



Simple Cost Effective Designs.

## E3 CNC Router Quick Start Guide

The purpose of this document is to give those new to CNC routing a quick start guide to help you can get your E3 CNC router up and running. This guide will also give you a very basic understanding of the CNC routing process.

Before starting the setup let me walk you thru the CNC routing process for the E3. You will need to understand the following processes to be successful with CNC routing. We have broken the processes into 3 different sections.

1. Create g-code – G-code is a text file that CNC machines use to coordinate the machine moves so that we get the finished product we want.
2. Ready the E3 – The E3 will need to be *homed* and set to the *starting work position*.
3. Send G-code file to controller – the file we create will be saved and opened in software to send to the controller

First you need to download some software and then setup the E3 CNC router. Please complete each step before moving to the next one.

### Installing the Arduino Driver

The E3 controller is an Arduino based controller and uses the Arduino UNO driver. The drivers can be downloaded with the Arduino IDE from the Arduino website. They also have an instruction page. We have also included a video to walk you thru it.

Arduino IDE (with drivers)

<https://www.arduino.cc/en/Main/Software>

Arduino Instructions for Windows

<https://www.arduino.cc/en/guide/windows>

BobsCNC Video walk thru

<https://youtu.be/FMmzWciWImU>

## Checking the Device Manager

Open the device manager and look into the “Ports(COM & LPT)” and ensure that the Arduino Uno (COM X) is displayed. Please note the COM port number as we will need this later. If you do not see the driver or want more details, please watch the video and install from the driver folder.

BobsCNC Video walk thru

<https://youtu.be/OSCxDyWG-XI>

## Installing the Universal G-code Sender Platform Version

We recommend Universal G-code Sender Platform Version for use with the E3 CNC router. We can go to the UGS website and follow the “How to run” section.

UGS Platform

[http://winder.github.io/ugs\\_website/guide/platform/](http://winder.github.io/ugs_website/guide/platform/)

UGS Platform download

[http://winder.github.io/ugs\\_website/download/](http://winder.github.io/ugs_website/download/)

BobsCNC Video walk thru

[https://youtu.be/trM1dm\\_DXk4](https://youtu.be/trM1dm_DXk4)

## Introduction to UGS Platform and Connecting the E3 CNC Router

Once the software is loaded connect the E3’s USB to your computer and open the UGS Platform. Then connect the E3 thru the USB to UGS. UGS is an excellent choice for the E3 and has many features to aid in the CNC process. We cover the basic’s in the video. You will want to explore to find more.

BobsCNC Video walk thru

<https://youtu.be/nxkoFwDeSyA>

## Verifying Machine Setup

The next step will be to connect and home the machine. This step must be performed every time you connect the E3 to the computer. Homing the router sync’s the computer and the E3. An outline of the process is to:

1. Connect the E3 to UGS
2. *Home* the E3
3. Move to the workpiece g-code starting point
4. Re-zero all the axis
5. Load the g-code file and run the program.

BobsCNC Video walk thru

<https://youtu.be/X78SN9WNdrs>

## Installing CNC software to create G-code

We recommend F-engrave to start creating g-code files.

BobsCNC Video walk thru

[https://youtu.be/N\\_RIRxXv1o8](https://youtu.be/N_RIRxXv1o8)