U of S Passive Analog Splitters for use with CAEN 792AA VME QDC

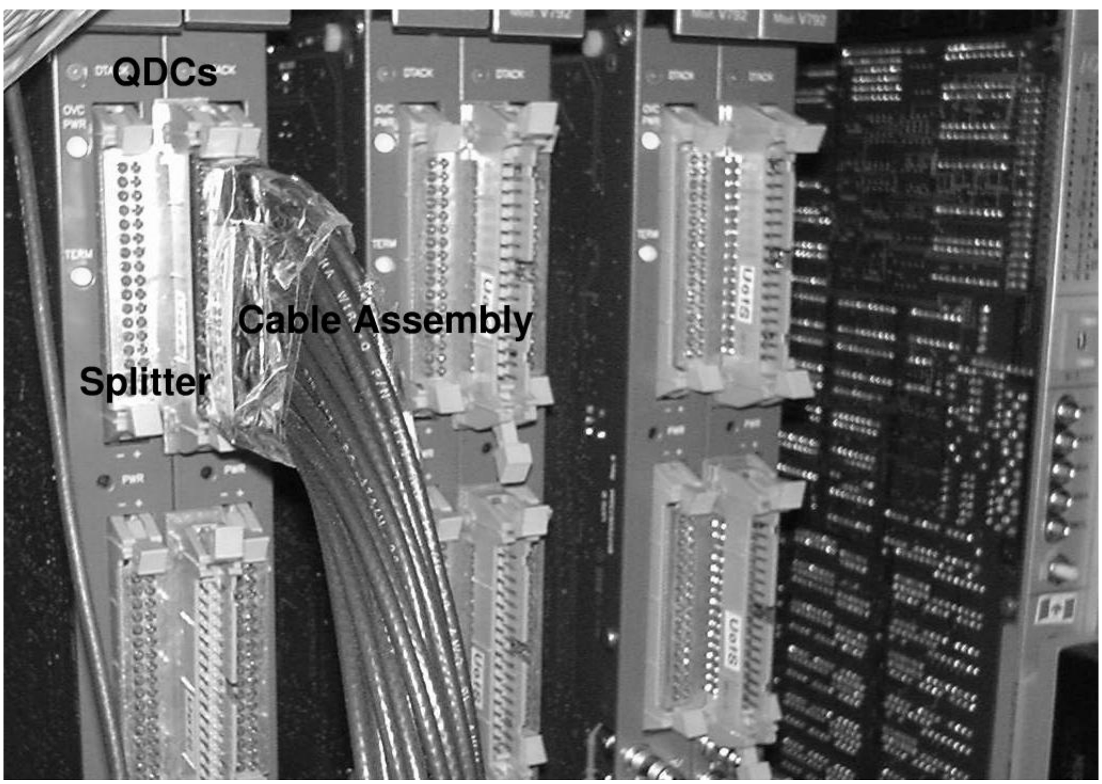
Ru Igarashi and Jennifer L. Robb

20th August 2003

Introduction

This report describes a passive analog splitter used to equally split analog signals from a cable assembly terminated by a 17-pin-pair rectangular socket connector similar to those used in ribbon cables (as described in [1]), and then pass the signals through a pair of rectangular socket connectors to a pair of CAEN V792AA QDC. As explained in the preliminary report [2] this is needed to simultaneously measure signals using differing gate lengths so as to provide pulse shape discrimination for neutrons in the Blowfish neutron detector array.

Figure 1: Splitters in Use



1 Design

As explained in the preliminary report, the splitter needed to accept 0.1"x0.1" pitch rectangular socket connectors (17 pin-pairs), split the signal with a voltage divider circuit, and pass the two signals to the QDCs via a pair of rectangular socket connectors of its own. Plan views of the assembled splitter are shown in Figure 4. The V792 uses 3M 3431 Latch/Ejector headers, and those were acquired for the splitter. Latchless headers were not used because the weight of cabling could conceivably unplug the cable from the splitter. On the other side, custom modified Samtec SSQ-117-01-S-D socket connectors (ordered as ASP-104465-01) were used. The main concerns are that the polarity of the splitter is not obvious and that it is easy to plug in the socket connector offset by one pin pair. The modification of the standard Samtec SSQ connector provided a center ridge which fits the center notch on the 3M header. This provides a polarity key and ensures the pin alignment. Samtec provides customization services for a nominal charge. All connectors are gold plated to eliminate oxidization which historically is an annoying problem responsible for loss of signals.

The board is standard FR-4 fiberglass, double plated with copper, etched with the circuit layout and tinned. The corners of the board are cut out to accommodate the latches from the QDCs. This will ensure that the weight of cabling does not disconnect the splitter from the QDCs. The end was not cut off entirely because mounting the header with screws gives additional support for the weight of the cable, off of the pins of the header.

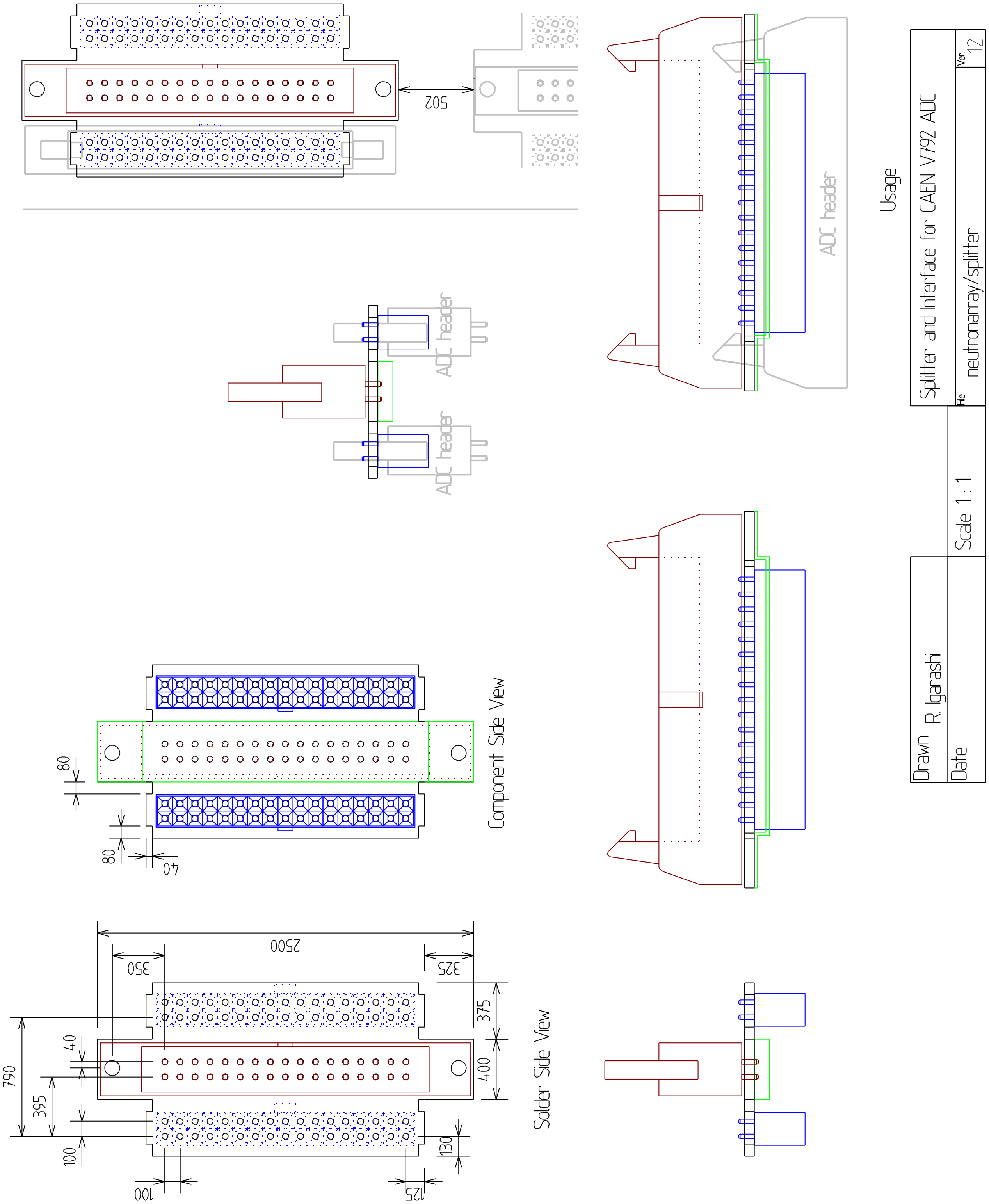
Notches were cut out to accommodate structures on the QDC headers. The notches were deep enough that it was evident that simply cutting the corner deeper would be too close to traces on the board and would reduce the available surface area for the latches to hold.

The QDC dimensions determined the placement of the connectors, so all that was needed was designing the layout of the printed circuit within the spatial constraints given. The layout of both sides of the printed circuit board is shown in Figure 3. The boards were printed by **Alberta Printed Circuits** (**APC**) using our layout. **APC** was selected for three reasons. They had an economical package for simple prototyping (30% better than a Regina-based company), they had a flexible design submission scheme, and they accepted files from many different printed circuit board programs, particularly the one we use, **PCB**. **PCB** [3] is a free UNIX program for multiple layer printed circuit board (pcb) layouts, originally written by Thomas Nau and now maintained by Harry Eaton. While it has its idiosyncrasies (e.g. many features are only available through keyboard commands rather than its many menus), it is highly usable. One minor issue is that **PCB** works with finished via diameters (finished vias are plated, which narrows the hole), and **APC** requires raw via dimensions. However, **APC** supports the use of **PCB** and supplies scripts that correct **PCB**-generated files for production.

The vias are plated and ground vias are connected by thermals to provide a common ground plane on both sides. The surface mount (SMT) resistors are mounted horizontally across the gaps. Not shown are the holes for mounting the latch headers; a bug in the layout software results in a failure to display simple holes (the actual manufacturing files include them).

The 0.1" pitch of the connectors and the thickness of the pads meant that the traces could not be much more than 20 mil wide (10 mil = 10/1000" = 0.254 mm). Traces should be narrower to reduce crosstalk, but narrower traces exacerbates a reduced transmission line impedance and increases the risk of reflection. The traces could not be much wider because they have to be threaded between the pads of the connectors. Ground striping (running ground traces between

Figure 2: Plan and conceptual view of assembled splitter (shown at scale)



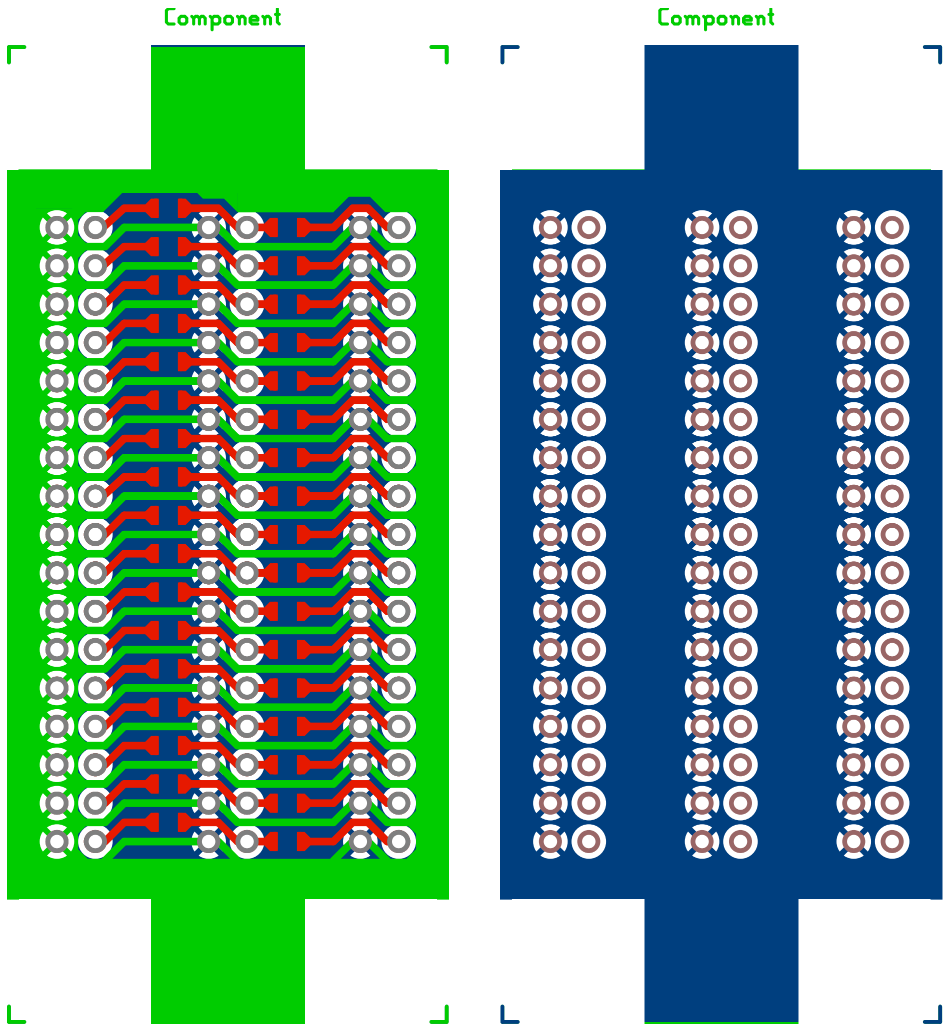
adjacent signal traces) reduces crosstalk.

All via pads were 12.5 mil thick. Pads for the SMT were 39.5 x 39.5 mil and are considered as small as pads should be. Both types of pads should perhaps have been elongated by 10mil on either side.

Vias for the connectors were originally designed with 35 mil finished diameter. However, **APC** required unfinished diameters and with the necessary estimated correction these increased to closer to 38 mil. Holes for the latch headers were 120 mil in diameter, though they could have been 110 or even 100 mil (the spec sheets suggested 106).

50Ω CRCW series 0805 package SMT resistors from Vishay were used. These are 1/10 Watt, 1% tolerance resistors, are 2mm x 1.25mm x 0.45mm (l x w x h), and have a good temperature coefficient (100 ppm/K).

Figure 3: Layout of printed circuit for splitter (2X scale)



2 Assembly Instructions

The circuit boards were printed as shown in Figure3. The layout was drawn with the free Linux program **PCB**. As discussed in [2], **Alberta Printed Circuits** offered a low cost and accepted layout files written by **PCB** , including supplying special compatibility scripts. This particular project was submitted by ftp along with an e-mail notice. Within 3 days, we had the printed boards (credit card payment is recommended, or arrange for a PO before submitting the design).

The price package used with **APC** was simply for etching the layout, drilling holes, and tinning. It did not include cutting anything more intricate than rectangular outlines. Thus, once we received the circuit boards, cutting out the corners and the notches was our responsibility.

There are 3 phases for constructing the splitters: 1) cutting and preparing the circuit boards, 2) soldering the surface mount resistors, 3) mounting and soldering the headers.

2.1 Cutting and Preparing the Circuit Boards

The printed circuit boards used to make the splitters come from Alberta Printed Circuits. They are originally a rectangle of FR-4 fiberglass board with the pre-determined circuit trace double-plated with copper and tinned. There are three main steps to preparing these circuit boards, cutting the corners, filing the edges and filing the notches.

2.1.1 Cutting the Corners

The pre-printed boards come as a 1150 x 2500 mil rectangle, and are desired to be a 1150 x 1850 mil rectangle with two 400 x 325 mil tabs on the top and bottom (along the 1150 mil sides), see Figure 1. Thus, the four corners of the original rectangle need to be cut out.

The following tools and parts are needed:

- circuit board
- corner-shear press

To cut out a corner:

1. The board should be positioned under the corner-cutter's right angle blade, the operator visually determining where the cuts will be made.
2. Leave about 0.5 mm of uncovered fiberglass visible around the trace.
3. Pull down the lever of the machine, allowing the blade to slice through the fiberglass and remove the corner piece.
4. Repeat for the other corners of the board. (4 cuts in total)

2.1.2 Filing the Edges and Notches

The following tools and parts are required:

- circuit boards with corners cut (see previous stage)
- sandpaper
- triangle file
- flat bench with right angle edge

The cut edges of the boards (both those cut by the manufacturer and those from the cut corners) must be filed down as follows:

1. Line up the piece of sandpaper across the edge of the table or block.
2. Position the edge of the board that is to be sanded against the sandpaper, with any of the board that is excess hanging over the table or block.
3. Run the board over the sandpaper repeatedly until there is no more unprinted fiberglass board visible on that edge (right up to the trace).

For the board to fit into the latched headers where it will be positioned, there needs to be eight notches filed into the board. These notches allow the board to fit into the header, but to remain held in place by the latches. The notches need to be filed into the 1150 mil sides of the trace (See Figure 1). Along these edges there is a 375 mil area, a 400 mil (400 x 325 mil) tab and another 375 mil area. They will be created as follows: The first set of notches is placed directly on either side of the tabs, and they need to be approximately 2 mm long and 1 mm deep (80 x 40 mil) into the copper trace.

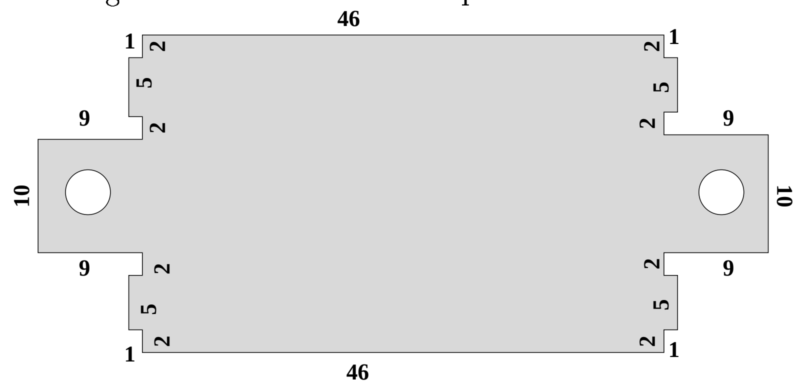
1. Position the triangle file in the area where the notch should be.
2. Run the file over the area until a dip appears in the trace.
3. Note that the notches will most likely be the shape of a rounded triangle, as opposed to a perfect rectangle.
4. Four of these notches should appear on each board, as there is a tab on the top and bottom edges of the trace, and each tab should have a notch on either side.

The second set of notches are filed from the edges of these same sides. The 2 x 1 mm (80 x 40 mil) notches will be directly on the edges of the board (which ends up cutting out the corners again). To make these notches:

1. Position the triangle file approximately 2 mm from the board's edge.
2. Run the file over the area until a dip appears in the trace.
3. Use either the file or the sandpaper to file down the side of the dip closest to the board's edge, creating a rectangular shaped notch.

4. Note that because these notches are on the edge of the board it is often difficult to maintain a 2mm width. Often a 3mm notch will occur unintentionally due to the technique, but this is acceptable as there is still enough space for the latches of the header to make contact with the board.

Figure 4: Dimensions of Prepared Boards in mm



2.2 Soldering the Surface Mount Resistors

The following tools are needed:

- soldering iron with a 1.2 mm wide chisel tip
- 0.020” or 0.015” solder, preferably with water soluble or "no clean" flux
- fine braid solder wick (for removal of solder if mistakes are made)
- very fine tipped stainless steel tweezers
- illuminated magnifying lens, or alternatively overhead desk lamp and large magnifying glass
- 0805 package 50Ω resistors (1% tolerance), total of 34 for each board
- splitter circuit boards

Each printed circuit board contains 34 spaces for the attachment of surface mount resistors, to complete the voltage divider circuit. The SMT resistors are soldered onto the boards as follows.

1. "Tin" 1 pad as follows:
   (a) Lay the tip of the soldering iron on the pad.
   (b) Make contact with the solder, touching the trace and the soldering iron, and a small layer of solder should melt onto the trace.
2. Set the resistor in place on the board using the fine point tweezers.

3. "Tack" the resistor in place by holding the soldering iron so that it is touching both the solder and the resistor's edge momentarily. Recommended technique for this procedure is to lay the flat part of the chisel tip in the crease between the resistor and the solder. This should attach one side of the resistor to the board.

4. Hold the solder against the other end of the resistor and apply the soldering iron to melt the solder onto the resistor and the board. (This should be in position on the other resistor pad.) Again make sure that the soldering iron is touching both the solder and the resistor's edge.

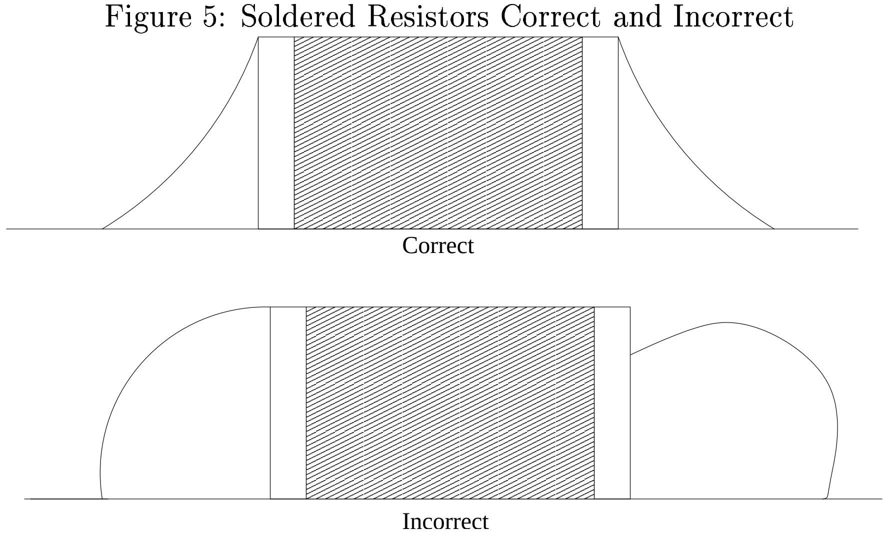
5. If the side that was attached to the board first looks like it needs a more substantial connection, add some solder to that area, using the technique in steps 3 and 4 above.

6. If mistakes in soldering are made and solder needs to be removed from the board, it can be done as follows:

   - If the solder to be removed is in contact with a resistor, remove the resistor by heating both connections with the soldering iron, and lifting it up with the fine point tweezers.
   - Lay the fine braid solder wick over the unwanted solder.
   - Press on the wick with the soldering iron in the area where the solder resides. Note that the pressure used here is important, as too much pressure may remove the copper trace from the board, but to little pressure will fail to melt the unwanted solder onto the wick.
   - Once the solder melts (this will be apparent by the small drop felt by the soldering iron) gently run the wick back and forth over the area to be cleaned. This should remove the unwanted solder.

7. Inspect the connections under the lighted magnifier to ensure proper connections. See Figure 5 for what proper application of solder should look like.

Figure 5: Soldered Resistors Correct and Incorrect



8. The digital multimeter can be used to test the resistance (should be approximately 50 ohms), and to test that no improper connections have been made on the board (continuity tests).

2.3 Mounting and Soldering the Headers and Socket Connectors

The following tools are needed:

- soldering iron with a 1.2 mm wide chisel tip
- narrow gage solder (doesn't have to be as narrow as 0.020")
- fine braid solder wick (for removal of solder if mistakes are made)
- circuit board with resistors **already** mounted
- 2 rectangular socket connectors
- 1 latched header
- 2 1/2 inch hex head screws

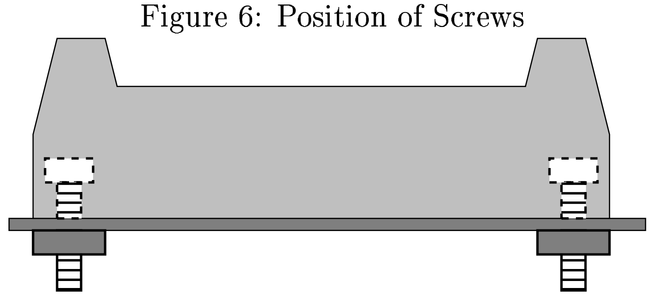
Each printed circuit board contains vias for a latched header and two rectangular socket connectors. The latched header is attached on the side *opposite* to the printed circuit. The socket connectors are attached on the *same* side as the printed circuit (and the mounted resistors). These components are attached as follows:

2.3.1 Mounting the Latched Header

1. Make sure that the header is placed on the board with ground in the right direction. The notch representing the ground side should match the vias that have thermals, and the signal side of the header should match with the vias that are isolated (on the side visible while mounting the header).

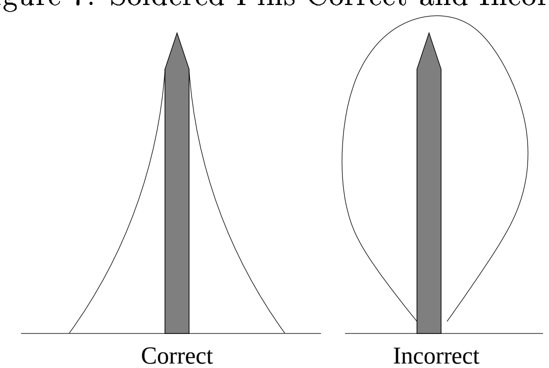
2. Attach the header to the circuit board with the screws, in the direction such that the bolt resting on the board. These screws hold the circuit in place while the soldering process is carried out, and they provide extra strength to the system once it is completely assembled. See Figure 6.

Figure 6: Position of Screws



3. Solder each pin as follows:
   (a) Touch the pin and the trace with the solder.
   (b) Apply the soldering iron to melt the solder onto the pin and the trace. (Make sure that the iron is touching the solder, the pin, and the trace).
   (c) Holding the soldering iron in place, feed the solder into the area, allowing it to melt around the pin. (The solder should flow around the pin and into the connecting hole). Note that it is normal to feed a lot of solder into the area, since the hole around the pin must be filled.
   (d) If there is an uneven amount of solder on the pin (the pin should have solder all the way around) more solder can be added.
4. Check each pin to see that the solder is making contact with the board. It is easy for a "ball" of solder to form on the pin without making full contact with the trace itself, and this must be prevented to ensure a proper connection. See Figure 7 for what proper application of solder should look like.

Figure 7: Soldered Pins Correct and Incorrect



2.3.2 Mounting the Socket Connectors

1. Make sure that the socket connectors are placed on the board with ground in the correct direction. The notch representing the ground side on each connector should be facing the same way as the ground notch on the already attached latched header.
2. Place the board and connectors on a table or other flat surface so that the board rests on top of the pins.
3. Solder each pin as follows:
   (a) Touch the pin and the trace with the solder.
   (b) Apply the soldering iron to melt the solder onto the pin and the trace. (Make sure that the iron is touching the solder, the pin, and the trace).
   (c) Holding the soldering iron in place, feed the solder into the area, allowing it to melt around the pin. (The solder should flow around the pin and into the connecting hole).

(d) For ground pins (those with thermals), note that the solder may flow onto the thermals or out onto the board. This overflow can be minimized by taking care to lift the soldering iron in a directly vertical motion when removing it from the board, as any movement over the surface of the board can facilitate the flow of solder away from the pin. It is not detrimental to the circuit for extra solder to appear on the board in areas where there is already copper present but it is **NOT** OK for the solder to bridge a connection between the board and the signal pins. Care should be taken to limit the amount of solder that flows out of the pin's immediate area.

(e) If solder needs to be removed from the board, it can be done as follows:

- Lay the fine braid solder wick over the unwanted solder.
- Press on the wick with the soldering iron in the area where the solder resides. Note that the pressure used here is important, as too much pressure may remove the copper trace from the board, but to little pressure will fail to melt the unwanted solder onto the wick.
- Once the solder melts (this will be apparent by the small drop felt by the soldering iron) gently run the wick back and forth over the area to be cleaned. This should remove the unwanted solder.

(f) If there is an uneven amount of solder on the pin (the pin should have solder all the way around) more solder can be added.

4. Check each pin to see that the solder is making contact with the board. It is easy for a "ball" of solder to form on the pin without making full contact with the trace itself, and this must be prevented to ensure a proper connection. See Figure 6 for what proper application of solder should look like.

2.4 Finished Splitters

The following figures (Figure 8 and Figure 9) show what completed splitters will look like.

Figure 8: Finished Splitter

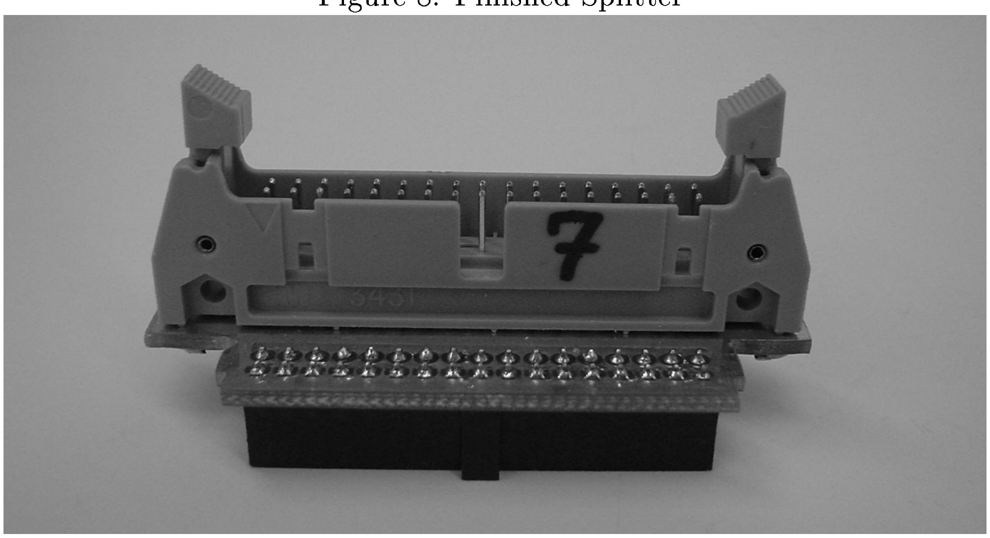
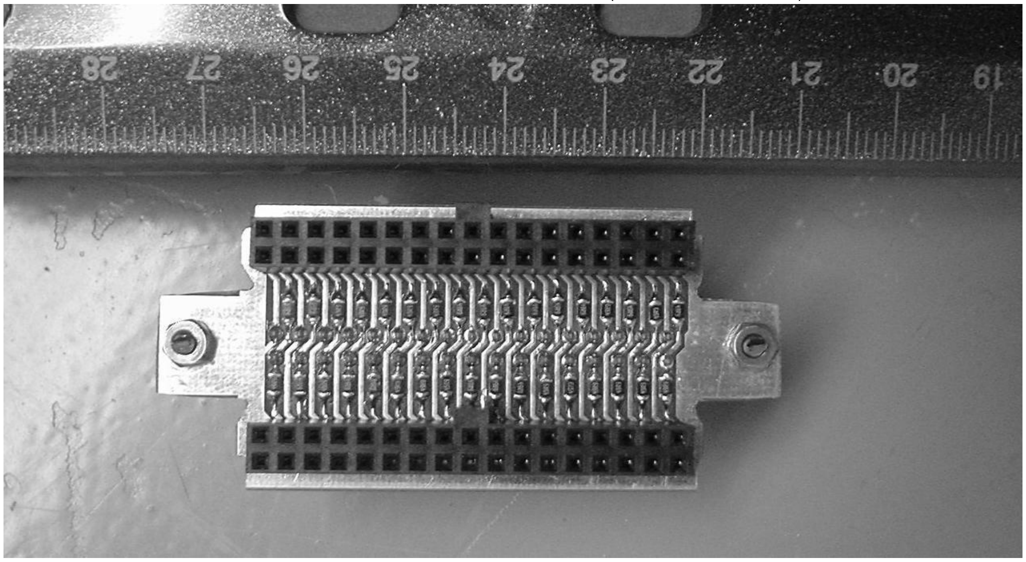


Figure 9: Finished Splitter (Bottom View)



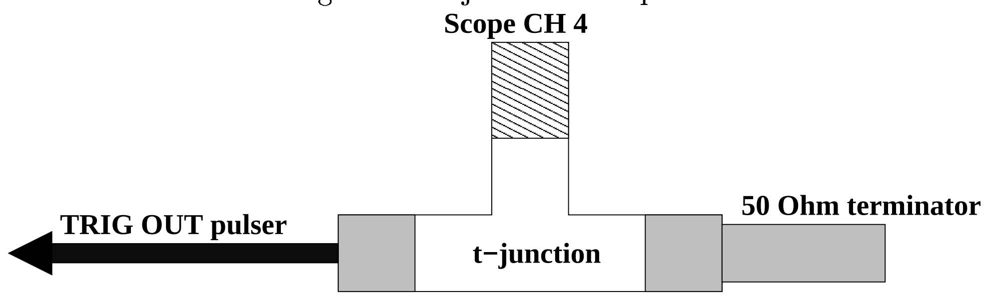
3 Testing Instructions

The splitters should be tested thoroughly before use. The testing described below is to prepare the splitters for use with RG174 Cables for use in CAEN 792AA QDC, the cables described in *U of S Cable Assembly for CAEN 792AA QDC* by Ru Igarashi and Ward Wurtz. For recommended testing the following parts and tools are needed:

- Tektronix 2465 400 MHz Oscilloscope (need 4 channels)
- BNC Model 8010 Pulse Generator
- A tested RG174 Cable for CAEN 792AA QDC

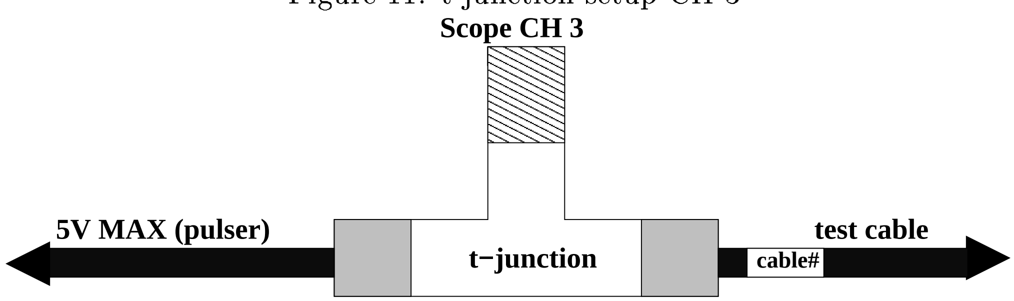
1. The following settings are needed on the pulse generator:
   - Frequency: 1MHz (gives visible line on scope)
   - Delay: 0.03 μs (minimal delay)
   - NORM/COMPL to NORM
   - Amplitude: About 1/2 max (knob turned half way)
   - Width: 0.1 μs
2. Plug the TRIG OUT of the pulse generator into one channel of the scope (eg. channel 4) using a connecting cable (eg. RG174 coaxial cable).
3. Ensure that there is no reflection due to an impedance mismatch. With the described operation there should be a 50Ω cable plugged into a 1MΩ port. This impedance mismatch can be dealt with by using a t-junction at the oscilloscope port, one end being plugged into a 50Ω terminator, & the other being used as the output channel (containing TRIG OUT of the pulse generator). See Figure 10.

Figure 10: t-junction setup CH 4



4. Another t-junction should be put the next channel of the scope (eg. channel 3). One end of the junction should be connected to the 5V MAX output of the pulse generator by a cable, and the other should be connected to one of the numbered BNC ends of the tested RG174 cable for CAEN 792AA QDC (should start with 0). See Figure 11.

Figure 11: t-junction setup CH 3



5. The other two ports of the oscilloscope should be connected to a specialized RG174 coaxial cable with one BNC end and one two-pin end. These will have already been made for proper testing of the RG174 cable for CAEN 792AA QDC, see the cable's documentation for more details.

6. The following settings are needed on the oscilloscope:
   - Trigger slope: +
   - Trigger source: the channel that is connected to the TRIG OUT of the pulser (eg. channel 4)
   - Delay: Can be adjusted to see pulse.

7. Viewing the channels connected to the pulse generator on the oscilloscope, adjust the amplitude on the pulser and the delay and scales on the slope to obtain a square wave with maximum voltage of about 4 or 5 volts.

8. Attach the RG174 cable for CAEN 792AA QDC to the splitter by putting the cable's socket connector into the header of the splitter.

9. Note that with the notch representing the ground side of the headers on the right the rows in the socket connectors on the splitter correspond to the BNC ends of the cable from 0 to 15, top to bottom. (eg. The top row corresponds to 0).

10. Looking at the number of the BNC end from the test cable plugged into the t-junction on the oscilloscope, connect the two ports with pin-end cables into the corresponding rows on the socket connectors of the splitter.

11. Viewing the channels of these cables on the oscilloscope, two identical pulses should be visible, one for each channel, and they should each have approximately half the amplitude of the original pulse. The scale that will most likely view these pulses the best is 500mV.

12. Move the pins into the next row on the socket connectors. There should be no visible pulse on the oscilloscope at the current scale (500mV), but when the scale is changed to approximately 20mV, a signal will be visible. This is cross-talk from the attached cable, and is acceptable in minimal amounts.

    - NOTE: Some of this visible signal is due to reflection. The reflection occurs at the BNC end of the cable that corresponds to the pins being tested for cross-talk, and at the socket connector plugs corresponding to the BNC cable hooked up to the oscilloscope which now have nothing in them. This reflection can be eliminated by adding 50Ω terminators to these areas. (A 50Ω terminator on the BNC end of the cable corresponding to the pins being tested for cross-talk and 50Ω resistors between the holes in the socket connector corresponding to the BNC end that is attached to the oscilloscope).

13. The cross-talk should be tested around each connection to ensure uniformity (irregular cross-talk could indicate problems with the splitter).

14. Each port on the splitter should be tested by repeating steps 10-13 for each numbered BNC end of the specialized RG174 cable and its corresponding rows on the socket connectors.

References

[1] "U of S RG174 Cable Assembly for CAEN 792AA QDC", Ru Igarashi and Ward Wurtz, Internal Report, Summer, 2003.

[2] "Design Considerations for Cable Adapters for CAEN V792AA QDC in the Blowfish Array", Ru Igarashi, Internal Report, 19 February, 2003.

[3] **PCB** home page is at: http://bach.ece.jhu.edu/~haceaton/pcb/