Micron Team Clinic Proposal 2010-2011

Multi-Phase Analysis of Power Delivery Networks on Large-Scale Circuits

Lavander Begay

Travis Fiehler

E. Michal Peterson

Thomas White

Advisor: Ken Stevens

Abstract

Modern day circuits have grown exponentially over time due to the dramatic decrease in the size of transistors. Noise on on-chip power delivery becomes a problem when dealing with the lower voltages on which these transistors operate because there is less room for error. On large-scale circuits, CAD tool simulations become lengthy and cumbersome for computers to complete accurately and efficiently. We propose the use of a method to use multi-phase simulation which uses the noisy power supply created from an earlier simulation for the power supply of the current simulation for the core of our project. We also propose to analyze different methods of simplifying a simulation (like replacing parts of large circuits with voltage controlled resistors or other simpler models) and the accuracy of different simulators that could shorten simulation times (namely HSIMplus and Apache).

Introduction

As integrated circuit (IC) operating frequencies and signaling data rates continue to increase the characteristics of the power delivery network (PDN) appear less and less ideal and the non-idealities of the PDN become more problematic, because there is less voltage and/or timing margin to work with. The ideal function of a Power Delivery Network (PDN) is to provide a low impedance path to deliver clean power to the IC's while not contributing excessive Electromagnetic Interference (EMI). Currently, simulation software models of circuits become extremely cumbersome for accuracy and simulation time when fully modeling a multi-billion transistor system due to the complexity of transistor level modeling. A multi-phase approach combined with simplified transistor models could potentially allow the simulation of multi-billion transistor systems to be modeled with some degree of accuracy.

The proposed project will investigate replacing current simplified models used in the two-phase simulation approach, with a Voltage-Controlled Resistor (VCR) in order to reduce simulation times. In conjuncture with the VCR, a multi-phase analysis, which uses a noisy power supply created from an earlier simulation as the power supply for the next simulation, will be investigated as a way of improving simulation accuracy. Additionally, a new circuit analysis tool from Apache will be compared against HSIMplus for accuracy and simulation time.

Background

During 2005, the majority of power related challenges were done on chip packages at sizes 130nm and larger. At 90nm and below, most designers had not yet experienced the severe power-related challenges that required implementation of new design flow rules. Research was done to determine the effects that this noise has on the timing and signal delay. Voltage (IR) drops and inductance noise within the circuits were found to be the main causes of these errors. Both inductive noise and IR noise types are caused by instantaneous current drawn by the circuit components. More specifically, the inductive noise is caused by the current change on the package leads and/or the inductance created on the point to point wire contacts. The IR drops in the circuit are created by the currents hitting the resistive power and ground lines. The static IR drop flow should be used to verify that the power delivery network is sufficiently robust to establish a baseline for IR drop analysis prior to running a dynamic IR drop analysis [1].

Dynamic power analysis is a transient simulation that evaluates the IR drop caused when a large amount of circuitry switches simultaneously, creating peak current demand on the power rails. To accurately determine these currents and voltages the simulation must have all resistor/capacitor (RC) segments included in the layout. Static power analysis is a vector-less simulation that evaluates the IR drop caused by high average currents flowing through a design's resistive power rails. Static power consumption consists of three components: switching power, internal power, and leakage power. Static power is calculated by averaging current over a specified period of time. Static analysis when compared to dynamic analysis provides less accurate current and voltage values of the power and ground segments [1].

Two-phase analysis uses a combination of dynamic and static methods to create an analysis method that has the strengths of both. The first phase uses dynamic analysis to capture the currents of a cell, while phase two creates a simplified representation of cell currents. Furthermore, it has been shown that phase two uses the dynamically created current models as piecewise linear sources replacing the actual cell to simulate the effect on the power delivery network [1].

Last year's team had conducted a study into the two phase approach using HSimPlus. They found that when they ran simulations on circuits, it took the computer a long time to simulate the voltages. As the circuits became larger due to transistor count, the simulation times increased as a result. They also found that there were propagation delays created in the circuit as a result of the parasitic capacitances and inductances contained within the power delivery networks [2]. If the noise created within the chips were reduced, the delays would be reduced as a result.

Methods

Each of the parts of the project spoken of above will require different methods and areas of special study to accomplish. These particularities are as follows:

1) The multi-phase analysis performer will first be required to learn a scripting language of some sort so that a script can be generated that will take the output files of a simulation result and change those results into an input

power supply model to be used in another simulation. Another script may also be implemented to run the simulations back to back rather than manually starting a simulation, running the file alteration script, and then running it through another simulation, for as many repetitions is desired by the performers. Knowledge of the simulators and models will also be necessary for this part of the project because SPICE models of power supplies will have to be generated from the simulation outputs to make them usable as inputs to another circuit, or part of a circuit. The major simulator used for this effort will be HSPICE, and thus knowing how to use this simulator will be required. HSPICE will be used because it is the understanding of the clinic team that the reason for the multi-phase analysis is to attempt to use the most precise simulator in a way that will make it simulate large scale circuits in a timelier manner. Understanding of circuit behavior is also an important requirement for this part of the project because it will be entirely up to the performer where and how to divide up the larger circuit to run the multiple back-to-back simulations to get a complete analysis.

2) The simplified modeling of a block of circuit (using the VCR) will require the performer to utilize great knowledge of SPICE modeling, and functionality of SPICE simulators. Specifically it will require the performer to determine how to create SPICE models that will implement the VCR in a standard design flow. There may be models available for the VCR, but they are currently unknown to the group performing the project. This means that the person performing the modeling will most likely have to generate a unique SPICE model and determine how to make the simulator recognize that as a model to use when simulating a circuit block. Varying model parameters could be used to make the simplified model match the part of the circuit being replaced more precisely. It will be the job of the person doing this analysis to determine how

much precision is beneficial before simulation time again becomes a big issue.

3) The in-depth simulator comparison will mostly require the performer to understand how the simulators work and how to use them. This portion of the project mostly consists of putting together a large amount of information about the performance of the different SPICE simulators available to the team. These simulators are HSPICE, HSIMplus, and, potentially, the Apache simulator. The person performing this portion of the project will run many simulations on each of these simulators and analyze all of the results from them, so that a good summary of the differences in performance of these simulators can be written. The intent here is to simulate all of the existing project circuits, and possibly some new ones with both of the package models available, and compare the outputs for accuracy and efficiency.

Contingency Plan

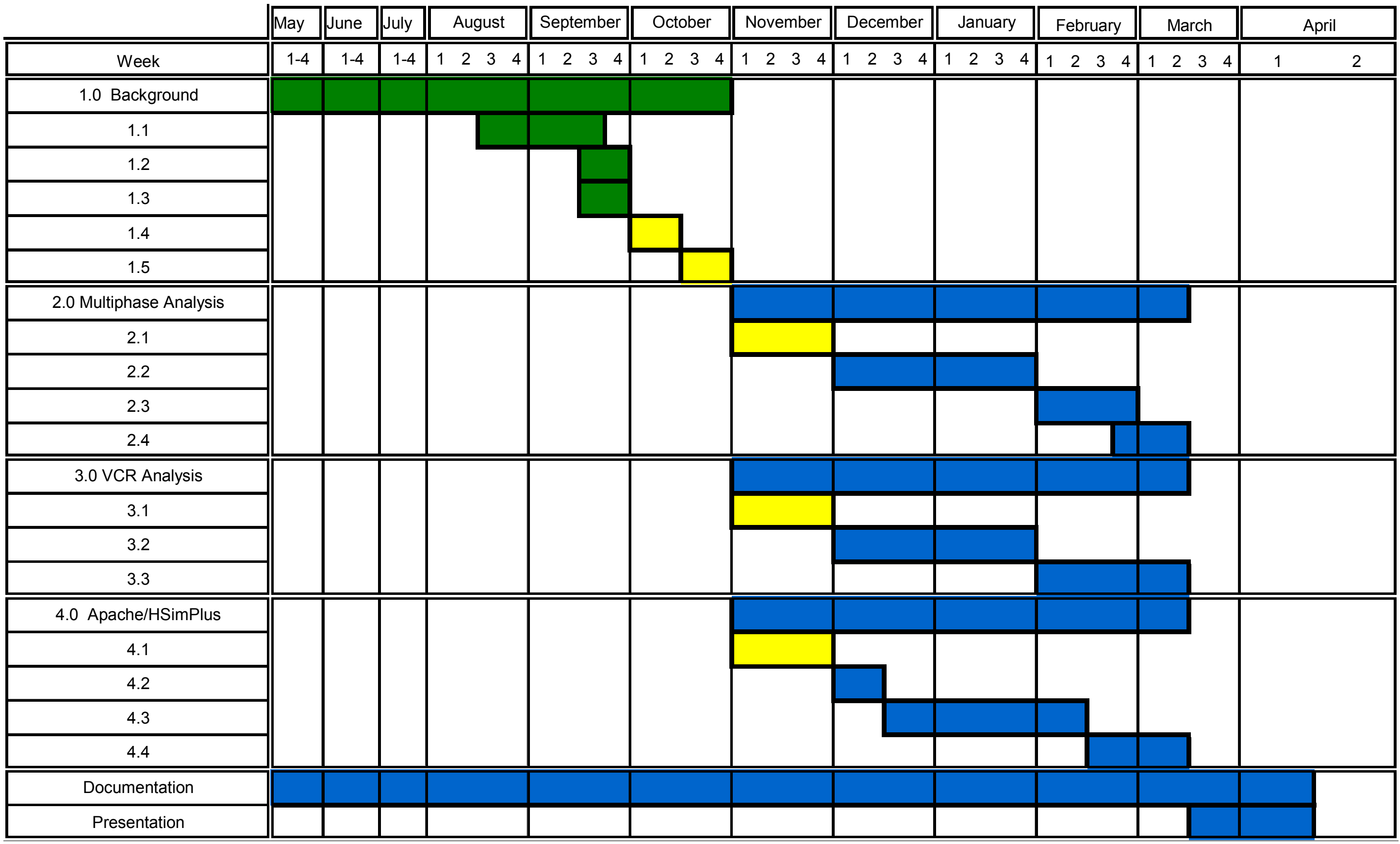
As we progress through this proposed project we understand that not all of these goals may be realized. There may be several areas of concern to us that may not allow us to accomplish all of the tasks we have taken on. Our priority is to develop a strategy for the multi-phase analysis, to be able to reroute output signals back into the input to accomplish some sort of a steady state input.

The person tasked to work multiple simulations with Apache and HSIMplus, for determining accuracy, speed, and efficiency, may not be granted access to the Apache software because of licensing issues or price of access. If that is deemed to be the case, this person will assist with determining whether or not the voltage-controlled resistor is a viable simplification model as opposed to multiple current sources.

At the point we determine that we are not able to simplify the models with a voltage-controlled resistance, the modeling becomes too difficult to be accomplished in the time allotted to the group, or the voltage-controlled resistance is deemed not practical as a simplification model, we will have the person(s) working on this portion to be shifted over to assist in working on the multi-phase analysis branch.

The multi-phase analysis has been deemed the core of our project, and we wish to have the most success within this portion. To begin with, we will start with small and simple circuits, to determine the viability of this process. We will be starting off with circuits that were used in last year's project. As we develop confidence in this method, and the approach has been deemed successful, we will begin using this method on larger, more difficult circuits. We will progress this way until we have enough data on multiple circuits to prove or disprove viability of this method or until we have used the appropriate amount of time for this project.

Gantt Chart



Tasks:		Deliverables:
1.1	Familiarize ourselves with previous team's project	
1.2	Instruction on teamwork, defining team roles	Team roles
1.3	Defining project scope	
1.4	Proposal and presentation for Micron (Rough Draft)	Rough draft
1.5	Proposal and presentation for Micron (Final Draft)	Final draft
2.1	Review output and input files	
2.2	Develop translation script(s)	Functioning scripts
2.3	Run simulations with last year's project circuits, to compare known output to unknown output for analyzing	Graphs, charts, informational session
2.4	Analyze outputs for circuit multi-phase analysis	Documented results
3.1	Research voltage controlled resistors (VCRs)	
3.2	Create VCR model(s)	VCR SPICE model
3.3	Testing and analysis of various VCRs for viability determination	Documented results
4.1	Learn Apache and HSIMplus	
4.2	Research circuits for simulation	
4.3	Perform simulations on Apache and HSimPlus	Simulation results
4.4	Compare and analyze data of both HSIMplus and Apache simulations for accuracy, speed, and efficiency	Comparison documentation
Documentation	Recommendation(s) to Micron for efficient PDN analysis, publishable paper	Published paper

References:

[1] Ammon Hardcastle, Kritian Blomquist, Jason Onieda, Micheal Tomer and Sze-Hsiang Harper, “Proposal for Micron Project: Two Phase Analysis” Oct. 4, 2010.
<https://docs.google.com/Doc?docid=0ASbZAvUhEjQyZGc0a242eG5fMTRnMnZibWh6eg&hl=en>

[2] Ammon Hardcastle, Kritian Blomquist, Jason Onieda, Micheal Tomer and Sze-Hsiang Harper, “Final Report: Two Phase Analysis” Oct. 4, 2010.
<http://code.google.com/p/uofu2009-2010micronclinicteam/ >