

Setup

Install drivers for 3018

Most 3018 CNC will use something like a woodpecker control board. Many of these boards use an inexpensive chip called the CH340 to talk to your computer. To get them working you'll want those drivers from the [manufacturer \(Note: site is in Chinese\)](#) or from another source like [Sparkfun](#).

Notes:

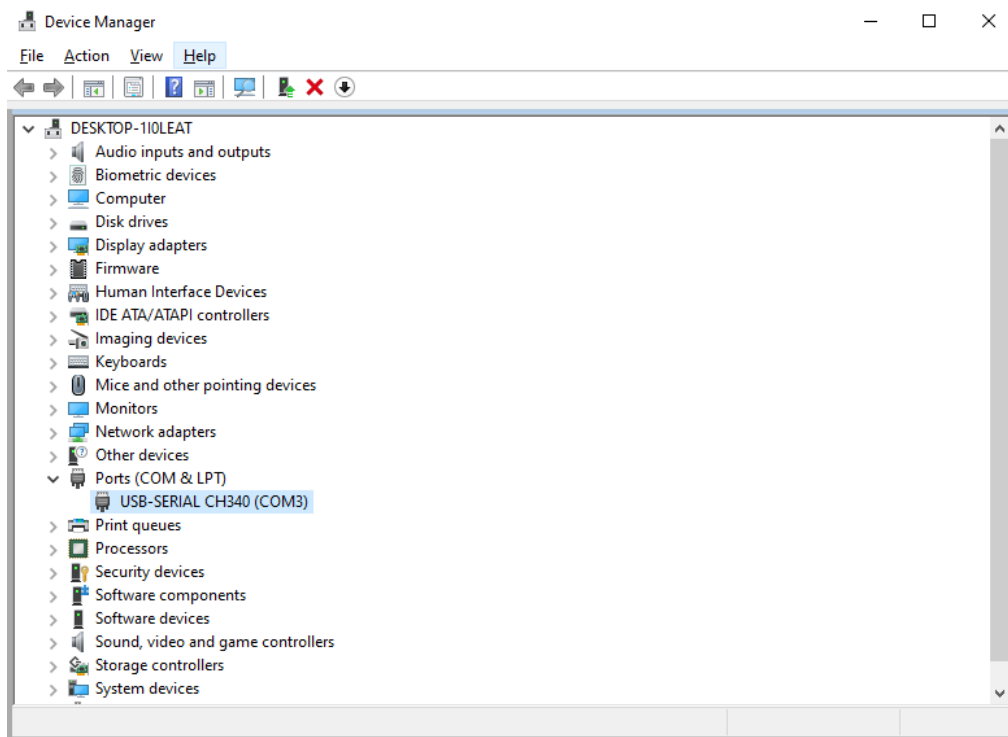
- Make sure you get the latest versions of the drivers, preferably straight from the manufacturer.
- You may get some strange messages during install: try again until you get actual 'success' message

Check the connection to the machine

After the drivers are installed test the connection in Windows:

Connect the device's USB cable to your computer

Use the Device Manager program found in Control Panel to find the COM port number:



If the COM shows up the driver is fine.

Controlling the machine with Candle

The 3018 control board is running firmware called '[GRBL](#)' (pronounced 'gerbil').

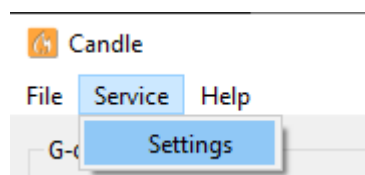
Your computer sends instructions known as 'G-Code' (aka Numerical Control or 'NC' Code) to the control board

Candle is a free G-Code sender that helps you do this. You can download it here:

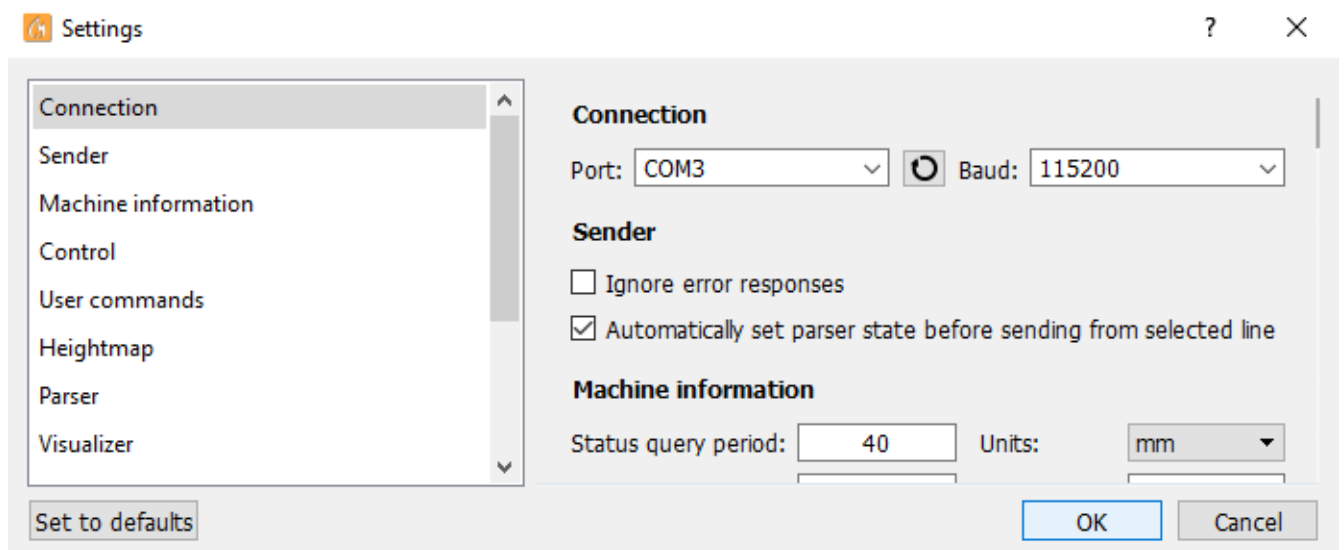
<https://github.com/Denvi/Candle>

<https://cncphilosophy.com/candle-grbl-software-tutorial/>

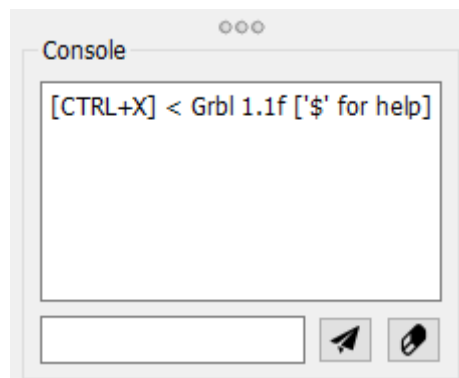
It's used to reposition the machine over your materials manually (called 'jogging') - this is a good way to know the device is working. Candle is also used for sending whole cutting programs.



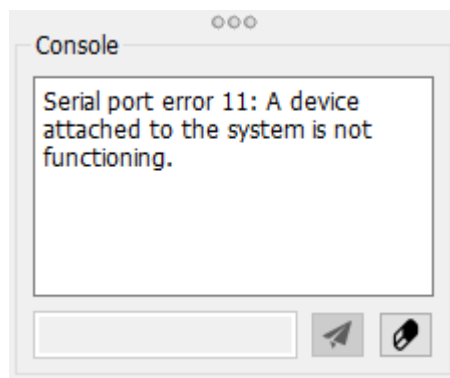
To set up a connection, select the COM port number found above and use 115200 BAUD:



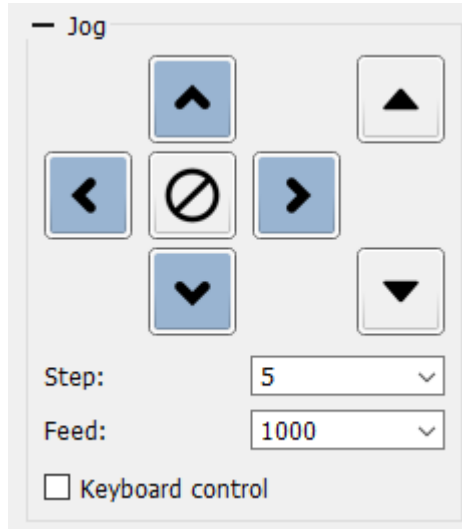
If you connect successfully you should see:

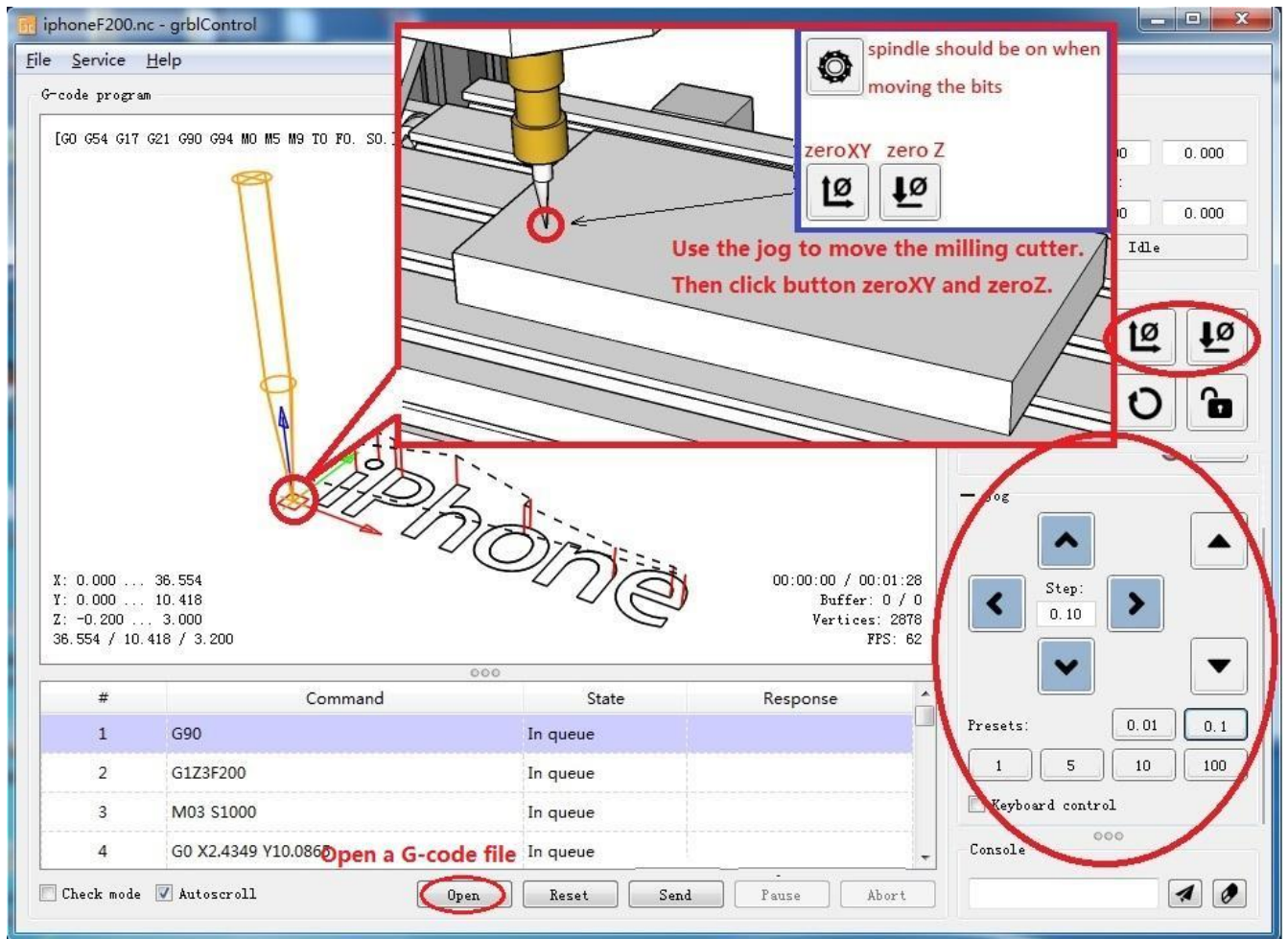


If not you may get something like:



After connecting, jog the device to center it on a workpiece. You may want to adjust the step setting depending on how fast/far you want the spindle to move each time you click a button:





Modelling and creating drill paths:

Fusion360 by Autodesk is a modelling and drill path planning software:

<https://www.autodesk.ca/en/products/fusion-360/personal>

It's free to use for personal use.

There are other alternatives like [Easel](#) but Fusion is a professional quality tool that does drill paths pretty well and has a lot of features.

It's included by default but if you need a reference the post processor Fusion 360 uses to generate g-code is here:

<https://cam.autodesk.com/hsmposts?p=grbl>

Another good reference:

<https://www.youtube.com/watch?v=ab8flNmpkbY>

Drawing

If you'd like to convert artwork to engravings (maybe for a print) [Inkscape](#) is a handy tool to have

You can use it to draw vectors directly, edit them or [trace raster drawings from scans or digital art](#)