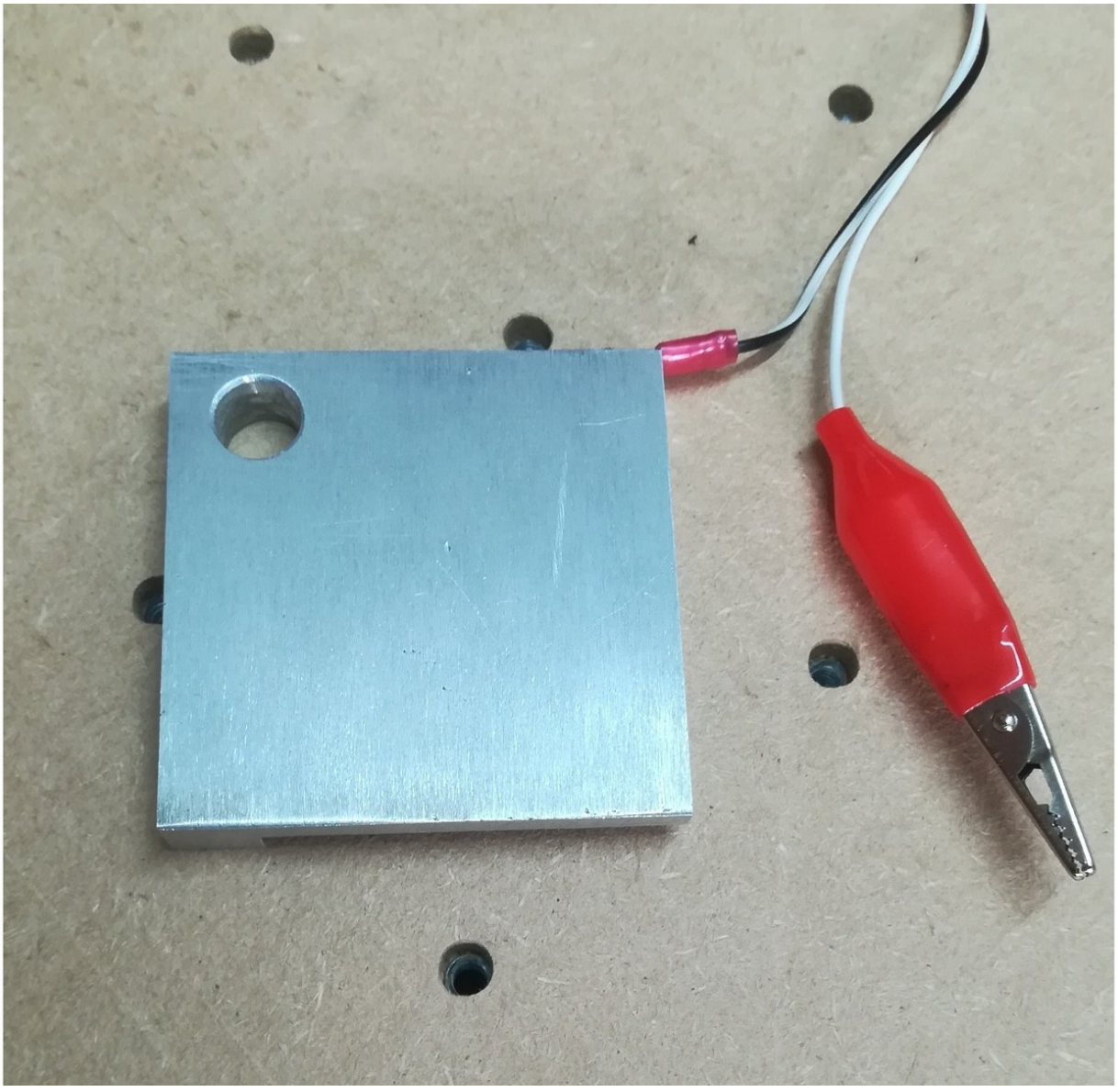
Setup a XYZ Touch Probe with a CNC xPro Controller

How to wiring and set-up a Touch Probe on the XPRO v3 controller.

Written By: James Fordham



© 2023

INTRODUCTION

In this guide we will run through the steps to connect and configure a touch probe to your WorkBee or OX CNC running an XPRO controller.

TOOLS:

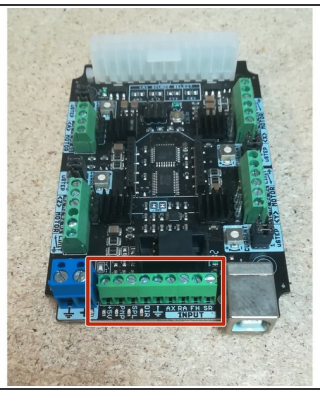
- Small screw driver (1)

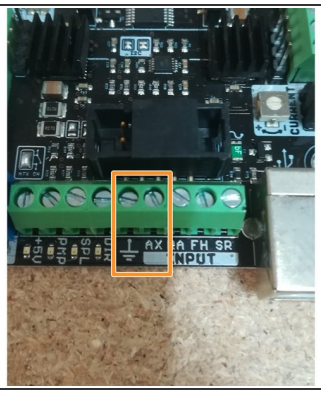
PARTS:

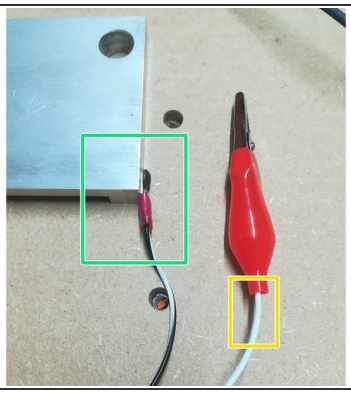
- Touch probe (1)
- CNC-xPro (1)

© 2023

Step 1 — Wiring the Touch Probe



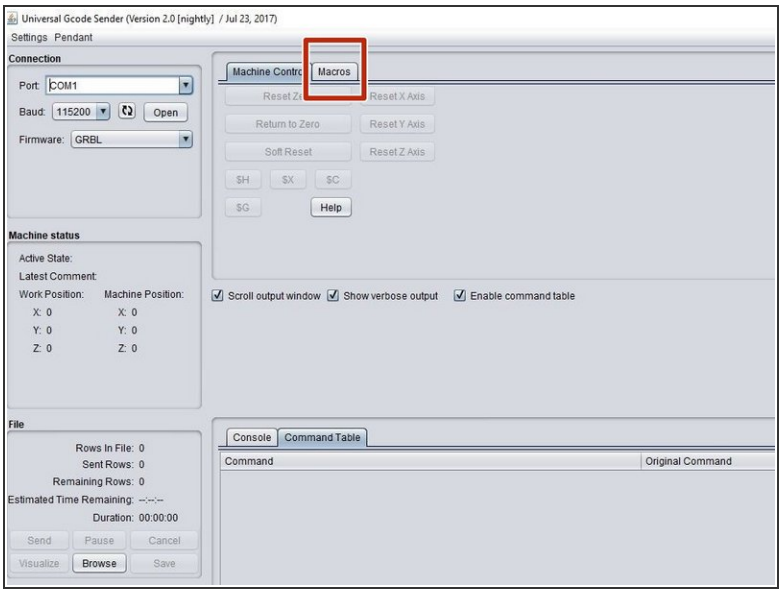


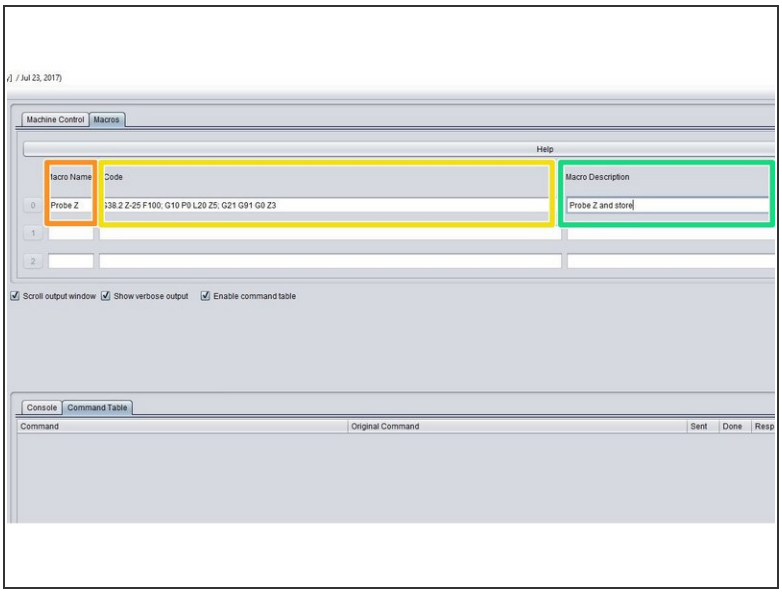


- Locate the Input screw terminals on the XPRO board.
- Identify the Ground and Touch Probe inputs.
- The wire attached to the crocodile clip, should be screwed into the ground terminal.
- The remaining wire (attached to the Probe) should be screwed into the probe terminal.

© 2023

Step 2 — Setting up the Macros





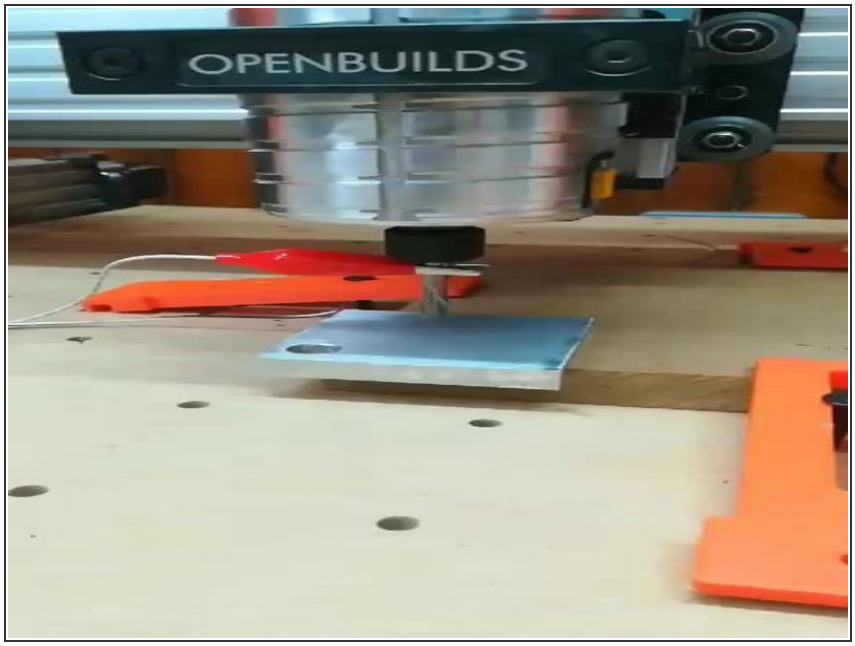
- Open up Universal G-code Sender (UGS) & connect to your WorkBee or OX CNC as normal.
- Locate and click on the Macros tab.
- ⓘ On this tab we can set-up a number of macros to work with the Touch Probe.
- The first macro we will make is to probe the Z axis and store this value.
- In the Macro Name field, give your macro a name. In our case we will use "Probe Z".
- The next field is where we enter the GCode for our Macro. To probe and store the Z axis information we enter the GCode below. These commands are explained in the documents at the bottom of this guide.
- ⓘ G38.2 Z-25 F100; G10 P0 L20 Z5; G21 G91 G0 Z3
- Lastly we can give our new macro a description such as "Probe Z and store"

Step 3 — Adding more Macros



- To add more macros follow the steps above and use the PDF and spreadsheet documents at the bottom of this guide to configure your G-Code commands.

Step 4 — Using your macros



Now that you have set up your macros, for each size bit you want to use (1/4, 1/8, 1/16 etc) it's time to

put them to action and find your zero point.

- Once you have set up your job, ensuring the work piece is clamped down securely, then home your machine.
  - Place the touch probe on the bottom left corner of your work piece.
- Now use the jog axis commands to centre the bit within the probe. Then attach the crocodile clip to the bit in your router.
- You can then run your selected macro by pressing the correct macro number.

ⓘ In this example we ran the macro to probe X, Y and Z for a 1/4" bit. When the macro had finished we told the WorkBee CNC to return to the zero point.

You are now set to use a touch probe and easily locate your work piece corners.