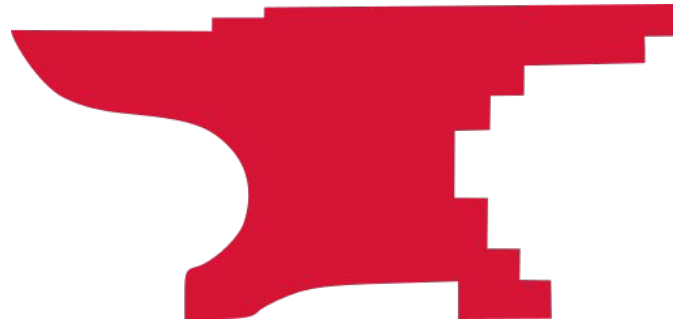# EBF CNC Workshop Class Slides

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. Each student sets up and runs a pre-prepared project.

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

- Stay 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 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.
- 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!
- 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.
- 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.

Machine Parts
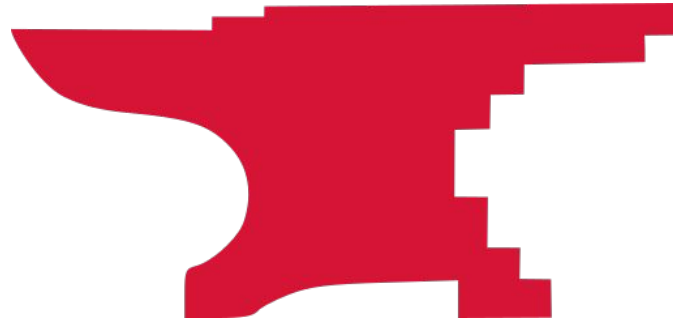
Hardware

1. Machine Power On/Off - multiple switches need to be done in order
2. Vacuum for vacuum table - must be on to secure workpieces
3. Air Compressor - must be on at 70 PSI
4. Dust Collector - must be on unless you want to clean up a mess
5. Tool holders and collets
6. Computer Software - WinCNC

Collet & ISO30

- The EBF uses ISO30 tool holders and ER32 collets.
- Use the correct collet size to match the shank of your bit.
- Use the tool holding fixture and wrench to loosen or tighten the nut on the tool holder. (On the end wall to the right of the machine.)
- Check the collet and nut and remove any accumulated sawdust left by the previous person. If the collet is packed with sawdust, it will not hold the endmill properly.
- Snap the collet into the collet nut first.
- In no event should you be straining to tighten the collar and collet.
- 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.
- Note the bit 'stickout' for your tool table (if you are making one)
- When placing an end mill in the collet, do not bottom the bit on the inside of the holder. Leave about 1/16" of space (the tool moves up into the holder slightly as the nut is tightened).
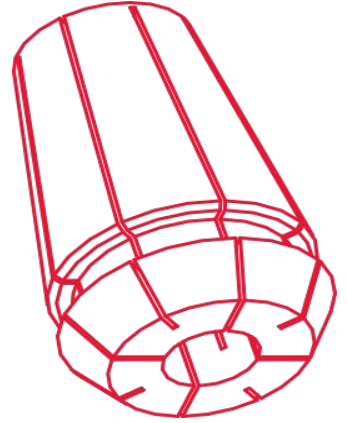
End Mills

FIRE HAZARD!

- Endmills can do become dull. Bear in mind that the 10 horsepower spindle will take a dull end mill and attempt to 'power through' your material. This can create excessive heat that can result in burning embers and sawdust being drawn into the dust collector.
- This can set the dust collector on fire.
- Make sure your tools are sharp.
- If you smell smoke while using the CNC, stop immediately.

# Fixturing - Securing Material

Fixturing

The EBF is a Big Powerful Machine.

This means that your work piece must be held *very very securely* to the table. You will need a bomb proof fixture that will guarantee that you work is held tight.

Fixturing

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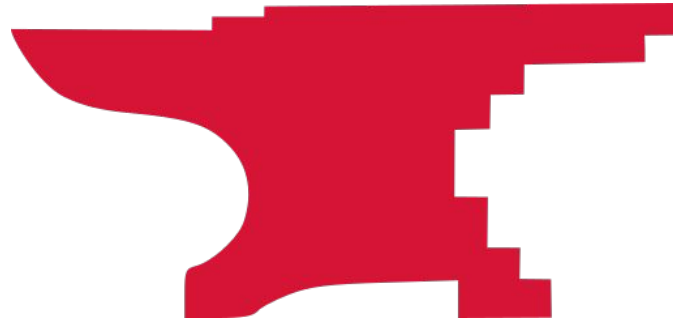- Use tabs for small pieces.
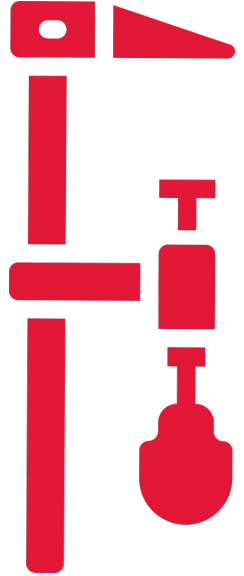
Fixturing

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

Clamp or screw all four corners of your workpiece to your spoilboard.
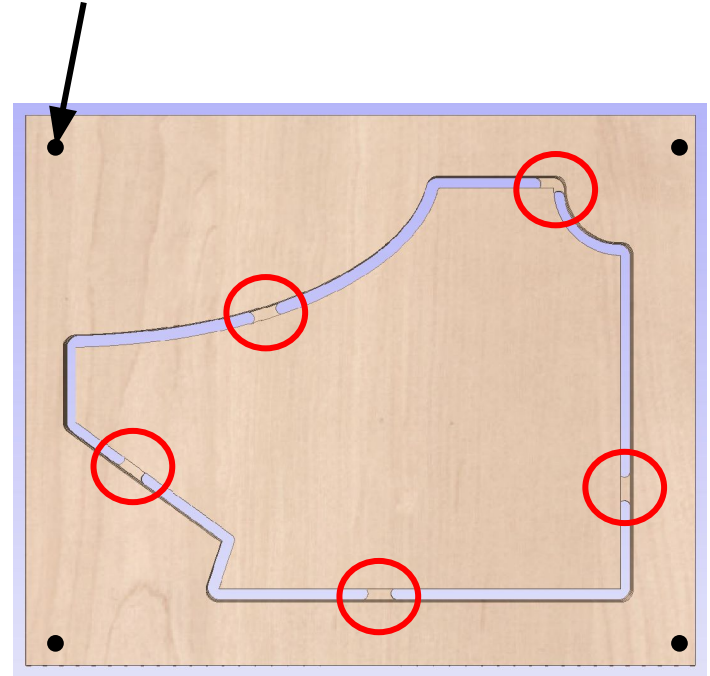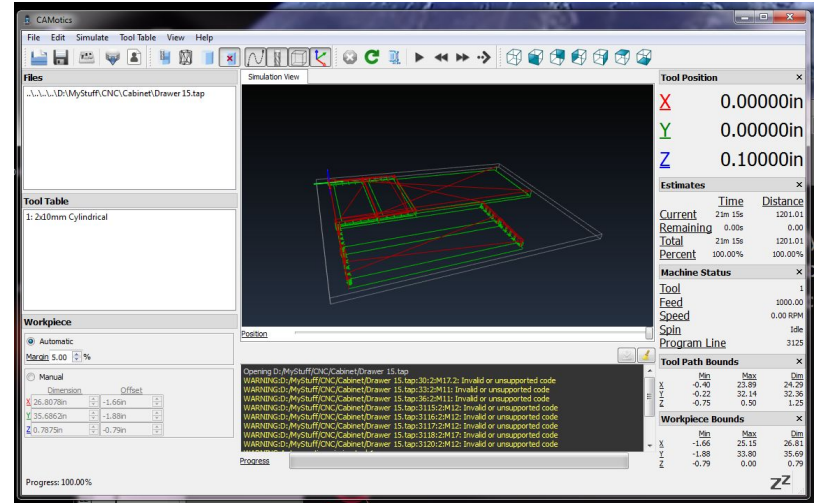
Toolpath Preparation

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
- Simulator in VCarve

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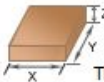
In short, there are a lot of things to get right!

Z Clearance

Your CAM software will likely have a setting for the safe Z clearance height and a Home / Start Position height for rapid movements between cuts. 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.

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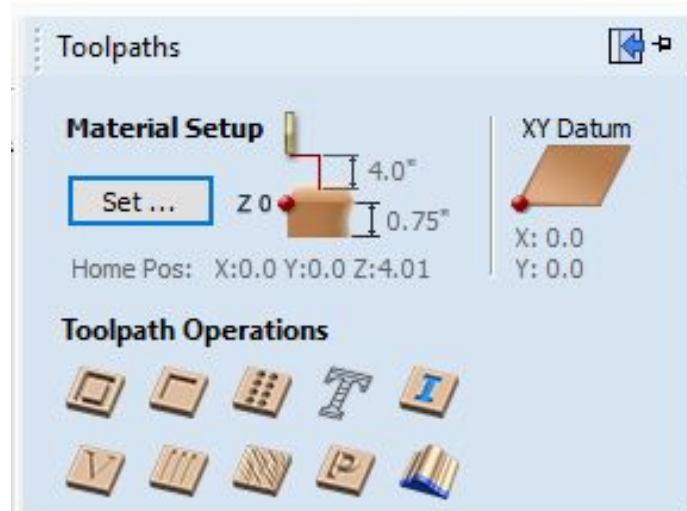
Cut Order

- 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.
- Use additional hold down devices.
- If you assume a hold down pressure of 1 pound per square inch (which you can't get by the way), a 10 inch by 10 inch piece would be held down with 100 pounds of force (not bad). A 1 square inch piece would be held down with a force of 1 pound (NOT good).
- Use tabs to hold your pieces together during cutting. An oscillating cutting tool or jigsaw are useful to cut them off afterwards.

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 quick access to the Emergency Stop button. WinCNC does not have an elegant restart or 'run from line' capability.
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 your clamps or fixture locating pieces.
5. 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!
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. Always raise the Z axis first (lest you break your end mill and get to wear the 'end mill of shame')
8. 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.

Step By Step

- There is a laminated step by step checklist attached to the machine. If you want a copy of your own, it is on the Makersmiths Wiki. There are both long and short versions depending on your familiarity with the machine.

- There is a Troubleshooting Guide and videos on the Wiki also.

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

- http://wiki.makersmiths.org/display/MAK/EBF+CNC

WinCNC

Title Bar

Menu Bar

Tool Bar

Command Line

Message Display Window

Transit Speeds

Jog Increments

Transit/Jog Buttons

Probe Enable

Custom Screen Buttons Section

Low Limit Switch Hit Notification.

Axis Window

Display to show G92 is set for X and Z. Top Green Box

Display to show Tool Measure set for Z. Bottom Blue

Status Bar

High Limit Switch Hit Notification.

Viewer Window

Referencing

- Whenever you start a new project on the EBF, you have to first home the machine. The machine will not move until it has been homed. 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 that spindle will move to the front left corner of the machine. This is why you put it inside the red tape.
- 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.
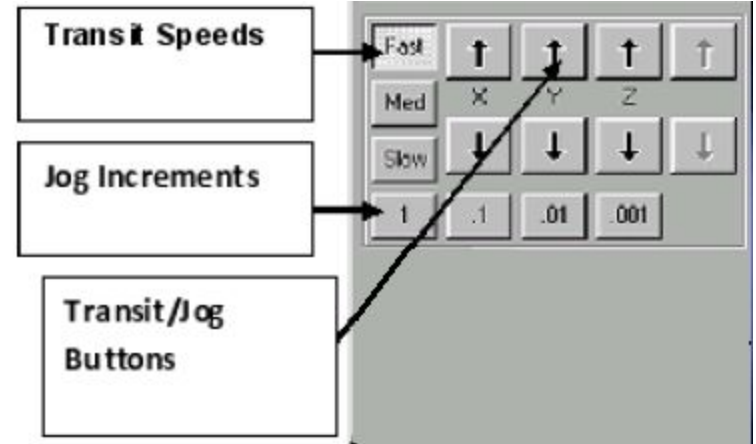
Setting 0,0,0

1. When you setup your toolpaths in your CAM software, you set where X0,Y0 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.
2. Other setups are possible but outside the scope of this class.
3. 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.
4. It doesn't matter if you set the X or Y first.
5. Once you are happy that the endmill is in the correct position, click the Set Work X Y Zero button in WinCNC.
6. On the EBF, you set Z0 using the touch off pad. Instructions are in the checklist.
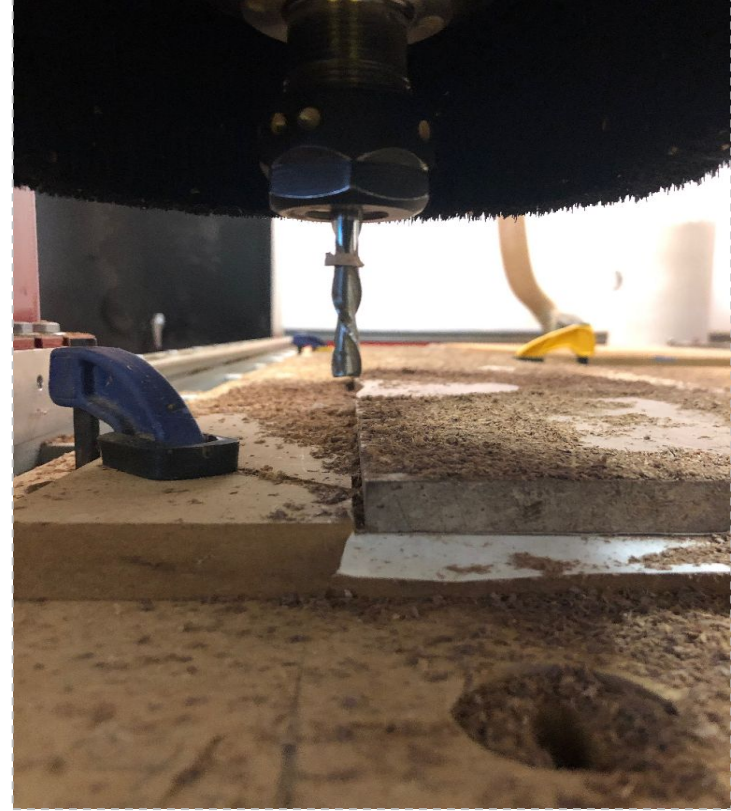
Running A Project
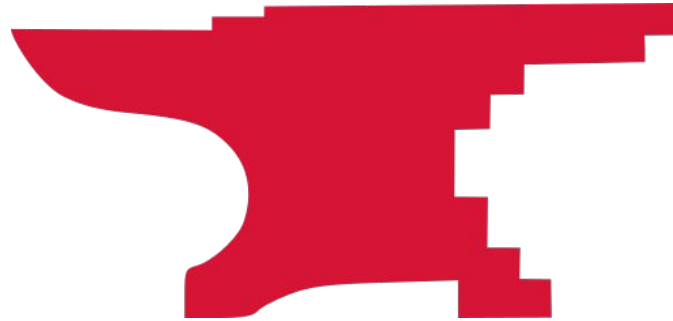
<u>Prior to Turning on the EBF</u>

1. Check the oil in the vacuum pump.
2. Check dust collector bags.
3. Move the control cabinet into the red taped area.
4. Turn on air compressor.
5. Check that compressed air is supplied to the table - at least 70 psi.
6. Load cutters/end mills into the tool holders and tool holders into the correct slots.
7. Adjust the dust hood if necessary.
8. Move rubber gasket material to the correct positions for your piece.
9. Put down your spoilboard

<u>Turning Machines On</u>

1. Turn on the EBF at main circuit breaker.
2. Turn on the local power disconnect via the big red lever to the left of the machine.
3. Turn the rotary switch on back of the control cabinet is to 'Marche'
4. Turn on the control cabinet power on left side of the control cabinet.
5. Open the cabinet and push the power button on the computer.

Setting Up To Cut

1. Launch WinCNC
2. Press "Home" on WinCNC
3. Press the "Set X0 Y0 Z0" button
4. Position your spoil board/fixture and work material.
5. Load your tool table in WinCNC if needed (not frequently used).
6. Touch-off each tool - Set Tool Z
7. Cover any unused holes in the vacuum table.
8. Turn on the vacuum by pressing the "Vacuum" button on WinCNC.
9. Check the security of your spoilboard, fixture, and/or work piece.
10. Touch off the tool on the top of your workpiece - Set Material Z. Make sure vacuum is on.
11. Jog the spindle to where you want your material X and Y zero to be.
12. Press the "Set Work X0 Y0" button.

<u>Loading GCode and Cutting</u>
1. Save your GCode file from your flash drive to the computer.
2. Turn on the dust collector.
3. Drop the machine pins if you raised the pins for aligning your work piece.
4. Open the GCode file in WinCNC. Press Enter. This will pickup the correct tool and start the spindle.
5. Once you hear the spindle spinning, press Enter again to start cutting.

Post Cut

1. Turn dust collection off
2. Press the "Go To Parking" button.
3. Lower the spindle Z height to just over the pad in the parking position.
4. Exit WinCNC - spindle will drop
5. Remove your material
6. Vacuum top, under and around machine
7. Empty dust collection bag(s) if full
8. Remove end mills from tool holders and replace all Makersmiths parts inside the cabinet.

<u>Post Cut</u>

1. Shut down Windows
2. Turn off cabinet (black switch on left side of cabinet)
3. Turn switch on the back of the cabinet to "Arrete".
4. Turn off power at the local breaker box behind the cabinet - big red lever.
5. Turn off EBF breaker in the breaker box in the metal shop.
6. Move the PC cabinet back out of the way and so that the metal shop door can be opened.
7. Put the large spoilboard back on top of the EBF to prevent shop dust etc. from getting into vacuum holes.

Before GO

1. Are the X, Y, Z zero positions 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. Did you confirm there is air pressure?
7. Did you turn on dust collection?

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 adjustments may help.
- 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.
- The sound of air leaking means air pressure may have been lost. If so abort the job.
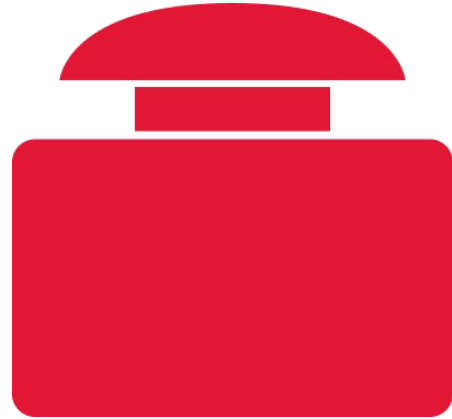
- If you hit a clamp, screw, or other object you didn't mean to cut - EStop the machine, check your end mill and replace if necessary, Home the machine, start over.
- Windows freezes/updates - the machine will stop if Windows freezes. You may have to start your cut over entirely.
- Cut depth is too deep - either overall or on a first pass - Estop machine, change the depth of cut in your CAM program, Home machine, set up X, Y, Z, reload the Gcode, go through start up checklist and restart project.
- 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 to avoid melting the plastic.
- Toolpath not cutting in the right place
- Parts 'jump' off the work surface
- You hit the limit switches

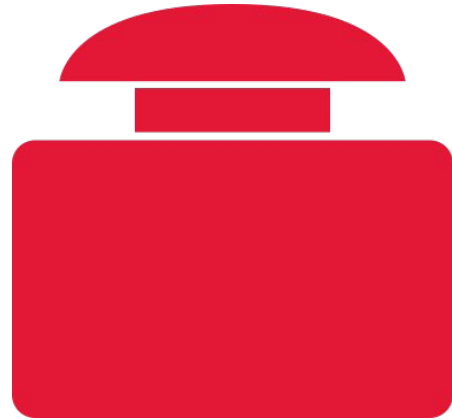# Recovery

Recovering from Estop

- Rotate the Estop switch to release it.
- 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.

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)
- 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!)

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 Home the machine and start over.

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. Remove endmills from tool holders (except flattening bit)
7. Inspect and clean the collets and holders to make certain that you leave them ready for the next user.
8. Put all tool holders and end mills back in the cabinet.

Clean These!

Other Info and Tips
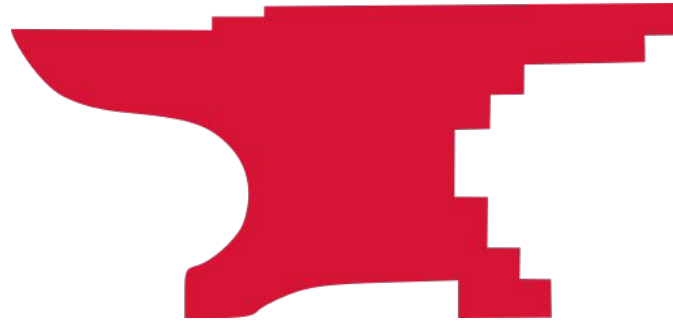
Materials

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, 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
- 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.

Other Tips

- Fusion 360 - Configuration for WinCNC. Remove the G43 line immediately following the first G00 move. This is tool length compensation that is automatically taken care of when you use 'Set Tool'.
- 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 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 quick access to the Emergency Stop button. WinCNC does not have an elegant restart or 'run from line' capability.
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 your clamps or fixture locating pieces.
5. 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!
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. Always raise the Z axis first (lest you break your end mill and get to wear the 'end mill of shame')
8. 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.