CNC Workshop
Pre-Class Materials

Pre-Class

Prior to a Big Red or EBF CNC class, students should read these slides and watch some of the linked videos so that they have a basic understanding of:

- CAD and CAM
- Gcode
- Feeds, speeds and cut depth
- End mill selection

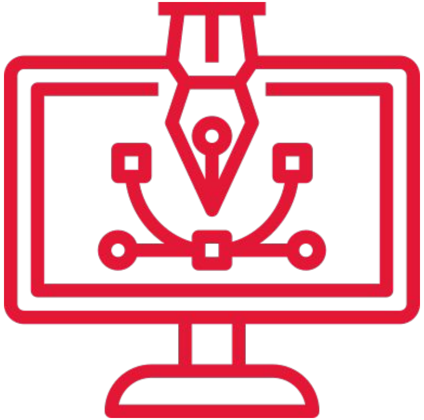
The videos linked to below are not the ultimate authority on any of the subjects covered but watching them will give the students a basic understanding of the topics prior to the class.

All guidance in the videos should be superseded by guidance given by the Makersmiths CNC instructors.

Watch this first: CNC Basics - What You Need To Get Started - 9:17

CAD

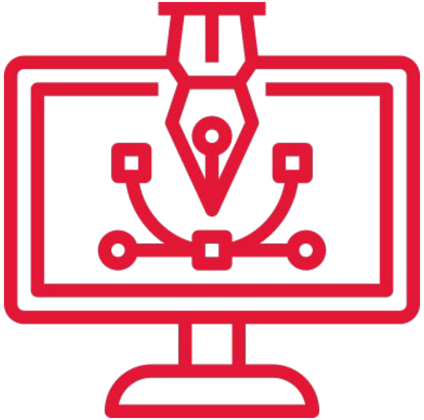
CAD is Computer Aided Design - it’s the software that you use to make/draw/design the thing you want to cut. At its most basic, that could be a vector drawing program such as Adobe Illustrator, Corel Draw, Inkscape and others. You can even draw in Vcarve. If you're doing 3D projects, you can use more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



CAD

If you are making signs and cutting out simple shapes, learn to use Adobe Illustrator, Corel Draw, or Inkscape. These are great programs to design for projects that only require two dimensional drawings. If you are cutting out more complex shapes or parts that will later be joined together to make a 3D object, you will want to use a 3D drawing program like Fusion 360 or Tinkercad.

- Inkscape Explained in 5 Minutes - 4:44
- Fusion 360 Tutorial for Absolute Beginners - 19:55



Inkscape, Fusion 360 (with limits) and Tinkercad are free to use so can be a good place to start if you don't already have a preferred program.

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 usually 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

CAM

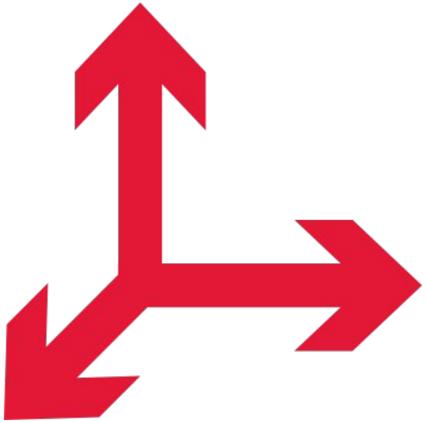
VCarve Pro has a “Makerspace Edition” which means you can use almost all of the functionality on your personal computer. The last bit of functionality, exporting GCode, can be done from one of the Makersmiths space computers.

- Introduction to VCarve - watch at least the first 10 minutes

Toolpaths

A toolpath is what you define in VCarve, Fusion360 or other CAM program for each cut or set of cuts you want to make. You set the parameters of the cut such as what end mill will be used, the feed and speed rates, the depth of cut, how high the end mill will raise between cuts (important for safe operation), where the project's X0 Y0 location is set, and many more.

For example, if you're making a sign, you'll have a least one toolpath for carving the letters (often with a V shaped end mill) and a second toolpath cutting out the shape of the sign (with a straight cut end mill).



GCode

GCode is the line by line instruction 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. It's a relatively simple programming language.

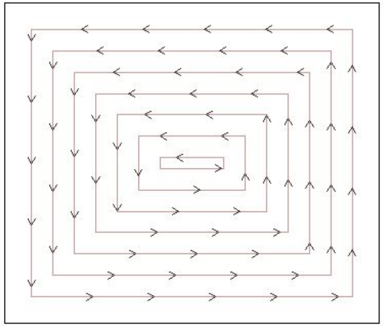
You save your GCode from your preferred CAM software. VCarve and Fusion360 can be setup to export code ready to run on the Makersmiths Big Blue or Big Red CNC machines. You can also manually write GCode if you're into that sort of thing.

You don't have to know how to write GCode, but it helps to be able to troubleshoot your projects.

GCode

<u>Gcode is a Programming Language</u>

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



Some Gcodes

Gcode Example Commands:

G0 = Rapid Motion (G0 X24 Y24 rapidly moves the machine to those coordinates)

G1 = Controlled Motion based on the set 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 (F100 sets Feed Rate to 100)

S= Spindle Speed (S12000 set Speed to 12000 RPM)

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

Some Gcodes

G92 X0 Y0 Z0 Sets a local zero spot for all axes.

Tn (n) is the tool number 1-8 so T1 loads cutting tool 1

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

M11C7 - vacuum pump on
M12C7 - vacuum pump off

Some Gcodes

G20 = all measurements in inches

G21 = all measurements in millimeters (But, EBF sometimes defaults to centimeters !)

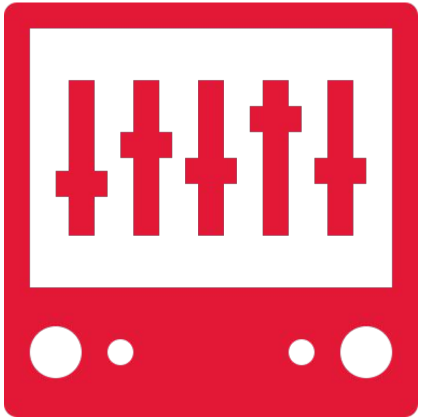
Macros: (sets of pre-coded instructions)

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 a Controller. The controller is the software that actually controls the machine. WinCNC is the Controller on the EBF CNC Machine, and Mach 5 is used on Big Red. Others include LinuxCNC, Flashcut, PlanetCNC, Mach 5, OpenBuilds Controller, and many others. The function of all is to convert a Gcode line into steps on stepper motors or other control switches.



Feeds & Speeds

Feed rate is how fast the machine moves around the table. Machines have a rapid feed rate which is how fast it moves when it isn't cutting. The controlled feed rate is how fast the machine moves while its cutting.

Speed is how fast spindle spins - usually in the 8,000 to 20,000 RPM range depending on the machine, the endmill, and the material to be cut.

A balance of Feeds and Speeds is key to getting clean cuts and extending the life of your end mills.

Feeds

If the Feed rate is too high relative to the Speed, you get deflection (bending) of your end mill, a reduction in end mill life, racking of the machine, excessive pulling on your workpiece, and likely a broken end mill.

If the Feed rate is too slow relative to the Speed, you can burn your piece, melt it (plastics), or just make your project take really long to complete.

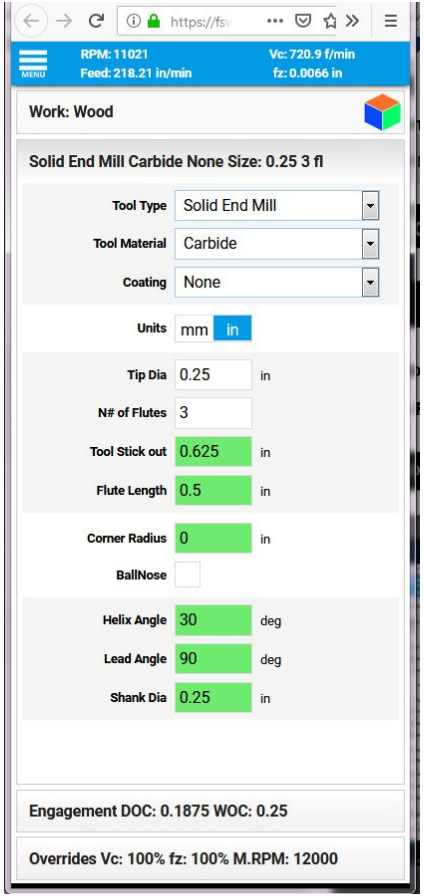
Speeds

If the Speed is too high relative to your Feed rate, you can burn or melt your work and puts a lot of strain on the machine when using a heavy/wide end mill.

If your Speed is too slow relative to your Feed rate, the end mill isn't cutting away enough material as it moves which will result in ragged cuts, possibly broken end mills, excessive pulling of your workpiece possibly dislodging it. 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

Depth of cut is how much material is removed on each pass. Multiple passes are usually required to cut all the way through a workpiece.

- 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

Videos

Some videos on Feeds and Speeds:

- Intro to CNC - Part 5: Feeds and Speeds - 7:40
- Calculating Feeds and Speeds A Practical Guide - 12:33

Starting Values

For wood projects on both Big Red and the EBF, using a ¼ diameter end mill, a 12,000 RPM Speed rate, 100 IPM Feed rate, and a cut depth equal to the radius (½ diameter) of your end mill are good starting values.

Smaller end mills may require lower Feed rates while larger end mills can be run at the same settings.

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 an 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.

End Mills

- You can use regular router bits in a CNC machine.
- Depending on the brand, the cutting surfaces may be carbide with the rest being high speed steel.

End Mills

- Do not use router bits with bearings on the CNC machines.

End Mills

- Many people use solid carbide end mills for CNC because they stay sharp longer. However they are more expensive.
- They come in multiple sizes, shapes, and with coatings to reduce friction.
- While the end mills on this page look the same, one is upcut and one is downcut - this equates to which way the spiral turns when it is in the machine.
    - 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

Endmills

FIRE HAZARD!

- Endmills can do become dull. Bear in mind that the CNC machine (particularly the 10 horsepower spindle on the EBF) 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.

½ and ¼ End Mill

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 fastest feed speed.

¼ end mill = .125 AT MOST depth per cut.

½ end mill = .250 AT MOST depth per cut.

Videos

The following videos focus on wood CNC end mills

- Bits and End Mills for Beginners - 28:39 - long video but goes into a lot of detail
- How to Choose the Best Bits for your CNC - 6:42 - made by Freud, an end mill manufacturer
- CNC Bits for Beginners - 8:50 - video on a few types of end mills for wood CNC projects

Workflow

<u>Sample Sign Workflow</u>

- Draw sign in Inkscape to correct dimensions
- Save as SVG
- Import SVG into VCarve
- Create toolpaths in VCarve
- Export GCode from VCarve
- Setup CNC machine
- Open GCode in controller
- Run project

Your workflow may be different depending on the programs you use, but they will always involve CAD, CAM, GCode and a Controller.