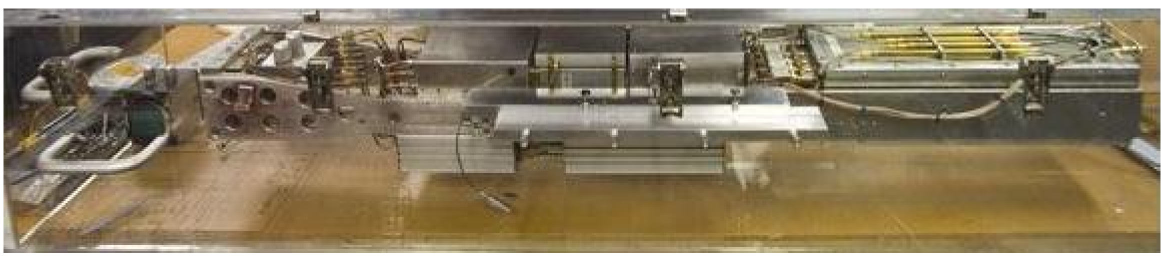
X-Ray Framing Camera Characterization Automation

2005 Summer Research Program

By: Nicholas Ramos
Advisor: Shawn Mott
2nd Advisor: D. Lonobile

The 60-Beam OMEGA laser system employs a number of x-ray framing cameras (fig. 1). The function of these cameras is to generate digital images of the target implosion using a backlight which silhouettes the target in x-rays. The x-ray framing camera, hereafter referred to as the XRFC, then registers the silhouette and records the data. The cameras are used to measure the implosion density and uniformity, and several other critical factors of the target implosion.



This is a standard x-ray framing camera that is used by the URLLE. X-ray framing cameras take images by backlighting the target with x-rays, and looking at the degree at which the x-rays penetrate the target.

Figure 1

All of the XRFC's in current use are several years old, and are still characterized according to their factory settings. Perceptibly, the original settings can degrade over time, and without proper maintenance and preservation, the settings can deteriorate to such a point that the image quality and accuracy is severely corrupted. But the cameras are difficult to calibrate, and accuracy can be affected by this. The reason for this is the considerable amount of time it takes to characterize an XRFC. It can take up to eight hours to completely characterize a single camera. A series of reading must be taken, 126 readings total, for each voltage setting and each bias setting. And with a high demand for the camera to be used for target shots, the cameras are rarely available for more than a few hours, let alone the eight needed to properly characterize a camera. The main reason for the tight schedule is the fact that for each individual target shot, the cameras themselves need to be adjusted and choreographed with the other cameras used in the target shot. This alone can take from one hour well up to four hours. Moreover, these cameras are being fine tuned according to the base settings which are more than five years old.

As is evident from the current situation, a faster and more efficient way of characterizing the XRFC's is needed. The proposal made was to design, construct, and test a fully automated module that can take all of the necessary measurements for the complete characterization of the XRFC. The past summer was spent designing the appropriate circuitry, and constructing the actual module. What will also need to be designed and constructed is a program to interpret this information, and put it into a Microsoft Excel file.

The benefits of this project will be that XRFC's will now be able to be characterized, and calibrated for higher image quality. It will take only ten minutes to take all of the readings, contrasting to the eight hours currently necessary. This module will also eliminate the problem of human error in the characterization process. A single mal-recorded setting could throw off the entire characterization of the XRFC. This would effectively cancel out eight hours worth of work, and forfeit an entire day's worth of possible use of the XRFC.

*   *   *   *   *   *   *   *   *   *   *   *   *

A module for the XRFC needed to be designed, refined, and constructed. The majority of the program was spent on the design and on refining the design. A program called *ORCAD Captu*re was used to create a draft schematic of the module. This program uses a series of symbols, all of which represent a circuit part in real life, to model the schematic. The overall schematic took roughly 2 weeks to complete and refine. It comprised a PIC Microcontroller, several bidirectional buses, and two RS232 converters. There was also a secondary board, the high-voltage (HV) relay board, made to function simultaneously with the main board. The secondary board would handle the flow of the high voltages put out by the XRFC. Voltage levels could reach in excess of 4000 volts. Several safety precautions were taken in the designing of the module in which the circuitry was to be placed.

The actual enclosure in which the circuitry was going to be put was an aluminum flanged inclusion. This would allow the box to be grounded in case of any shorts or any surges. Also, the HV relay board was coated in a non-conductive material in order to isolate the current. Extra wide traces were etched into the board for added safety. Lastly, the HV relay board and the main board were kept a distance away form each other in the enclosure. All of this had to be taken into account when designing the circuitry.

The enclosure itself is approximately 8” x 4”. This can put some restraints on the spacing of the circuitry. The boards needed to be fairly compact, although the density of the components was not too tight.

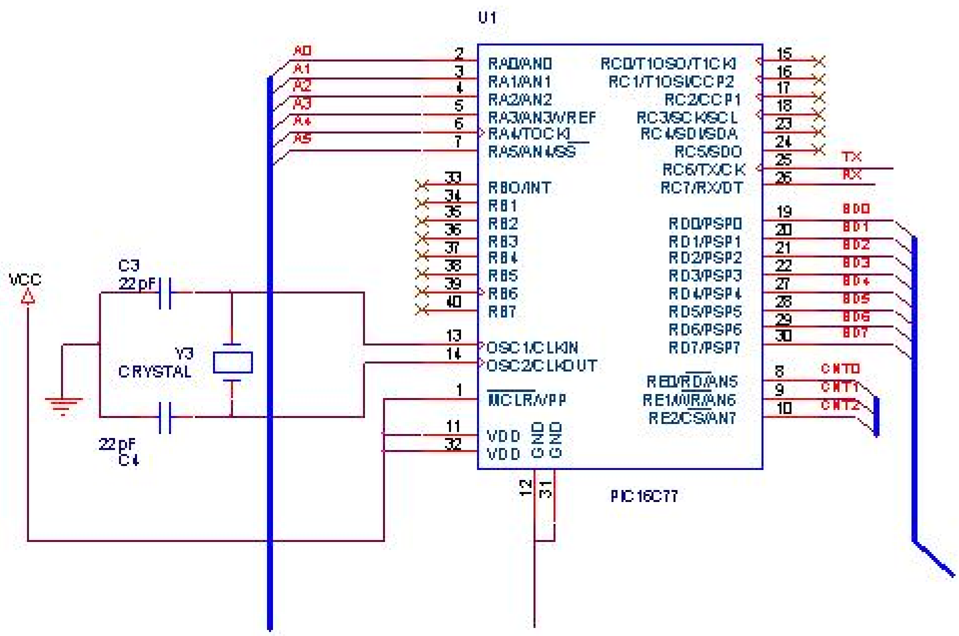
We were able to plan all of this out using a sister program called *ORCAD Layout*. This program allowed us to import an *ORCAD Capture* file and have the Layout program actually draw out the connections and give us a 1:1 scale of what the finished product would look like. About 3 weeks was spent on the layout. Part of the reason this process takes so long is the many errors that occur when drawing a part. Every single variable is checked by the computer and matched up with factory standards. If something is not up to spec, then the program will identify the error and where it is located.

After the boards were completed in Layout, the files were converted to a factory file. A machine can interpret the information in these files and generate the boards. The boards were produced off site because of the need for high efficiency and quality. There was not the necessary time to allow a board to be produced at the lab. The files were emailed out and the board was manufactured and sent back about three days later.

Upon receiving the board, the process of populating the board started. The board comes bare, with only the traces etched in and the holes and vias drilled. It is the customer’s job to solder in all of the components which are to be placed on the board. This is a relatively quick process, depending on board size. Populating the board took about one afternoon to do. After populating the board, it was tested and measured under low voltage. The HV relay board was also constructed and populated, and was not tested for safety reasons. It was simple enough that testing was not required.

Several problems occurred with the main board. All were easily remedied within a few days. After the problems were solved, the board was put into the aluminum enclosure with the HV relay board and was attached to the power supply. A final test was performed, and everything checked out.

At the very start of the program, all that had been accomplished was a list of the needed parts, a basic part requisition, and a general schematic of the boards that were to be produced. The schematic lacked RS232 converters; completed wiring for the PIC Micro controller, resonators, and the design of the HV relay board still needed to be completed. All of this needed to be designed and refined before any prototyping could be completed. There was also a considerable amount of time taken in order to double check all of the components and placements. The schematic was completed within a few weeks. Many capacitors were also put in place. The capacitors would smooth out the electrical path and keep many of the electronic components in the circuit safe.



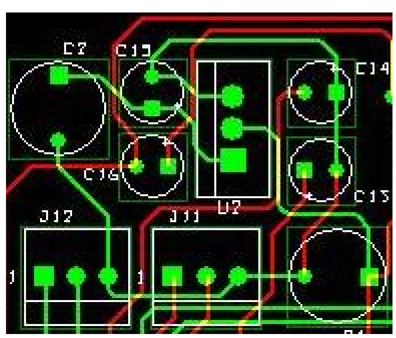
This is the PIC Microcontroller in ORCAD Capture. It has a resonator wired to it externally, as well as several busses leading away to the HV Relay Board, twin 6850 chips, and the RS232 converters.

Figure 2

The PIC Micro controller (fig. 2) was the main focus of the board, and virtually everything was connected to it. The final wiring of the PIC required careful planning and understanding of each of the output areas on the controller. Also, resistor packs were needed to control the amount of current and voltage that was being passed into the busses. The correct resistors needed to be found and placed in proper conjunction with the traces in the diagram. Finally, the connectors and HV relay board needed to be designed. For connectors, simple six-prong connectors were used. Instead of having the RS232 9-PIN connectors on the board, we lead wires between the RS232 db9-PIN and the db6-PIN connectors on the board. This allowed easier construction and the ability to replace the boards without replacing the entire network of db9-PIN connectors.

The 4000 V HV relay board was fairly easy to design on the *ORCAD* program. Consisting simply of five HV relays and a single six pin connector, the board took very little time relative to the main board. But careful consideration would need to be taken in order to ensure safety during construction.

One final step of the design of the two boards was the gathering of the "footprints" for each individual part. A resistor has a different footprint than a capacitor, and two capacitors may even have different footprints from each other (fig. 3).



There are several different capacitors in this image named (C2, C13, C14, and C15). Each has a different footprint than the next.

Figure 3

There were several minor footprint errors associated with the layout we had designed. We discovered this upon uploading the design files into *ORCAD Layout*. The program proceeded to give several PCB footprint errors. It appeared that there were errors in the footprint files. The footprint files appeared to be incapable of converting to *ORCAD Layout* files. These footprint problems consumed much of our time while we located the source of the problem. There appeared to be a faulty piece of data in the footprint files we had provided the program with. After the problem was resolved, transfer of the *ORCAD Capture* files was carried out, and then the design in *ORCAD Layout* commenced.

The *ORCAD Layout* program allows a designer to place traces on the bottom or top of the board, making it possible to wire up very complex circuits. Silk screen layers are also applicable, along with many other features. We used *Layout* to finish up the final wiring of the main board and the HV relay board. The main board wiring went on without a problem, but careful consideration of the HV relay board's high voltage traces needed to be taken into consideration during its final design. Extra wide traces were put in place, along with the consideration of isolating the high voltage traces. Each trace was placed a minimum distance form any other trace. Also, traces were kept as far away as possibly from any of the high voltage pins on the relays.

There were several problems that needed to be corrected. One problem was the two 6850 IC chips had been manufactured with the wrong footprint. This problem was easily corrected. We constructed a small daughter board that had the correct footprint for the 6850 chips. This entire process took no more than a couple of days. And as a final safety precaution with the relay board, an insulating gel was coated over the traces to further isolate them from any arcs that might occur during operation.

The final step in the completion of this phase of the project was the manufacturing of the aluminum enclosure for the HV relay board and the main board. We had come to the conclusion that the relay board was going to be in a separate enclosure from the main board. This would isolate the high voltage from the main board, and would allow proper grounding of the relay board. Also, the second enclosure for the HV relay board needed to be taller in order to provide adequate clearance for the HV relay board. The board needed to be a considerable distance away from any metal surfaces as a safety precaution. Both enclosures, the main board enclosure and the high voltage enclosure, were joined by an insulated six pin connector. The enclosure was sealed, and completed within a week, and the boards were secured inside along with the power supply. This completed my phase of the project. The programming of the code, a GUI, and the PIC coding still will need to be carried out.

Acknowledgements

Dr. S. Craxton

Mr. Lonobile

Mr. S. Mott

Mr. R. Bahr