

CUDA GPU Accelerated VLSI Circuit Modelling

Solving circuits is non-trivial, and finding a parallel solution on the GPU is even more so. An accurate physical model of just a single transistor introduces much complexity, as physical devices introduce parasitic elements and non-linear effects.

The industry standard for circuit analysis is SPICE, Simulation Program with Integrated Circuit Emphasis, integrated into many modern simulators. I plan to base my project on the algorithms used by SPICE, and use CUDA GPU programming, but limit the scope to circuits with ideal resistors and capacitors and non-ideal transistors to limit complexity for the available timeframe.

While I do not expect produce an industry-ready solution, I would like to take this opportunity to explore the current CPU solutions I have taken for granted and re-imagine them on the GPU. This will incorporate both my studies and experiences in Electrical Engineering and the class focus on parallel processing on the GPU.

Milestones

11/19 Milestone 1:

- Finish researching and gathering literature on topic
- SPICE netlist and model parameter file parsing
- Outline & begin necessary code for 1st order solutions of simple circuit blocks

11/26 Milestone 2:

- Be able to solve a DC simulation of simple circuit types, compared against hand calculations
- Add in 2nd order and higher effects if it works

12/03 Milestone 3:

- Transient simulation – figure out how to model complex RC ladders accurately in time
 - Start with lumped or Elmore delay model, or go straight to equations?
- Work on some graphical data representation or data export

12/10 Final Deliverables:

- DC and transient simulations on GPU for reasonably complex circuits
- Plot voltage and current data for any device or node in circuit
- Result accuracy comparable to other simulators on the market (e.g. Cadence, ngspice)

References:

[SPICE Acceleration on GPU](#)

[CUSPICE](#)

[SPICE Algorithms and Internals](#)