

Cadence OrCAD and PSpice

Computer models can be used in a variety of applications and processes. They are used more often than one might think, from weather analysis to the stock market to twitter trends. More often than not though, they are used in development and design applications. As an electrical engineering major, I use models in my everyday curriculum. One of the most prominent and useful models I have been able to use is OrCAD Capture CIS and PSpice.

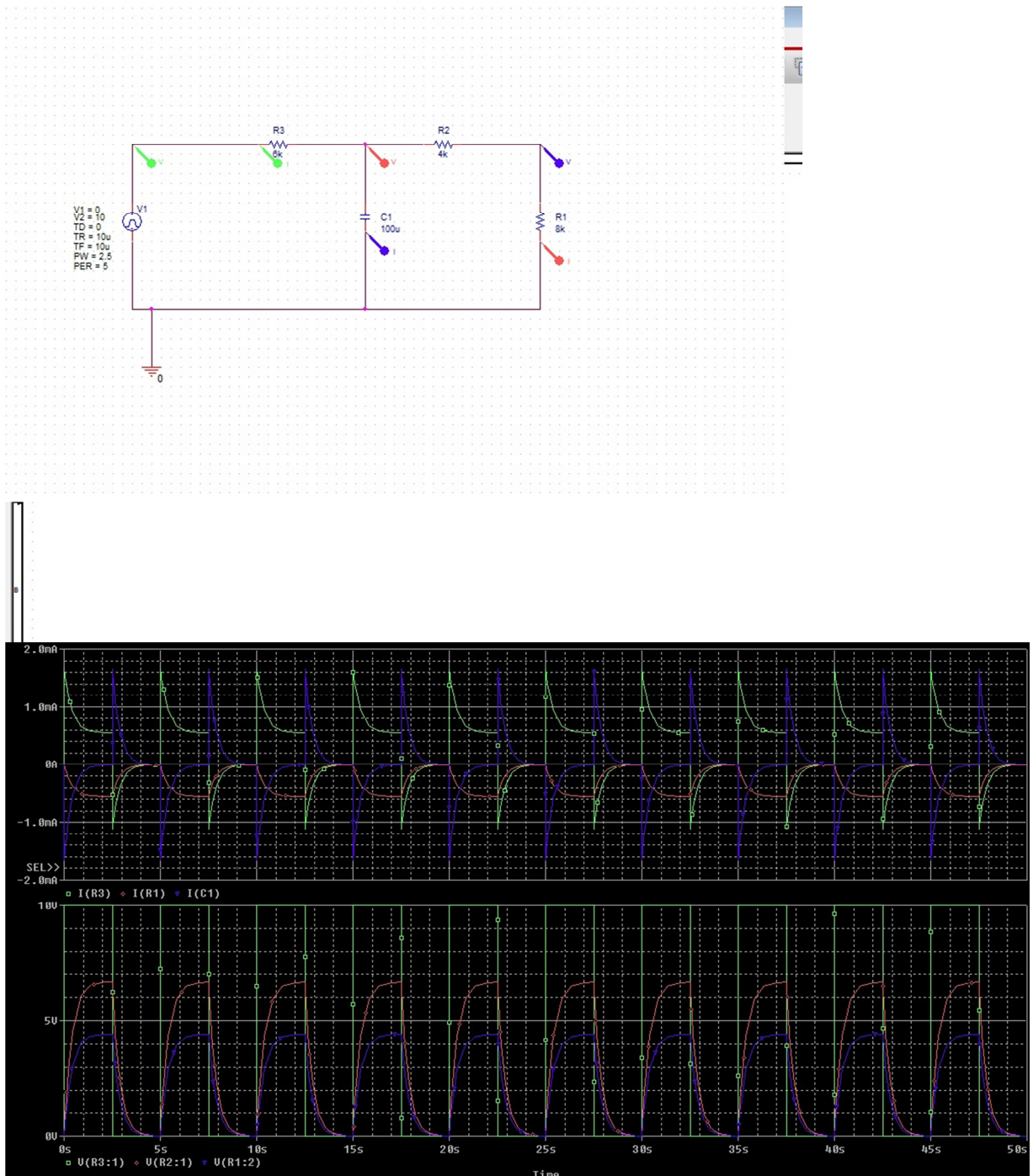
These programs are powerful applications used to design and simulate circuits digitally, rather than constructing the circuit and testing it manually. In electrical engineering, this is an extremely valuable asset to have. Testing a wired circuit is more tedious than one might think, when all the different methods required for testing are taken into account. If you have resistors and a voltage source only, to measure just the current and voltage of the individual elements you would need a multimeter and multiple arrangements of this multimeter just to test those two values of a circuit element, not to mention different processes to achieve these measurements. To measure voltage you must know where to place the probes, and for current you must reconstruct the circuit to pass its current at a particular place through the multimeter to achieve a reading. If you add a capacitor, inductor, or alternating current source, and even more hardware and processes are needed. These include function generators and oscilloscopes, which must then be tuned to the proper specifications and wired into the circuit appropriately to achieve the required readings. The OrCAD program solves these problems with ease, once you understand its operations and processes. All these readings can be obtained by placing a digital probe where you need it in your virtual schematic and running a simulation.

This program is fairly large, but considering all it can do it doesn't require much of the system. You need a Pentium 4 (32 bit) equivalent or faster, Windows XP, Vista, 7, or Server 2003, a minimum of 512MB of RAM, although 1G or more is recommended, 300 MB swap space or more, a CD-ROM drive, and 65,000 color Windows display with minimum 1024x768, but 1280x1024 is recommended. Needless to say a better system will run it faster and more efficiently and fluid, but these are the minimum requirements. It only requires one person who knows what they are doing to operate, which is great for companies looking for an efficient way to get a lot done with their circuitry R&D. It is fairly expensive, with a price tag of \$4355.00, but honestly for what all it can do, that's not such a bad price. It will pay for itself with savings in testing equipment and salary for workers having to perform all the measurements. This software is obviously a simulation model. The circuit is built in a virtual schematic, then a simulation profile is created and parameters are set, then you run the simulation and the results are stored in a profile. By being able to simulate all this, it saves parts, time, labor, and employee resources, which greatly benefits the field of electrical engineering.

The inputs for the simulation are the virtual schematics, the virtual parts list (which every parameter of a part can be modified to your individual needs), and the run time of the simulation, to keep things simple. Much more can added and modified, but the detail is too great to go into. The model then takes your constructed circuit, runs it and analyzes with the virtual probes the user placed, and records the desired data. It then outputs a graph of the data however you set it. This can be voltage-to-time, current-to-time, voltage-to-current, whatever the user wants, which can be saved or printed. You can then modify the circuit and run it until the wanted results are obtained. Many large companies and universities use this program, including our very own University of Alabama. It is implemented in many different electrical and computer engineering

classes, and is very useful in teaching the ideologies behind circuits and their behaviors.

Attached are some model outputs from my engineering labs.



References

Cadence OrCAD Capture and Capture CIS. 2011. Cadence Design Systems.

<http://www.cadence.com/products/orcad/orcad_capture/pages/default.aspx>

Cadence OrCAD Capture CIS. 2011. EMA Electrical Design Automation.

<<http://www.ema-eda.com/products/orcad/capturecis.aspx>>

<<http://www.ema-eda.com/store/t-products.aspx>>

Jaber, Abu-Qahouq. Electric Circuits 225 Laboratory- Lab 1. Houser Hall, University of Alabama, Tuscaloosa, AL. 13 Sep 2011.