CNC (Computer Numerical Control) is a process that uses a computer to take input from a design or pattern and then translate that into machine specific commands to produce the desired parts. This type of control comes in many shapes and sizes, but at its core it is all about taking an input, such as the coordinates of where the machine head needs to be, and turning them into commands for moving those components at different speeds. The final product would be called G-code. So for this article we will be looking at a USB controller and CNC software (Nest) and the G-code that it produces. This example is using a JigBASIC program that can control any machine that has a USB interface. This is the only thing we need to buy as most machines come with these already, if not they can be purchased as per our needs (prices will vary per machine model). We also need to make sure we have some sort of "map" or "coordinate" to translate those coordinates into commands. Some machines like an A0 or A2 plotter will need different G-code when compared to a Sylica CNC plotter, but for this example we will go with a "generic" plotter. To start you need to make sure your USB controller is properly set up, drivers are installed etc. etc. Basically make sure it works before trying to make G-code for it! The software used in this article is called Nest and can be downloaded from the link below:. It can be downloaded trial version without the license information ($10), or the full licensed ($35). The only other feature that is different between the two is the ability to export g-code from a larger number of programs. http://www.usnest.com/usb-cnc/software/index.html The next step is to load our g-code into Nest, for this example we will use a file called "jigbasic.gcode" from their website which can be downloaded from here: http://www.usnest.com/downloads/#gcode . You may have to rename the extension from ".gz" if needed, but for this example it will work as it is assuming you have Linux installed and all files are in their default directory location (/usr/bin). If this is not the case you can find the required files by clicking the link below: http://www.usnest.com/downloads/#gcode . Now we need to import our g-code into Nest. We can do this either manually or automatically by using an "automatic feeder" (AF). The only thing we need to do is first create a new file and name it as "file.gcode". Next we will right click on this file and select "auto-af". This will tell Nest that we want it to read this file and then send it as G-code to the machine.

228eeb4e9f3262