CNC Overview
- Design
- CAD
  - Sketchup
- CAM
  - Sketchucam
  - Gcode
- CNC Interface
  - Gstreamer to GRBL
  - Mach3
- CNC machine
- CNC wiring

Why CNC?

- fast
  - rapid prototyping
  - spindle motor applies 500W of energy to cutting
- accurate
  - tight joinery
  - easily machine parts that connect to bearings and motors
- reproducible
  - make 1 or make 100
- wide range of materials
  - plastic
  - wood
  - ply wood
  - MDF
  - aluminum

Design
- Philosophy
  - Try often. Fail often. Learn.

- Collect information
  - select a material
  - calipers
    - measure mechanical parts
    - measure material thickness

- Draw a paper sketch
  - think about how the parts will be assembled.

CAD
There are many computer aided design tools to choose from. Each tool offers a variety of features and are available for free or through buying a license. Assuming that you are new to CNC I have found Sketchup to be a good CAD tool. Sketchup is free for non-commercial use and if you stick to 2D design the learning curve is fairly simple. I started with Sketchup several years back and I still use it today because it is a quick and easy to tool to design with. Sketchup also has a free plugin that will allow you to generate gcode files.

I personally like to design with pen and paper sketches so I am using Sketchup as a drafting tool to capture and document my designs.

CAM
Computer aided manufacturing is the process of converting a CAD file into a set of machine instructions that will drive the physical CNC machine. The output file from the CAM process is a file of gcode commands. The first things to consider are which mill bit and what material type and thickness you plan to use. These parameters are important because they define how the CAM tool will generate cutting paths. For instance the mill bit diameter and length define the cutting kerf width and the maximum possible cutting depth. The CAM software uses the mill diameter to offset the cutting path so that the edge of the mill bit actually cuts along the lines defined by you CAD drawing. The CAM tool allows you to define the mill diameter and material thickness for each job. These properties are job specific.

The mill bit diameter creates a kerf in the material as the CNC job cuts through the work piece. The CAM software offsets the cutting path to accommodate for the kerf. There are two cutting options depending on which side of the CAD drawing line you want the kerf to be on. Inside cuts, will cut along the inside of a closed object which will create a hole. Outside cuts will cut along the outside of a closed object to cut out the outline of a part.

CNC Control Software

I will describe two different ways of interfacing to a CNC machine. The GUI interfaces offer similar features. The GUI needs to be able to provide a way to jog the CNC machine so that you can position the mill bit over the work piece. The XYZ position can then be zeroed so that all cutting will be relative to the origin you have set. The GUI also needs to allow you to select a gcode file that you will use to cut the part. Finally start, stop and pause buttons are provided to control the CNC operations.
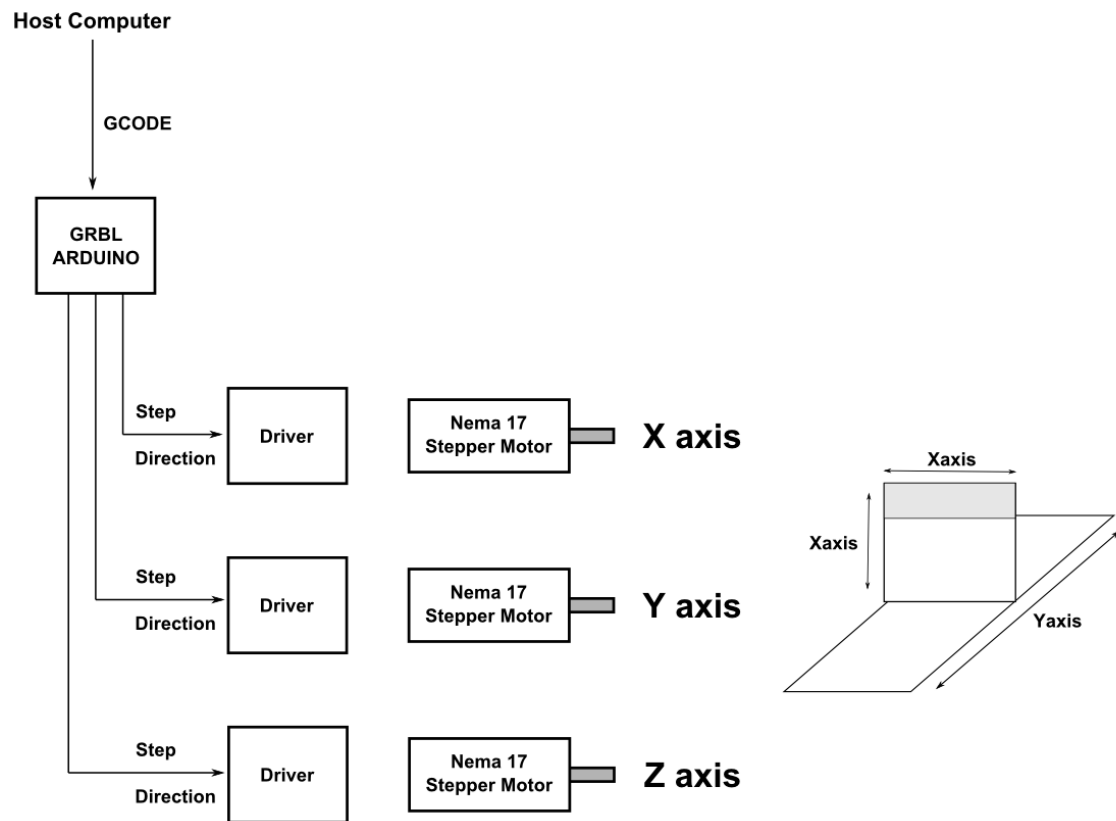
Mach3

Mach is a Windows based application that converts gcode into step and direction signals that drive the CNC machine. This is a well established tool, however, the physical interface to the CNC machine uses a 25 pin parallel port which is getting rare. If you choose to use Mach, you will need a Windows computer with printer port dedicated to your CNC machine.

Gstreamer/GRBL

The open source community has provided a two part solution to connect a computer to your CNC machine. An application called Gstreamer is installed on your computer. Gstreamer supports Linux, Windows and OSX operating systems. The gstreamer application provides a GUI interface that allows you to send commands and gcode to a serial port.

The second part of the solution is an Arduino based GRBL controller which receives the serial commands sent from Gstreamer. GRBL interprets the serial commands in real time and generates the step and direction pulses that drive the physical CNC machine. The advantage of this setup is that you can connect to your CNC machine using a USB to serial cable which is available on virtually every computer or laptop.

CNC machine using Gstreamer/GRBL

Host Computer

GCODE

| GRBL ARDUINO |

Step
Direction
| Driver | | Nema 17 Stepper Motor | **X axis**

Step
Direction
| Driver | | Nema 17 Stepper Motor | **Y axis**

Step
Direction
| Driver | | Nema 17 Stepper Motor | **Z axis**

Xaxis

Xaxis

Yaxis

Managing Complexity
- CAD (Sketchup)
  - start with simple 2D designs
  - lines
  - rectangles
  - circles
- CAM (Sketchucam)
  - 1/8" end mill
  - inside cuts
  - outside cuts
  - tabs
  - export gcode
- Machine setup
  - get help from the HACK613 community
  - GRBL and MACH both require machine dependant setup which is complex
- Machining Parts
  - Keep your first jobs short
  - Don't waste time or material on a large project until you master a small project
  - Practice cutting with air cuts and simulation (Camotics)

Terms

CAD - Computer Aided Design
- 2.5D - X,Y two dimensional machining with a few pocket features that use the Z axis.

CAM - Computer Aided Manufacturing
- pocket - a milling operation that cuts into the surface of the work piece but does not cut through the full thickness.
- gcode - an industry standard set of machine movement commands. These commands are interpreted by the CNC machine controller to set the step and direction pulses which drive the physical motor axis
- inside cut - a milling operation that offsets the mill bit 50% of the bit diameter towards the inside of a closed line. This is used to cut holes.
- outside cut - a milling operation that offsets the mill bit 50% of the bit diameter towards the outside of a closed line. This is used to cut part outlines.
- centerline - a milling operation that leaves a kerf of 50% of the bit diameter on either side of a line.

CNC - Computer Numerically Controlled
- feeds and speeds - the feed rate defines the velocity of the spindle while cutting (in/min or mm/sec) and the speed is the rotational rate of the spindle in rpm
- chip loading - describes how effectively the mill bit flutes can remove material from the kerf. This describes the combined effect of the feed rate, spindle speed and the flute count/diameter of the end mill.
- fixture - a clamp to hold down your work piece while cutting.
- jog - using the CNC GUI interface it is possible to move the spindle around manually by sending step and direction commands. These moves are called jogs. You will use the jog to position the spindle over your work when setting the origin.
- zero - this defines the origin of the job. All gcode operations are reference to the origin at X=0 Y=0 Z=0
- job - a single gcode file that will be used to cut your work.
- ER11 collet - this is an 11mm nut that holds the bit in place by using a variety of collets. Each collet is designed to accept a specific mill bit diameter ie)1/8" or 1/4"

Mill bit properties
Mill bits are not the same as drill. A mill bit is designed to cut through material on the edge of the bit. While drills only cut at the tip as they cut vertically into a work piece.
- flutes - the number of spiral grooves in the mill bit
- up cut mill bit - removes material from the work piece by causing the cut chips to rise in the clockwise rotation of the mill bit flutes. This keeps waste material from clogging the cutting kerf.
- down cut mill bit - removes material from the work piece by causing the cut chips to be driven down in the clockwise rotation of the mill bit flutes. This causes the waste chips to be forced into the kerf. Normally you only want this for full depth cuts of thin material as the downward force keeps the work piece from being lifted up.