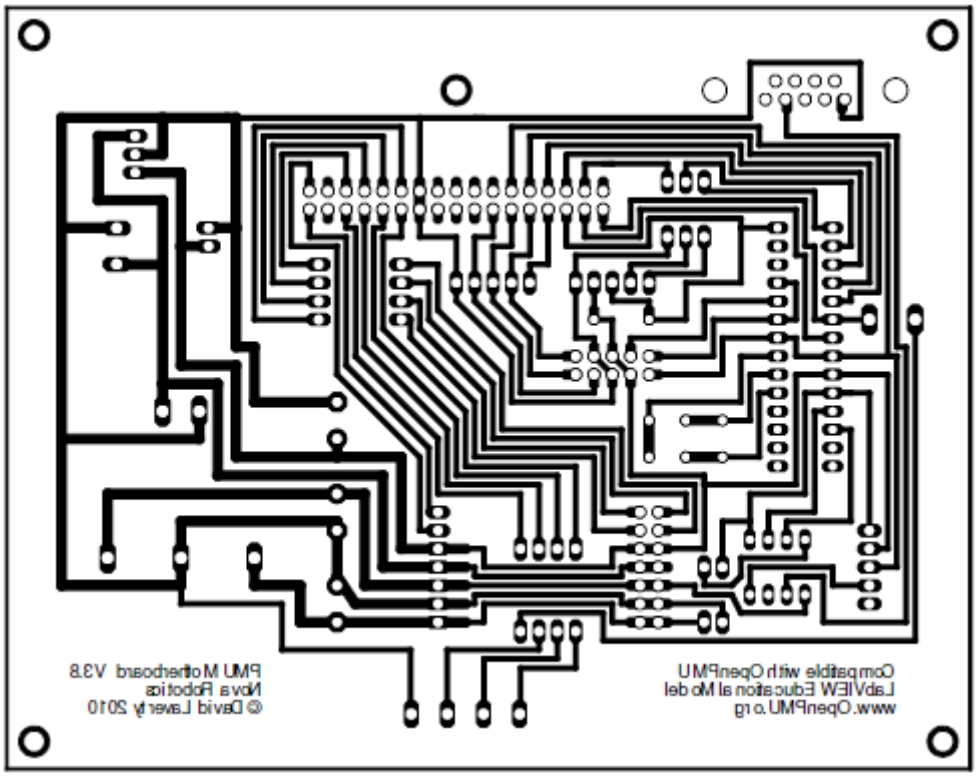
OpenPMU

Open Source Phasor Measurement Unit



PCB Assembly

Document Version: V1.1 2012-05-29

Abstract:

This document contains details you will need to fabricate and populate a PCB for use with OpenPMU and a National Instruments DAQ (USB-6009-OEM).

Authors:

David Laverty

Karl Weisser

Document History:

2012-05-29	V1.1	Updated Title and Logo (This Version)
2012-05-28	V1.0	Original Version

OpenPMU PCB Assembly Instructions V1.1

This document contains details you will need to fabricate and populate a PCB for use with OpenPMU and a National Instruments DAQ (USB-6009-OEM).

IMPORTANT NOTE: These PCBs are for use at mains voltage. ***Do not assemble or use*** these unless you are working in a laboratory environment and have appropriate safety training.

For hobbyists and those without safety training, use only the non-mains voltage model for signal generators.

Components

You will need to decide which version of the board you would like to assemble:

1. Hall Effect Model
   - Can monitor mains supply (up to circa 260 V_{rms})
   - Very accurate
   - Expensive
2. Transformer Model
   - Can monitor mains supply (up to circa 260 V_{rms})
   - Fairly accurate, introduces small distortions at higher voltages
   - Very cost effective
3. Signal Generator Model
   - Best when monitoring mains supply is not requires
   - Directly monitor voltages up to ± 10 V_{peak}
   - Safest for teaching environments
   - Cheapest

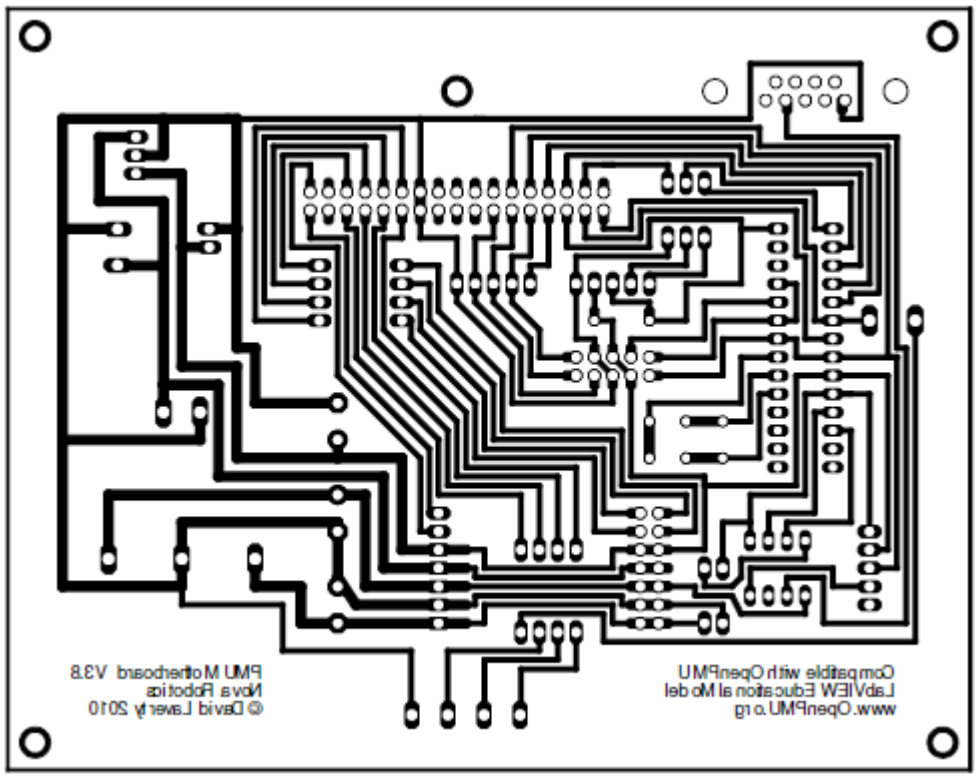
Component listings for each model are available in the attached Excel file - click here.

PCB Fabrication

Download the set of PDFs for the OpenPMU LEM PCBs - click here. PCBs are single sided and can be produced in even basic PCB fabrication facilities. If you would like the PCBs in different formats, please contact the author via the user account "clockdoctor".

Select and fabricate the PCB relevant to the design you would like to build.

There is also an optional front panel indicator PCB. It is recommended this PCB is fabricated and populated also.



PCB Population

Start populating the PCB, following the component placement schematic by soldering the:

1. IC holders
2. Screw terminal posts
3. IDC header posts
4. LEDs
5. Voltage Regulator
6. Capacitors
7. Resistors
8. TRACO power unit (necessary only for Hall Effects / external signal conditioning)

Once populated, insert the RS232 driver and the PIC 18F252 microcontroller.

Attach the USB-6009-OEM using a bolt through the mounting hole. Make a ribbon cable to connect between the USB-6009-OEM and the PCB.

Apply a 12V power source to the power supply post. Check that the power LED indicator lights.

PIC Programming

Once the PCB is fabricated and populated, you will need to programme the PIC microcontroller (PIC 18F252). To do so, you will need the Microchip ICD2 or compatible.

Download the source code (**to follow**), compile and build in MPLAB (www.microchip.com/) and download to the microcontroller.

Note: Make sure that the ribbon cable is disconnected between the USB-6009-OEM and the PCB. Reattach the ribbon cable once the microcontroller has been programmed.

Testing

Download the test programme attached (**to follow**) to confirm that the various functions of the PCB are operating correctly.