

16CH FSCV Interface Board v1 - Manufacturing

Instructions

Introduction

Included in this document and zip file are the necessary files for a company to manufacture the enclosed printed circuit board (PCB). Also included is a simple technical drawing that some manufacturers may request.

Most companies will first ask a few basic specifications about the board before uploading the design files located in the **Gerber and Drill Files** folder. In the past we have used PCBMinions to manufacture this PCB and the instructions and specifications below are tailored to this company. However, other manufacturers can be used, though their ordering input specifications may differ.

Viewing the Eagle Files

The .brd and .sch files can be opened by Eagle. Licensing will vary by institution/individual.

Viewing the Gerber and Drill Files

The files contained in the **Gerber and Drill Files** folder can be viewed using gerbv (<https://sourceforge.net/projects/gerbv/>)

PCB Specifications

Below are the specifications of the board that must be defined for manufacturing. All other specifications, such as quantity and color, are up to the end user.

- **Dimensions:** 20.4mm x 48.3mm
- **Layers:** 2
- **Minimum Trace/Space:** 6mil
- **Minimum Hole Size:** 10mil
- **Thickness:** 1.6mm
- **Surface Finish:** HASL with lead
- **Copper:** 1oz
- **Material:** FR4

Files to Upload

Once all of the options have been selected, zip the contents of the folder **Gerber and Drill Files** and upload this file.