B A S T L
INSTRUMENTS

# CLUTCH v1.0 - Assembly Guide

bastl-instruments.com

INTRODUCTION

This guide is for building Clutch module from Bastl Instruments. It is good to have basic soldering skills and to be able to identify electronic components before starting this kit. However if you have never soldered before, check out this tutorial first[1]. We even included one of the best quality solder we have found to help you solder everything faster and better.

The Clutch module consists of just one board. All the parts comes in two bags separated for Bottom board and Assembly parts. See Bill of Materials (BOM) for detailed list.

---

[1] http://www.instructables.com/id/How-to-solder/

ASSEMBLY   BOTTOM
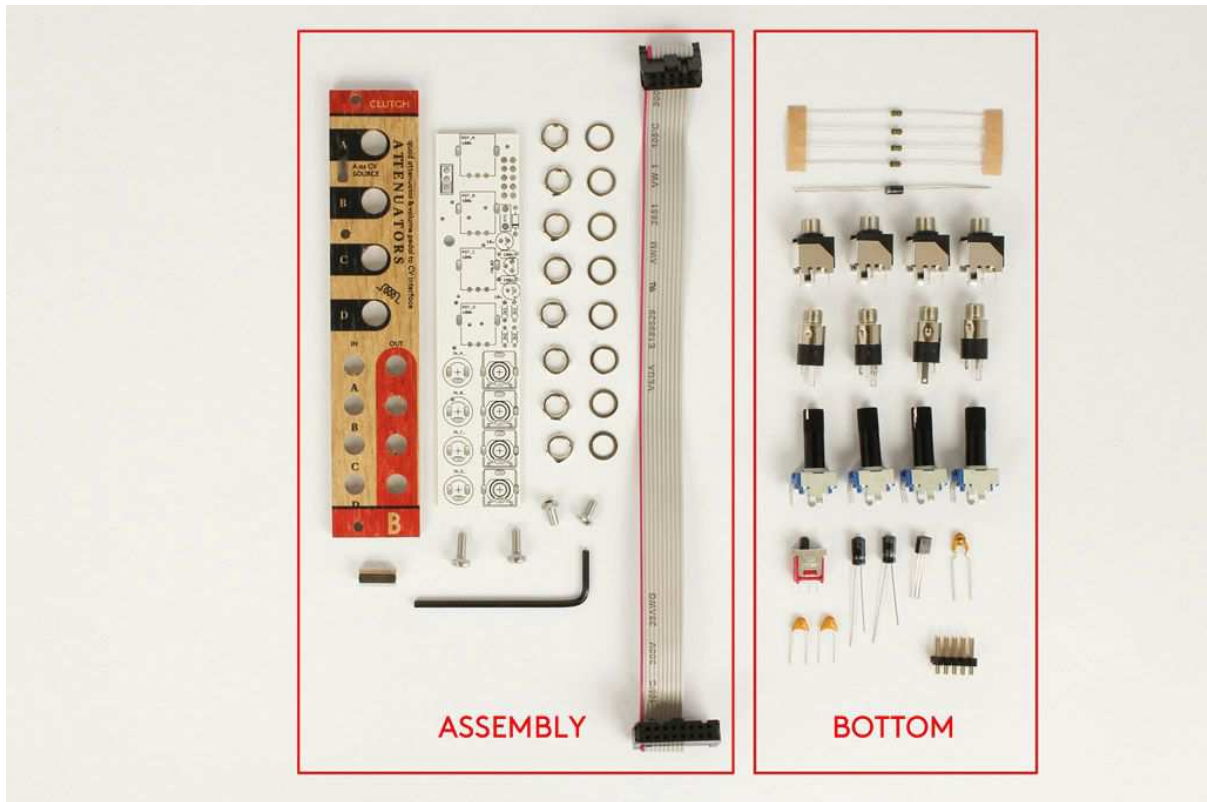
Before starting this kit, prepare the following tools:
- Soldering iron (15-20W)
- Flush cutters
- n2. hex screwdriver or allen key (enclosed with kit)
- Phillips screwdriver (cross)
- Wrench No. 8
- Protective eyewear

We suggest that you work in a clean and a well lit and ventilated environment to avoid accidents or losing any of the small components.

Also briefly go through this guide and make sure that you understand all the steps.

BOTTOM BOARD

Let's start with the soldering of **resistors and diode**. There are only four resistors of one value (2k2) and one 1N4007 diode. Insert them on respected places and solder them. **Watch out! Diode is polarized**. Make sure that the marking ring on the diode body matches the marking on the circuit board. Then snip the leads as close to the PCB as you can (be sure to make this step on all remaining leads in the course of this guide).
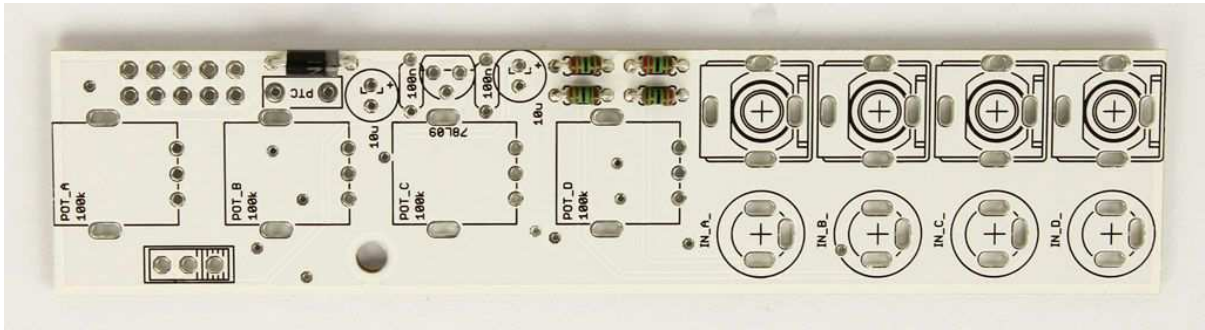
Now add other small parts. Let's do this in the following order:

- **100nF ceramic capacitors** (2x, marked "104")
- **78L09 voltage regulator** (1x, watch out for **orientation** here - flat side of the component has to match the outline drawn on the PCB)
- **10µF electrolytic capacitors** (2x, these are **polarized**, too! There is a plus (+) sign on the PCB that should match the longer lead of the capacitor)
- **100mA protective fuse** (1x, looks quite similar to ceramic capacitors)
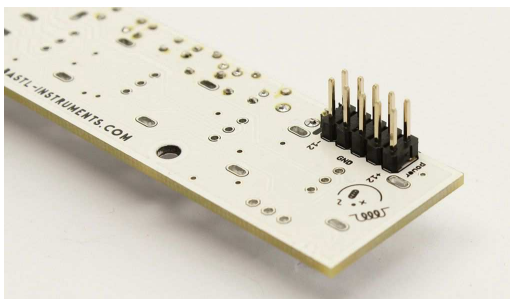
Turn around the PCB and solder **2x5 pin male pinheader**. Be careful to solder it straight. You may first solder one of the pin, then take the board in your hand and re-heat that pin while pressing down on the header to align it (be careful though, you don't want to touch the pin you are heating up). Wait for it to cool and solder the rest of the pins.

Turn the PCB back around. Secure the **spacer** with the **screw** from the bottom. Then insert one of the last components (**don't solder yet!**):
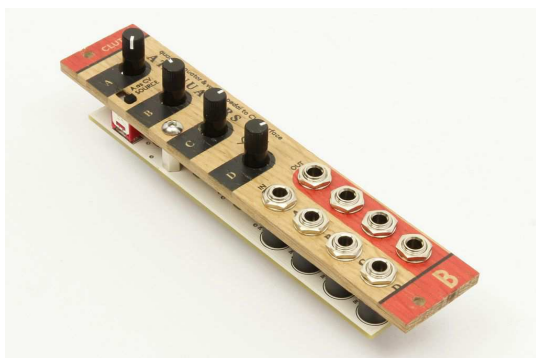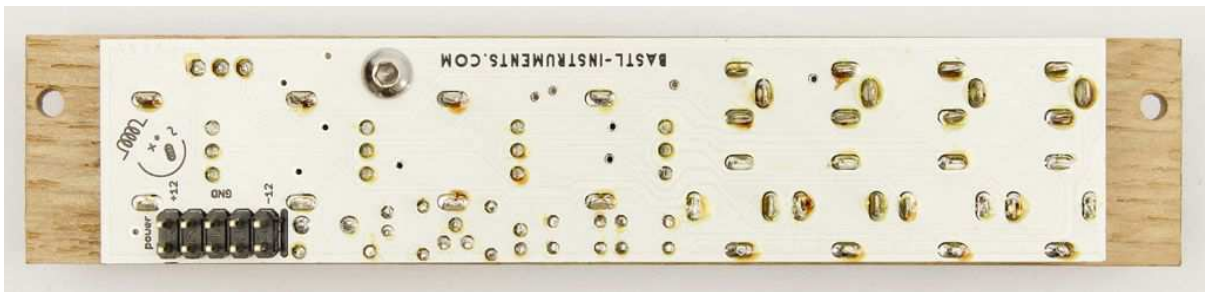
- **Potentiometers** (4x)
- **Mono jack connectors** (4x)
- **Stereo jack connectors** (4x)
- **Switch** (1x)

Lay down the **front panel** on the board and add one **screw**, eight **washers** and eight **nuts**. (don't tighten the jack nuts too much as you may damage the panel!).

Check that all all the components came through easily (especially the switch). Finally you are ready to **solder rest of the components**.

Congratulations! You have made it through. Before you connect anything, make sure that your system is disconnected from power. Also double check the polarity of the ribbon cable, the red cable should match the -12V rail both on the module and on the bus board!

TROUBLESHOOTING

First check out the DIY F.A.Q. on our website.

If you are having some more trouble, the best thing is to take a nap! Especially late at night!

If you are still in trouble you can send the detailed description of the problem with enclosed high-resolution photos on diy@bastl-instruments.com.

If you think that you are unable to make the module work on your own, consider our "Come to Daddy" service.