Evaluating Altium for Circuit Simulation Using the EIC – DIRC Laser Interlock Circuit

Tyler Lemon, Mary Ann Antonioli, Peter Bonneau, Aaron Brown, Pablo Campero, Brian Eng, George Jacobs,
Mindy Leffel, Marc McMullen, and Amrit Yegneswaran
Physics Division, Thomas Jefferson National Accelerator Facility, Newport News, VA 23606
March 7, 2023

This note discusses an evaluation of Altium's circuit simulation capabilities using the EIC-DIRC laser interlock circuit.

The Detector Support Group (DSG) frequently develops and designs custom PCBs, the latest a PCB for the EIC-DIRC laser test area's interlock system. Frequently, during the initial design process, there are uncertainties in component selection or how components should be connected to get the desired behavior. In these situations, a simulation of circuit components and behavior is extremely helpful. For PCB design, DSG uses Altium, which also has a built-in simulation peripheral. To evaluate this simulation feature, the EIC-DIRC laser interlock circuit schematic was modified and simulated.

Before running the simulation, a schematic preparation is needed. First, the power inputs in the circuit must be replaced with simulation sources. Next, any component without simulation models must either be replaced with generic models or a model must be found for that component. Then, any interlock input to the circuit must be replaced with a simulation component whose value can be changed over time. Finally, circuit outputs must be replaced with components that can indicate the output status, like LED models, or shorted out and simulation probes placed to monitor the output.

For the EIC-DIRC laser interlock circuit simulation, the schematic preparations involved replacing specific logic gate components with generic, simulation-capable models. All logic gates selected had generic models available with the exception of the eight-input NAND gates. These two components were each replaced with generic two-input NAND gates. This resulted in one input channel being available for monitoring an interlock signal, with the remaining three monitoring interlock circuit status readback signals. Additionally, all Schottky barrier diode sets were removed from the schematic because they did not have equivalent, simulation-capable parts. Removing these parts instead of creating a custom model was selected as the course of action because in normal, steady state operation, they do not affect the signals the diodes are connected to since their purpose is to route excess current to ground or the power supply during over-voltage or under-voltage conditions. An additional preparation was to remove any connectors on the schematic for LED indicators, replacing them with simulation probes. The probes effectively behave the same way as the LEDs by reading 0 V where the signal is low (LED off) and reading 5 V when the signal is high (LED on). The final step in the preparations was replacing any input whose value would be changed during the simulation with a pulsed DC source configured to mimic the voltage levels if a user presses a button or an interlock trips.

Once the schematic has been modified, the simulation parameters must be configured. Altium has four simulation modes—operating point, DC sweep, transient, and AC sweep. In operating point mode, the circuit is run in DC steady state to simulate how it would behave in normal, steady-state operation, allowing component power dissipation, voltage, and current draw behavior to be observed. DC sweep analysis mode allows one or more voltage sources to be increased from a starting voltage to an ending voltage with set voltage steps (all parameters are user-settable). DC sweep mode allows circuit behavior for a wide range of DC voltages to be studied in a single snapshot. Transient analysis mode runs for user-set duration with set intervals between analysis points. The difference between operating point mode and transient is that in transient analysis mode, parameters of the circuit (input values and voltage levels, for example) can be modified at certain points in time to see how the circuit behaves with those changes. AC sweep analysis mode allows the frequency of one or more AC source to be swept across a frequency range with set frequency steps (all parameters are user-settable). This AC sweep analysis mode is helpful in seeing applicable circuit's frequency response, like low- or high-pass filters. For the EIC-DIRC interlock circuit, the most relevant mode is transient analysis.

To simplify the circuit before its simulation, two other changes were made to the laser interlock circuit. First, the control unit status readback was hard-wired to have the expected values for when the external control unit is used. Second, the filter readback and control logic gates were hardwired to always have the expected value for when the external control unit is used. Figure 1 is the simplified schematic used for the simulation. Table I contains the voltage pulse parameters used to model the reset button, sweep button, and interlock input.

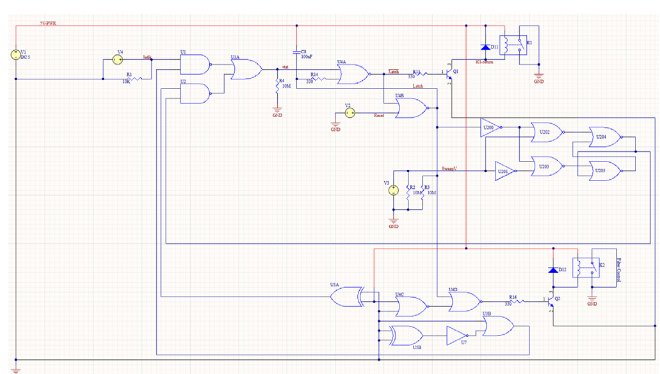

FIG. 1. Altium schematic of simplified laser interlock circuit used in simulation.

Input	Schematic designator	Pulse type [V]	Delay until pulse [s]	Pulse duration [s]
Reset	V2	standard 0–5	2.5	0.5
Sweep button	V3	standard 0–5	2	2
Interlock 1 In	V4	inverted 5–0	7	2

TABLE I. Parameters of voltage pulses used as inputs to laser interlock circuit.

For the output of the simulation, five digital probes were used for the interlock status, latched status, sweep button press, reset button press, and the interlock input. The digital probes relay those signal values in terms of 1 indicating the signal is high (5 V) or 0 indicating the signal is low (0 V). Figure 2 is the resulting plot of digital probes over the 10-second duration of the simulation, showing that the simulated circuit behaved as expected, in the same way as the previously created physical prototype.

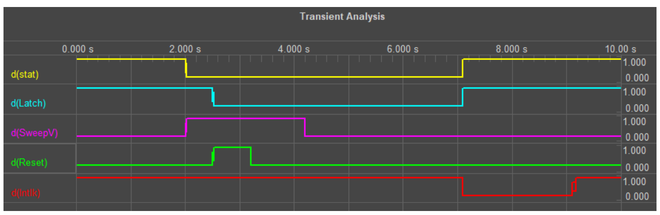

FIG. 2. Output of Altium simulation's transient analysis mode.

For all waveforms, the small box/oscillation feature (for example, at ~9 seconds on the red interlock signal) indicates the output is briefly in an unknown state, which is how a real device would behave as its output changes value, Fig. 3.

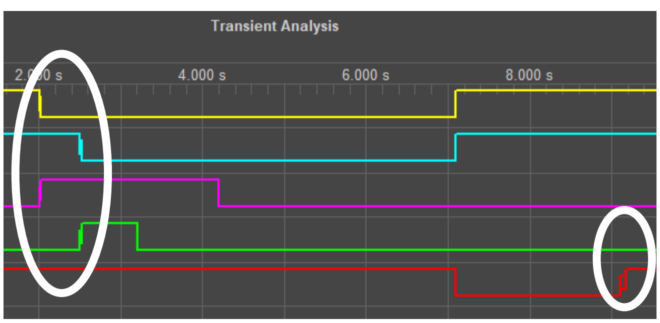

FIG. 3. Zoomed in view of FIG. 2 shows the brief oscillation.

In evaluation of Altium as a circuit simulation program, a few points were learned. First, Altium does not have an interactive simulation that allows the user to change parameters as the simulation runs, mimicking sitting in front of a physical circuit and changing parameters. While this is not a problem, it just requires more thought in setting up the circuit simulation to properly configure the circuit inputs and monitor the desired outputs. Additionally, since not all components have simulation models, it is best to perform the simulation before selecting specific components for the circuit to avoid having to make modifications to the circuit schematic. Altium simulation is an essential tool in DSG's tool box for simulating circuits.