

Circuit Simulation

some humbling thoughts...

Manfred Wendt

Fermi National Accelerator Laboratory, Batavia, IL 60510, U.S.A.¹

Abstract. A short, very personal note on circuit simulation is presented. It does neither include theoretical background on circuit simulation, nor offers an overview of available software, but just gives some general remarks for a discussion on circuit simulator needs in context to the design and development of accelerator beam instrumentation circuits and systems.

BEAM INSTRUMENTATION DEVELOPMENT: FOR WHAT DO WE NEED CIRCUIT SIMULATORS?

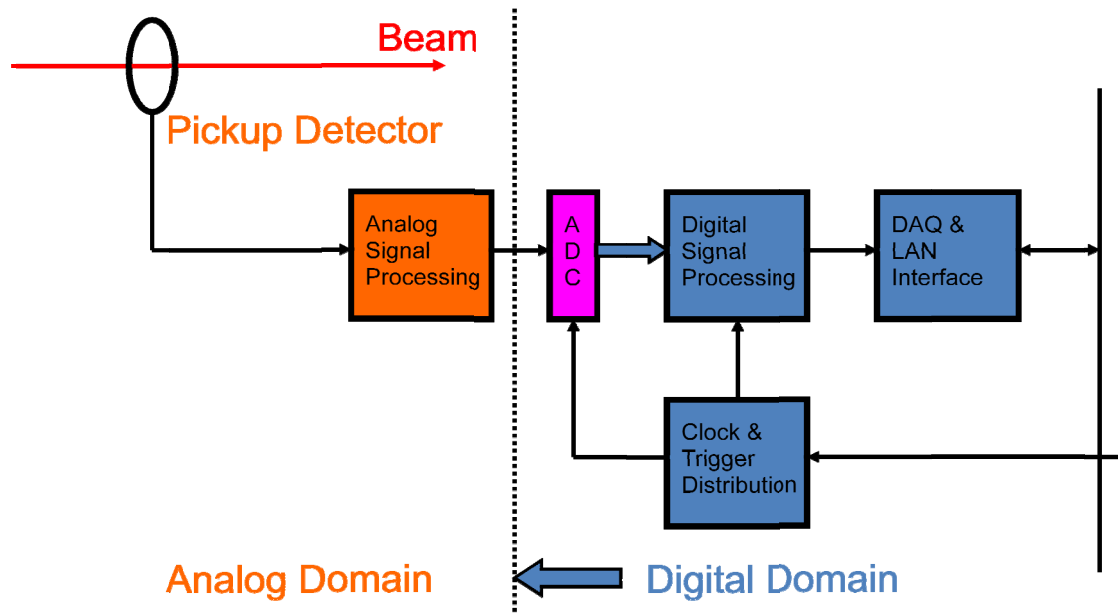


FIGURE 1. Schematic of a typical beam instrument, e.g. BPM, toroid, WCM, PMT, etc.

Fig. 1 illustrates a typical beam instrument used in a particle accelerator, consisting out of a pickup detector (often part of the beam pipe vacuum system), an analog front-end (Analog Signal Processing), and several “digital” blocks including the analog-digital converter (ADC). Before spending lots of funds in money and time for purchasing and working with a sophisticated circuit & system analysis software

¹ This work was supported by Fermi National Accelerator Laboratory, operated by Universities Research Association Inc. under contract No. DE-AC02-76CH03000 with the United States Department of Energy

product, it is worth to discuss how and for what circuits it may be applied in the development of a beam instrument. Some personal observations are:

- As Fig. 1 indicates, the borderline between analog and digital domain in the signal processing chain is pushed further and further towards the detector, and this trend will remain.
- “Classical” circuit simulation tools, e.g. SPICE and its variants (time domain), Touchstone and similar products (frequency domain) are focusing on the circuit level simulation, usually in the analog domain.
- Model data is unavailable for many complex semiconductor components, e.g. ADCs, DACs, track&hold amplifiers, limiters, etc.
- Digital signal simulation is mostly based on vendor supplied simulators, and very product specific, e.g. Quartex II for Altera FPGAs, etc.
- Even today, there is no single simulation tool available to simulate a complete beam instrument, starting from the beam, through pickup, analog and digital signal processing, timing and triggering all the way up to the data management level at the LAN adapter.

A conclusion of these observations should not be to exclude circuit simulation from the development of a beam instrument. Even though a complete front to end simulation is neither available nor desirable, critical parts or subsystem (particular analog sections operating at RF or microwave frequencies) of the beam instrument may deserve an in deep circuit simulation and analysis, e.g.:

- Gain stages
- Passive sections (filters, diplexers, hybrids, sections with distributed components, i.e. transmission-lines).
- RF & microwave circuits
- Layout effects, cross-talk, reflections and grounding
- Noise and temperature analysis
- Pickup detectors (button, stripline, cavity and other monitors)
- High-speed digital I/O (CLK distribution, PECL & LVDL circuits, single-ended to differential transitions)

CIRCUIT SIMULATION SOFTWARE

We may categorize circuit simulation software as follows:

- **“Brainware”**: Use pencil and paper and apply the Kirchhoff’s and Ohm’s laws for simple, often idealized sub-circuits. Big advantages are, in-deep understanding of the circuit and an analytical result. Unfortunately this method is limited to linear, sometimes oversimplified circuits.
- **Freeware**: The best known time domain circuit simulator – SPICE – actually is a freeware product, developed at the University of California at Berkeley. The latest version (3f5), as well as many variants are free available to download (google “free spice simulator”).
- **\$\$\$ware**: Professional circuit simulator products are often based on the original Berkeley SPICE, but add some handy features like schematic entry, a very flexible output display, data management, etc. SPICE or equivalent

time domain circuit simulators are often combined with, or added to layout systems. High-end “design suites” consists out of a set of different simulators (often called “solvers”), sometimes including electro-magnetic (EM) modeling.

Circuit simulators:

- Fully **analyze** an electrical circuit based on implemented models for the circuit components.
- Often have limited **syntheses** capabilities.
- Usually solves the circuit topology by **numerical approximation**, thus no exact, analytical result is available.

The models for the circuit elements, i.e. the mathematical definition along with the parameters, used for the numerical computation in the circuit simulator, are of crucial importance:

- Lumped circuit elements are usually based on the ideal model formalism, e.g. capacitance, inductance, resistance. A real-world circuit component (capacitor, inductor, resistor) is more complex; typically has frequency, temperature, etc. dependent characteristics, “stray” elements and often non-linear effects.
- Distributed circuit elements are modeled based on known analytical solutions or approximation of the EM problem, usually solved in the frequency domain. This implies a limited parameter range, and a Fourier transformation (inverse FFT) has to be applied to use the model in the time domain.
- Nonlinear effects, temperature characteristics and noise behavior are available for some circuit models.
- Some circuit simulators offer the implementation of user specified circuit models and/or stimulation sources.

Circuit simulation software uses:

- **Linear solvers:** DC, AC, S-parameter, etc. simulation in the frequency domain at steady state.
- **Nonlinear solvers** (e.g. “harmonic balance”): Used for a small signal analysis around a specified DC working point. Offers spectral, distortion and other characterization of the system’s frequency harmonics.
- **Transient solvers** (e.g. “SPICE”): Approximate solution of the circuit’s system of differential equations. This time domain approach includes all transient effects with given initial values. The method may be limited when computing models of highly resonant circuit elements (resonators with very high Q-values).
- **Special solvers:** For arbitrary geometries of resonators, transmission-line components, antennas, etc. EM solvers are available to approximate the geometry applying Maxwell’s equations. An S-parameter based solution is often available, thus the result can be used for further computations in a circuit simulator. Some analytical mathematical software products add circuit simulation routines as add-on to provide an analytical circuit simulation. Professional circuit simulation software usually includes

statistical tools to optimize a set of circuit parameters to a given goal(s), which adds some limited circuit synthesis capability.