Sided work