Video Lighting Interface Card

Barbara Johnson, Oklahoma State University
Mentors: Lance McBride
Summer 2004

Keywords: camera gating, Orcad, PIC Programming, PCB layout

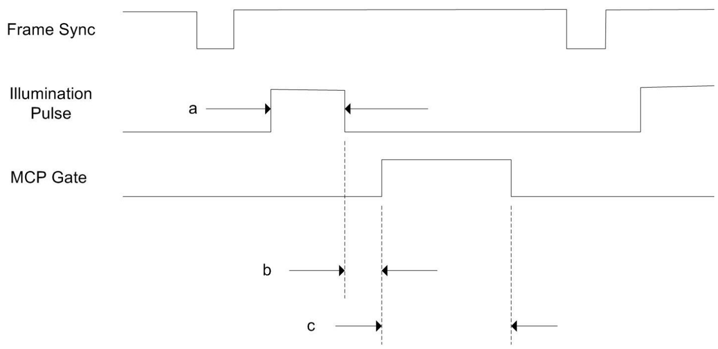
ABSTRACT

Electrical design has many stages. The process of designing a printed circuit board is covered through the design of a video lighting interface card. The card takes information from the user through hyperterminal. A microcontroller takes the information and translates it into information the camera can understand. The microcontroller controls the zoom, focus, and iris motors on a G10X16MEA lens as well as lighting. Accommodations are made for expansion and revision throughout the design process. This is necessary for troubleshooting and later modifications since the design is for an idea that is still in the experimental stages.

INTRODUCTION

Monterey Bay Aquarium Research Institute (MBARI) employees have been working on a low light imaging module. This module can be put on either an autonomous underwater vehicle (AUV) or a remotely operated vehicle (ROV). Using both gated and non-gated lighting, the module can be used to count both non-bioluminescent and bioluminescent creatures.

The gated lighting system involves a pulsed light which is carefully synchronized with the camera’s electronic shutter, or the micro-channel plate (MCP) gate. This allows only objects at a specified distance to be visible. The lighting can pulse with a variable pulse width and intensity to control it. This pulse creates a wavefront. The wavefront travels to the object to be viewed, and light bounces off the object to the camera lens. When the light reaches the camera, the camera is gated to view the object. Until the camera is gated, no image is produced. This is what allows only objects at the specified distance to be captured. The camera and lighting must be carefully synchronized, but the result is a clear image of objects at the specific distance.



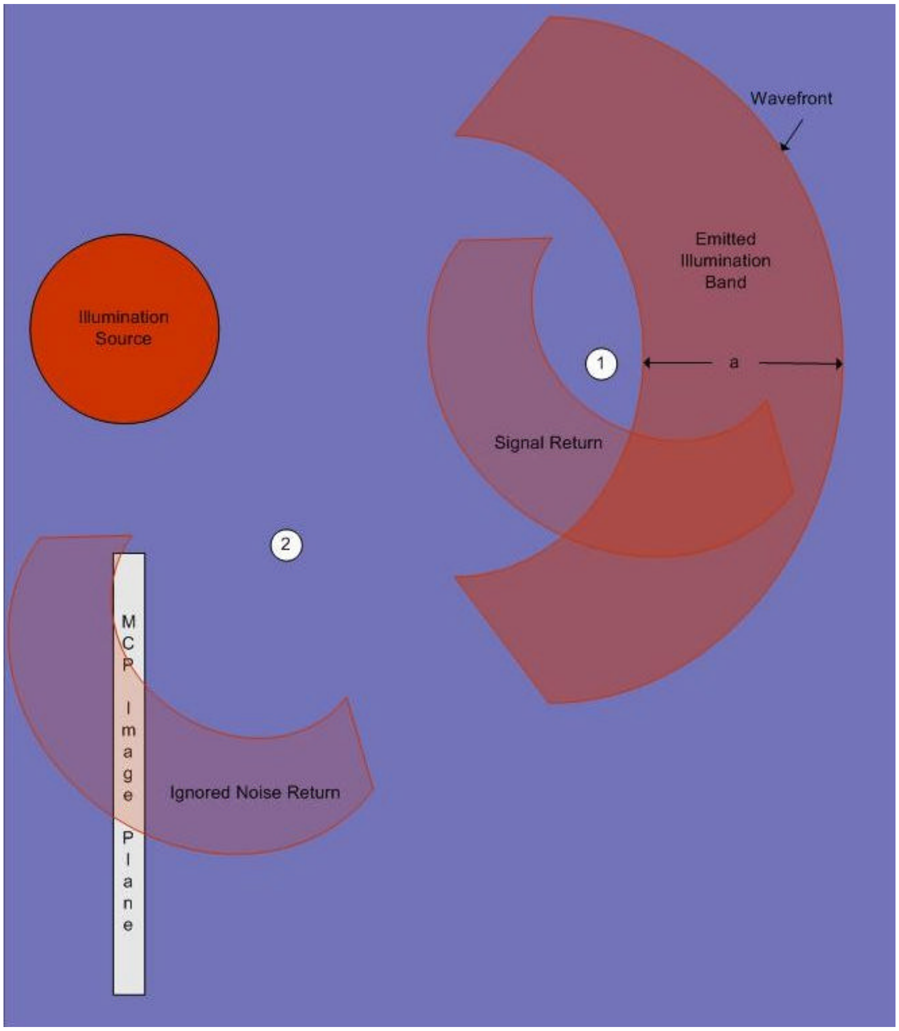
Descriptions:

a. Width of emitted illumination band

b. Distance from illumination/camera plane to first signal return plane

c. Camera integration period. Distance illumination band is observed.

This is a timing diagram drawn by Lance McBride to explain how the lighting pulses must be synchronized to both the video signal and the camera's gate.



This diagram drawn by Lance McBride shows how the wavefront behaves.

While learning to work with the new module, scientists need to control the different camera settings, lens controls, and lighting details. This paper details the methods used to design a card which allows scientists at the surface to control these settings through a serial interface. Later, when scientists know which controls are most relevant and most often used, a more user friendly interface, like a graphical user interface (GUI), may be developed.

MATERIALS AND METHODS

PIC PROGRAMMING

The dsPIC30F6014 (PIC) is used as a microprocessor to interpret user commands and control outputs. MPLAB IDE v6.5 allows programming in assembly, C, or a combination of the two programming languages. Each microprocessor has its own assembly instruction set. The dsPIC30F6014 has a new, larger instruction set. A basic understanding of the microprocessor's commands and syntax is needed before any complex programming can be accomplished.

Unfortunately, this particular PIC uses a slightly different syntax than the code samples available in the compiler's help files. In learning how to program for this PIC, many hours were spent on the phone with the Microchip's technical support staff. Eventually, Microchip sent sample code as well as corrections to code.

To conquer this new language, a total of eight programs were written. The first program involved trying to port a beginning program that Lance McBride wrote in order to become familiar with a different PIC. This attempt failed when it was determined that the instruction sets of the two microprocessors are too different. The program flashed a light emitting diode (LED). In lieu of porting Mr. McBride's program, a new one was created from scratch. Pins were already routed to control four LEDs on the demo board, so the program created flashed one of them. At this point, much had been learned about the general structure each program needs to work with this particular microprocessor as well as how to use an in-circuit debugger and communicate with technical support.

All the subsequent programs written flashed LEDs in different patterns. This became a way to check if the program was doing what was expected without the use of the in-circuit debugger. A debugger is very useful, but going through a program line by line is sometimes not practical. These programs were instrumental for learning how to use interrupts and to communicate using a universal asynchronous receiver-transmitter (UART) connection to the computer's hyperterminal as well as for learning how to use a computer's hyperterminal. At this point, attention was turned to drawing the actual schematic.

SCHEMATIC DESIGN

To draw the schematic, Orcad Capture v9.2.5 was used. Mr. McBride showed how to create new parts in Orcad based on their datasheets. Finding appropriate parts also proved to be challenging. It is a valuable skill, however, and became easier with practice. One by one, the basic components were pieced together with a basic understanding of the final design. While working on a design, however, it is subject to change. Even the most minor changes must be taken into account with a view of the entire schematic.

After a basic preliminary schematic was drawn up, a design review was held. The purpose of a design review is to get input from other engineers. Their questions and comments can help a designer see the design in a different way and make sure nothing is forgotten. This design review lead to the inclusion of a water sensor, a humidity and temperature sensor, and a pressure sensor. While most PCB boards don't require such sensors, this board will end up in a housing under several meters of water. The sensors allow the conditions in that housing to be monitored and a failure, such as a leak, to be reported to the user.

As the more parts were added to the schematic, the A size (8.5" by 11") paper became too crowded. Orcad allows the size of the paper to be increased, going up to B, C, D, and even E size sheets. Each letter increase represents a doubling in paper size; the dimension of the shorter side is doubled so that E size paper is 34" by 44". This is size of paper that the complete schematic requires. Each time the paper size is increased, the parts in the schematic must be spaced out so that new parts may be added between them if necessary. This can be a lengthy process because when parts and wires are moved in Orcad, sometimes wires are crossed and unwanted connections are made.

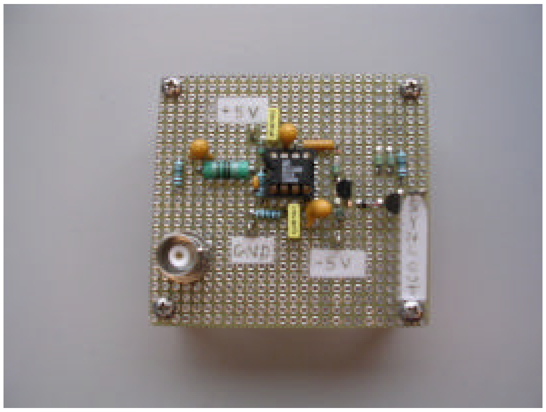
Orcad can also perform a design rules check (DRC) which should be passed before the schematic is put into Orcad Layout. The DRC tells you if there are unconnected nets or two parts with the same designation, etc. Each of these can be fatal to the success of the design and must be fixed.

A bill of materials (BOM) can also be generated by Orcad. The BOM lists all the parts in the design and allows the designer to make sure that he has a part number for an actual part that corresponds to each item in the BOM. Such a list also allows the designer to make a budget and order the parts in an organized manner.

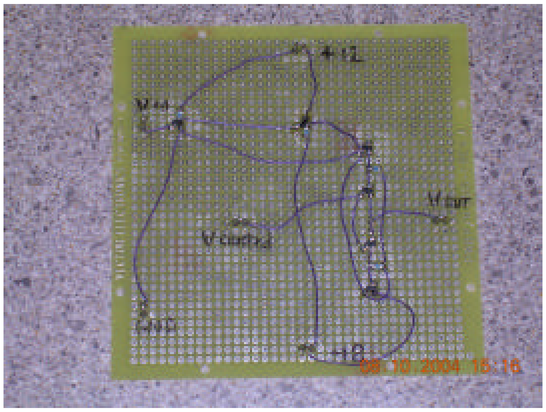
PROTOTYPING AND SIMULATION

In order to ensure that new ideas will work, prototyping is necessary. For this project, new circuits were designed to drive the lens motors and to control the lighting. A sync stripper circuit used to synchronize the lighting to the video signal was also prototyped. These circuits were designed with specific parts in mind. Parts were then ordered and the different circuits assembled.

The sync stripper takes a standard RS-170 black and white video signal as an input and outputs just the horizontal sync and vertical sync digital information. Once the analog video signal is stripped off, the digital sync signals can be used to synchronize the light pulses. This prototype worked correctly, but the ISG-880 camera used outputs both vertical and horizontal sync signal information separately from the video signal. Although this circuit works, it never made it to the final schematic since the circuit was no longer needed.

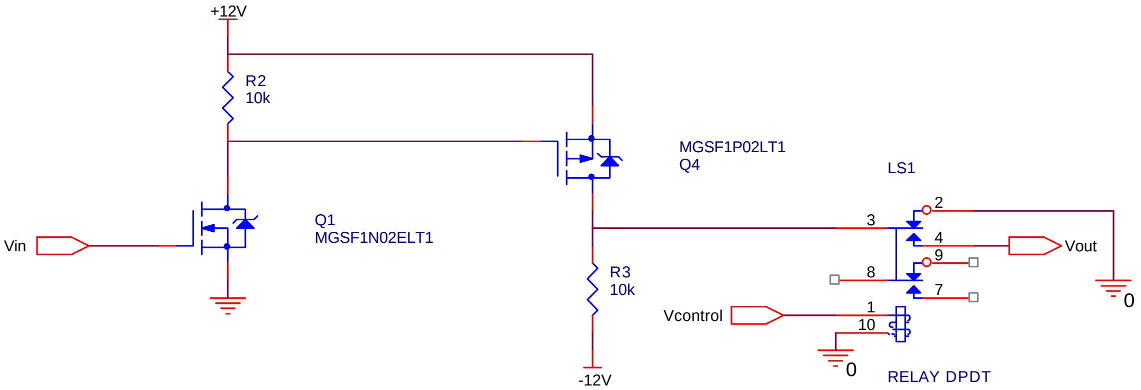


This prototype was built by Tom Marion at the request of Lance McBride.



The next prototype was built by Tom Marion.

The next prototype for this design was an attempt to find a solid state solution to the problem of switching between +12V, -12V, and 0V. This could also be done with a relay, but relays, being electromechanical, tend to wear out after a few years whereas solid state devices will work indefinitely. This circuit was built with parts on hand at MBARI that closely matched characteristics of the specific parts in mind. When this happened, it was discovered that the characteristics did not meet minimum requirements for the circuit to work. New parts were ordered with higher tolerances, but the design still failed. After the design was more closely investigated, relays were chosen. Even with relays, some metal-oxide semiconductor field-effect transistors (MOSFETs), are required. The parts in schematic shown below were only tolerant to 20V. Since one of the transistors sometimes requires a difference from +12V to -12V for a total of 24V, the MGSF1PO2LT1 was not a viable option. A transistor which is 30V tolerant should work, however.

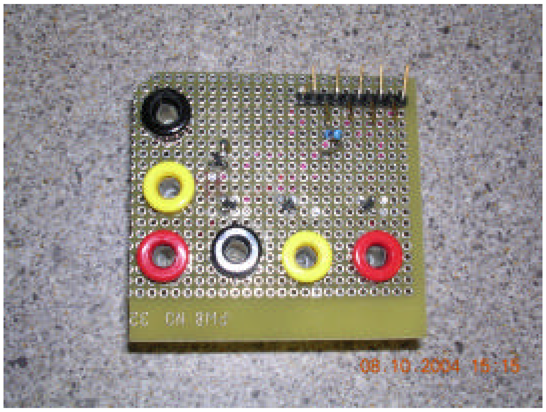


This is the basic schematic for the design used, complete with relay.

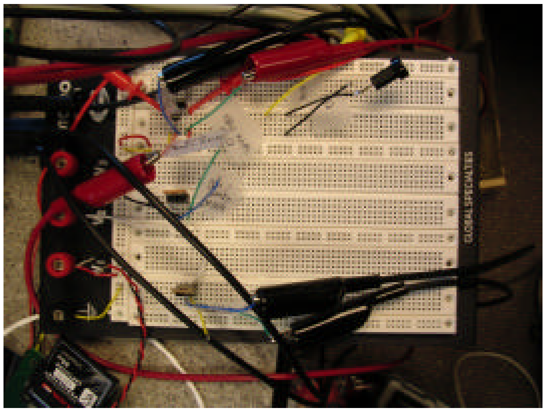
Since parts take time to order, the schematic was simulated in Orcad. Parts in Orcad require many cryptic parameters to be defined. The simulation part for the MGSF1P02LT1 was found at ON Semiconductor’s website. The design failed with this simulation. The parameter involving the 20V breakdown voltage was changed to represent a breakdown voltage of 30V. With the new parameter, the simulation worked correctly, so new 30V parts were ordered for further prototyping.

Simulation can be an effective step in design. Much like prototyping, simulation can provide further information on why a circuit will or will not work. Each part in the simulation must have its parameters accurately defined. Since the part parameters are so cryptic, however, this is not always a viable practice. With prototyping, the actual components are used and therefore provide a more accurate representation of whether or not a circuit will work correctly.

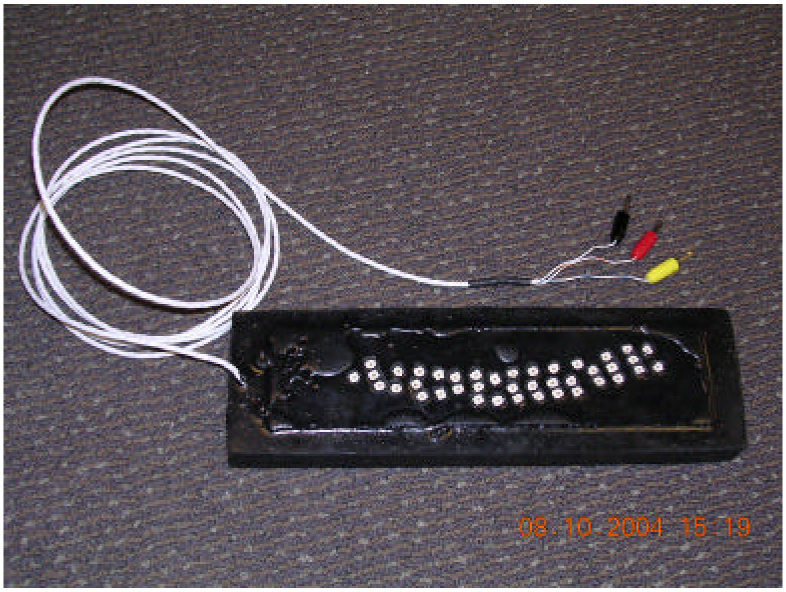
Two more circuits were prototyped, both for the lighting system. The lighting currently available are LEDs, both individual and in a panel. The panel of LEDs draws up to about 6.4A of current, which is much more than any of the MOSFETs in the second prototype could handle. The next prototype consisted of MOSFETs with tolerances much above those needed, partly to ensure their success, but also because MBARI had these parts available. When possible, it is much faster to use parts immediately available instead of ordering parts for prototyping. This third prototype also did not work. Upon closer inspection of the datasheet, half the transistors still had current tolerances that are too low. New parts were found and ordered.


08.10.2004 15:15

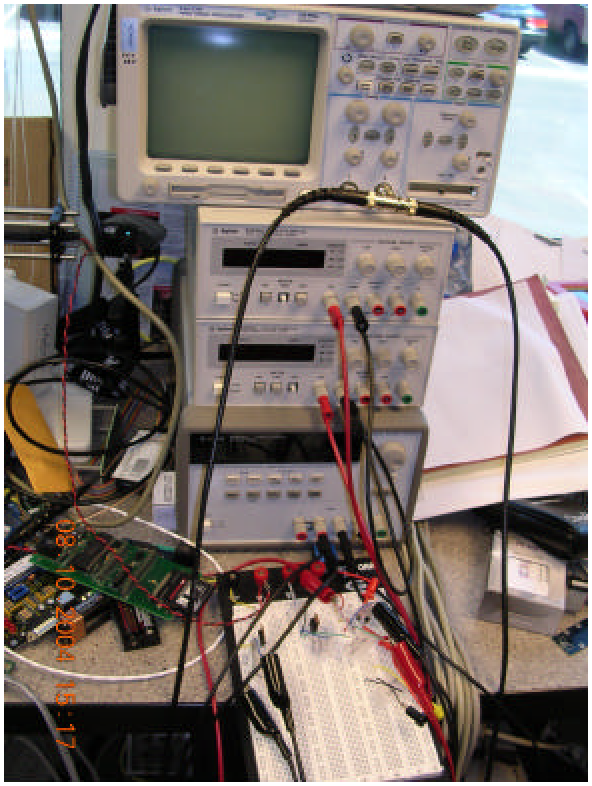
The third prototype was built by Jose Rosal.
The small transistors cannot handle much current.



The next prototype was done on a protoboard.


08.10.2004 15:19

This is a photograph of the LED panel used. It is connected to the system in the next picture.


08 10 2004 15:17

This is a view of the protoboard and its connections to an oscilloscope and power supplies.

Prototyping on a protoboard is much easier when working with through-hole components since it is easy to move parts and connections around. When working with surface mount parts, however, the parts are too small to place into a protoboard. Connecting wires on a prototype can sometimes be the cause of problems, so each connection must be carefully checked before a design idea is discarded. The last few prototypes were done on a protoboard. None of them have worked so far, but more parts have been ordered along with a design change which should fix the problem.

PCB LAYOUT

Laying out a PCB design requires many careful considerations. Boards can have different numbers of layers, which can be either routing layers or plane layers. The connections used to carry signals on the same layer of a PCB board are called traces. Signals carrying high currents need wider traces. Fast signals on traces can act as transmission lines, causing interference in other lines and should therefore have short traces. A digital signal can cause significant noise in a neighboring analog signal. Everything must be connected the way the schematic is wired, but the parts and wires cannot necessarily be laid out in the same way. Space can also become an issue.

Bypass capacitors are used to provide power close to a part and to reduce noise. Capacitors store energy, so if they are located close to a part that needs power, there can be a short trace leading straight to the part to power from the capacitor even if the power supply is inches away. Bypass capacitors also act as low pass filters, so the high frequency associated with the switching of a digital signal is filtered out.

Power planes and ground planes are often used instead of traces. The whole board requires a common ground, so it is convenient to be able to merely place a via anywhere on the board and have access to ground instead of routing traces. A via is a hole in the PCB lined with conductive material. Connections can be made to a via on any layer of a PCB board and is a good way for a signal to be routed from one layer to another.

A more extensive prototype than those listed above was needed to prove that the interface between the FIFO and the PIC works correctly. The FIFO is a BGA (ball grid array) part, making it nearly impossible to solder leads to each pin. Therefore, a PCB prototype was also sent out involving just the basic FIFO and outputs to the timing for the light source.

RESULTS

The schematic design is complete, but has not been fully tested. In order for the complete schematic to be tested, the PCB board must come back and parts placed appropriately. The PIC must be programmed and the board will be tested in use. The PCB board is designed, but the actual board is not back yet. The assembly programming language specific to the dsPIC30F6014 was learned to some degree, but the programming for the video lighting interface card is not done.

DISCUSSION

The original plan for the design was ambitious for the time allotted. The internship was a great success in its endeavor to educate. An end product was produced, but all goals were not reached. The microcontroller still needs programming, but at least the programming language was learned. The idea of synchronizing timing pulses and the camera gating to filter out unwanted material is an interesting one and worth completing.

CONCLUSIONS/RECOMMENDATIONS

Engineering projects require a concept of design, but one that is flexible and open to change. Software tools, such as Orcad and MPLAB, are vital to modern electrical design. Having these tools is not enough, though. Time must be spent to learn how to effectively use these powerful tools. Prototyping with actual parts is another important component.

ACKNOWLEDGEMENTS

I would like to thank Lance McBride, for helping me through my first design project and holding class in his office for concepts that I have not yet learned in school. He also set up the project, made materials and people available to me, and allowed me to share his office. He will also be finishing up the last details of the project, including the actual programming of the PIC and the integration of the video lighting interface card.

I would also like to thank George Matsumoto for setting up the internship program and enriching the intern experience.

Thank you to Kim Reisenbichler for helping me to gain an understanding of the project from a scientist's point of view.

Thank you to Jose Rosal for helping me with my PCB layout and for putting it into Orcad Layout. He also built some of my prototypes and helped me find parts to use in them.

Thank you to Dick Littlefield for making footprints for my parts.

Thank you to Tom Marion for building my first prototype.

Ed Mellinger helped me to understand power supplies and provided me with a digital camera to take pictures for this paper and my presentation.