A Revised High Voltage Board for the Consolidation of Front End Electronics on the Tile Calorimeter of the ATLAS Detector at the LHC

Robert Reed, Bruce Mellado

School of Physics, University of the Witwatersrand, Johannesburg, South Africa

E-mail: robert.reed@cern.ch

F. Carrio[1], P. Moreno[1]. C.Solans[2], J. Souza[3], A. Valero[1]

1 - IFIC Spain, 2 - CERN, 3 - UFJF Brazil

Abstract. A high voltage board, located in the mobile drawer integrity checking system (MobiDick4), has been redesigned to incorporate temporary fixes. The revised board will be part of the new mobile testing equipment which will consolidate the front end electronics on the ATLAS detector and verify their working status for data taking in 2015. The boards will be manufactured in South Africa and tested at the electronics laboratory located at the University of the Witwatersrand.

1. Introduction

The world's largest particle accelerator, the LHC, confirmed the discovery of the Higgs Boson on 14 March 2013 [1]. The LHC consists of a 27km ring situated under the French and Swiss border and is home to The European Organization for Nuclear Research, known as CERN. The discovery was independently observed by two experiments the ATLAS and CMS detectors. ATLAS (A Toroidal LHC Apparatus) is the largest of all detectors, shown in Fig 1. It consists of a series of concentric rings namely the Inner detector, Tile Colorimeters and the Muon Spectrometers. These components are all used in unison to identify and determine the energy and direction of particles produced in the proton-proton collisions.

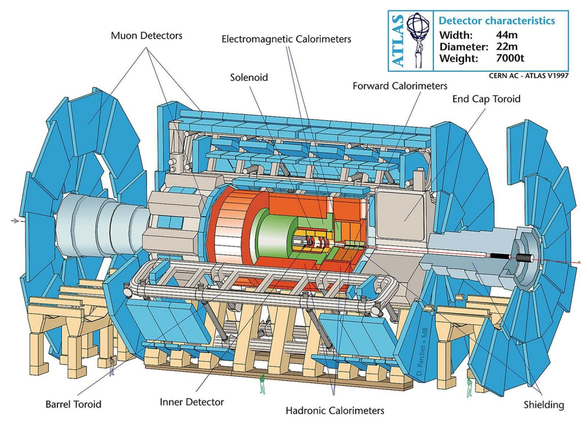

Figure 1: ATLAS detector showing structure of the sub-detector systems.

The Tile Colorimeter (TileCal) is primarily used to measure the energy and direction of hadrons and jets as they are produced. This is achieved by measuring light pulses that are boosted into signals by the photomultiplier tubes (PMTs). These signals are then processed by the front end electronics which determine if they are interesting events worthy of being permanently stored. This triggering system is housed in a super drawer. There are 256 such drawers in the ATLAS detector and during the long shutdown in 2013/2014 a consolidation integrity check will be performed on the electronics [2]. This is done by the Mobile Drawer Integrity Checking (MobiDick) system [3].

The new MobiDick4 includes several state-of-the-art daughter-boards which provide different functionality to the motherboard. This combination allows fifteen different tests to be performed on the super-drawers. The technology used in the HV board, like that of the LED board, is over ten years old. Many components are now obsolete and the overall design has had many patches made since its first version. This calls for a redesign of the board to implement temporary fixes and include newer components. The University of Witwatersrand has been involved in redesigning the HV Board and the LED Board to incorporate temporary fixes and patches to the boards since their first design. The prototypes are complete and plans for manufacturing models are under way in South Africa.

2. Previous Design

The first design of the high voltage (HV) board can be seen in Figure 2. This design had a few flaws which were fixed after installation. The first prototype was not completely isolated from the motherboard. This had potential issues such as current loop-backs. The LEDs were in an inconvenient location. The power supply could not provide the +5V that the board required. As a result there were a few temporary patches and ideas for the revision board.

These were namely:

- Optocoupler was added
  - Converts electrical to optical and back to Electrical
  - Isolates HV board from motherboard
- DC/DC Converter added to +24V cable
  - Provides +5V from the +24V rail
- LED connectors replaced with pins
  - Allows LEDs to be placed off the board

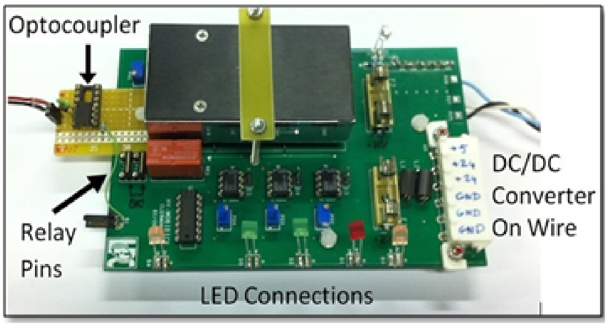

Figure 2: Previous design of the High Voltage board. Temporary patches have been applied.

3. Schematics and Component Layout

The patches above were incorporated into the new design. The schematics determine the logical connections between components and this is the first step in the redesign process. Along with the fixes described above there were a few additional things that were added to the board. The relay switch has pins which determine the on/off state. These were made to be semi-permanent (by use of zero ohms resistors instead of clip pins) so that they could be soldered into place which prevents accidental switching.

Figure 3 shows an snippet of the schematics and how the patches were included into the design. The schematics were captured from the existing PDF designs using Orcad Capture. New parts had to be designed and all patches were included into the logic. This required additional logic to be added into the board.

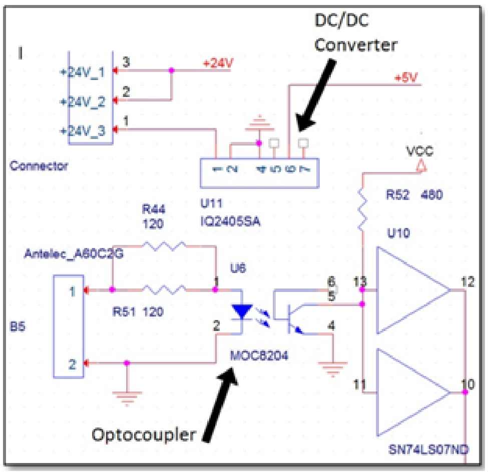

Figure 3: Schematics including the Optocoupler and Converter patches.

Once the schematics was completed the layout of the board was then considered. The steps followed were quite simple. The board geometry was setup (dimensions and restrictions such as heights). Each component was then placed in an efficient manner as to improve signal quality and reduce noise. The ground plane was set out so that all grounding was easily achieved using vias or placed directly on the plane. Once the grounding was complete the power lines were then routed followed by all inter-component connections. Figure 4 shows the physical layout of the board with all components and routes shown.

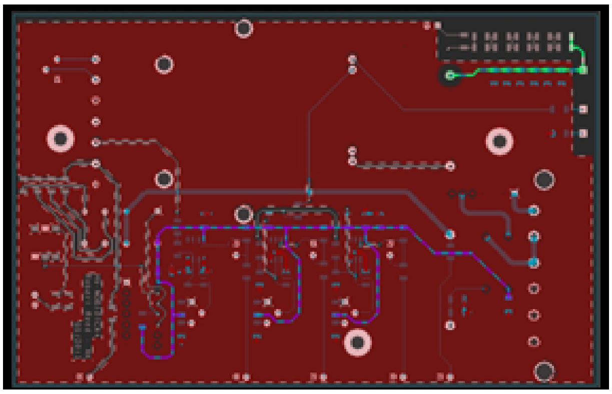

Figure 4: The physical layout of the board with all routing and components shown.

4. Production

The board was manufactured in Valencia at the Instituto de Fsica Corpuscular (IFIC). The smallest components were soldered first as to offer less obstruction when mounting the more complex and larger components. The typical size of the components can be seen in Fig 5. The soldering requires a steady hand to align the components correctly. During the soldering process various issues were addressed with the board (short circuits and incorrect component footprint). This is common with proto-type boards and it allows modifications of the board before it goes into production.

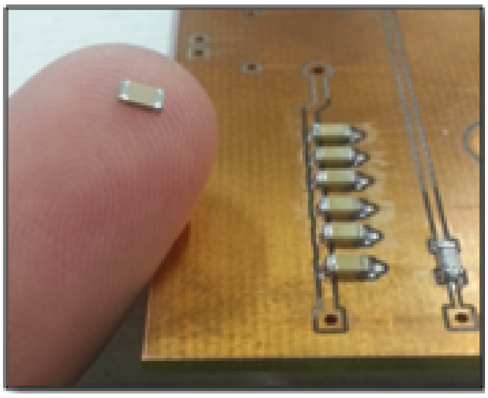

Figure 5: Schematics including the Optocoupler and Converter patches.

5. Testing

Before powering up, the board needed to be tested. The testing procedure went as follows:

- Check for shorts using multimeter and after mounting each component
- Check each component after soldering and fix any issues
- Power board up with no high voltage and check for short circuits
- Make sure all input signals for Ultravolt are in correct range
- Place Ultravolt onto board and measure output voltage to confirm it is -830V

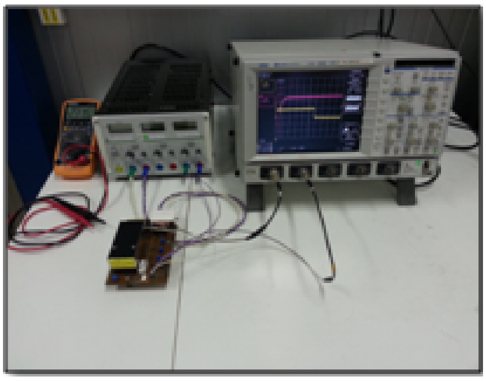

Figure 6: Setup for testing the high voltage board.

There were some issues with the board which were due to manufacturing faults but they were easily fixed. The board powered up with no issues and all input voltages for the Ultravolt were in the correct range. When testing the high voltage output a fault was found with a few capacitors which can be seen in Figure 5. The voltage rating for the components were incorrect resulting in the high voltage output being grounded. The components had to be replaced with a higher voltage rating in order to handle the high voltage of the output. After the board was performing as designed and producing -830 V the delay between the "on" signal being injected into the board and the actual high voltage being produced was then recorded. This was measured using an oscilloscope. The setup can be seen in Figure 6. The oscilloscope then allowed the data to be captured and plotted for analysis.

Figure 7 shows the results. The blue line is the input signal. The voltage rises to its maximum value which is the "on" signal. This signifies the beginning of the timing. The red line is the high voltage output. As can be seen it stays at zero until a certain point where it rises rapidly to its designed inverse voltage. This output is measured at -2.6 V since the oscilloscope was connected to the LED rather than the high voltage directly. This was to prevent burning the scope. The time delay between the input and output signals is 9.1 milliseconds. This is quite slow but does make sense since the relay is a mechanical switch.

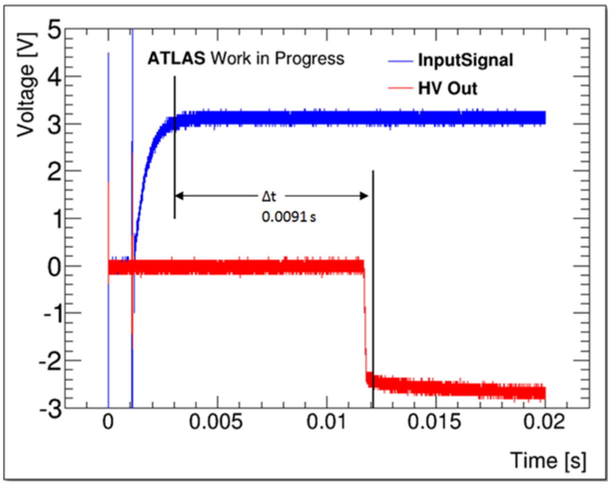

Figure 7: Time delay between the input and output signals.

6. Conclusions

After a few fixes and slight adjustments the board functioned as designed. The prototype allowed for further modifications to be made on the board before going into production. The redesign of the board was a success. The board is now being used in the latest models of the MobiDICK units and will be used in the future designs for the upgraded system in 2022.

7. Plans

There are plans to redesign the board so the output voltage may be variable. This would allow more flexibility in future designs of the system. The board will now be produced by South African industry which shall be quality tested before going into production.

References

[1] Aad G *et al.* 2012 *Physics Letters B* **716** 1–29 ISSN 03702693 URL `http://www.sciencedirect.com/science/article/pii/S037026931200857X`

[2] Solans C, Carri F, Kim H Y, Moreno P and Reed R

[3] Carrio F, Moreno P, Masike T, Reed R, Sandrock C and Kim H Y