SPICE AS AN AHDL

Analog and Mixed Signal conference

by Charles E. Hymowitz
Intusoft San Pedro, CA, 7/94

ABSTRACT
This paper will discuss the following questions: Is SPICE an AHDL and is it a viable alternative to currently proposed AHDL languages? Second, should AHDL constructs or SPICE syntax compatibility be the starting point for analog extensions to VHDL?

THE Requirements Of An AHDL
A variety of different AHDL vendors and publications have proposed numerous requirements for an AHDL. [7, 9, 10, 12, 14, 20] They are shown in Table 1.

Table 1 - The Requirements Of An AHDL
This table is composed of requirements put forth by various organizations and authors.

1) Supports an architecture in which the model libraries are completely separate from the simulator. They are not built into the simulator itself. This would allow easy addition, maintenance, and support of model libraries.

2) Allows modeling of multiple technologies, each in its native units and unique equations, for supporting designs that incorporate both electrical and nonelectrical elements.

3) Vendor-supplied models and user-developed models, including new primitives, are created using the same modeling language, rather than offering users a different language than that used by the vendor.

4) Supports hierarchical model descriptions for simulating large circuits with commonly used blocks and elements. Support for multiple behavioral descriptions of a given entity. Describes any type of device with any number of pins at any level of abstraction.

5) Supports graphical modeling and model synthesis tools for easy modeling of complex devices. Modeling environment should be usable by non-experts.

6) Supports multiple simulation platforms and in multiple simulation environments. Must run on PC environments and be compatible with Berkeley SPICE and major derivatives.

7) Models written on one workstation (e.g., Sun) can be used on another workstation (e.g., HP) and PCs. Models operate in standard CAE frameworks. It must accept Berkeley SPICE models.

8) The language should be an integral part of the simulator, not an add-on.

9) The language should not be vendor specific.

10) The language should be expressible in a form that is both human and machine readable.

11) Supports an existing library of models.

12) Does not prohibit new simulation algorithms from being easily added to the simulator.

13) Uses the same language for both analog and digital models to support continuous time (analog) and discrete time steps (digital) in a single simulation. Supports both s-domain and z-domain descriptions of devices for use in modeling sampled data systems and frequency response. Support for both analog and digital signal/node types.

14) Statistical and reliability information can be assigned independently or correlated to any model properly for Monte Carlo analysis and statistical analysis of device performance.

15) Allows you to mix connectivity statements with mathematical expressions in models. Supports a wide range of descriptive constructs (nonlinear differential equations, transfer functions, and tables, conditional branching)

16) Allows you to reference variables and calculated parameters from other models to set initial conditions in the model itself.

17) Supports design documentation constructs.

18) Syntactical compatibility with VHDL to allow use of VHDL constructs.

Under closer examination, not all of these so-called requirements make sense and some are even contradictory. The main reason AHDL vendors propose requirement 1) is not, as one is lead to believe, the portability of models. In most cases, AHDL simulators are marginally better than simulators based on SPICE 3 if at all. [1] Thus their main offering is the additional language constructs. Hence the bogus separation requirement.

The fact that the underlying code for SPICE's primitive elements is integral to the simulator does not in any way invalidate SPICE's ability to support portable, easily maintainable models. SPICE models are indeed portable and are currently being shared throughout the electronics industry. The fact that the SPICE model syntax is standardized is the key to portability. This assertion that AHDL models are portable and that the AHDL modeling language is a separate entity is merely a false guise. A simple example illustrates this point. The following is an If-Then Else statement from the Mast¿ language used by the Saber¬ simulator.

If (rand_pos == yes) { pos0_err = pos0*random() } else pos0_eff = pos0

The vendor claims that this construct is "separate" from the simulator. This is very misleading. This construct IS separate from the simulator code but it is not INDEPENDENT of the simulator. It does NOT self execute. In order to be used this statement must read in, parsed and understood by the simulator. The exact parallel situacion exists for a typical SPICE statement such as "R1 1 0 1K". By the way, an equivalent If-Then-Else construct in IsSpice4, a popular SPICE 3 based program would be:

B1 pos0_err 0 V= rand_pos > 0 ? pos0*vran : pos0

(AHDL proponents should be aware that similar If-Then-Else constructs are available in most modern day SPICE simulators!)

Acceptance of any language would require all compliant vendors to support the selected constructs (function calls, hooks into the simulator's structure, the passing of data values, etc.) and implement the underlying parsing and solution of such constructs. This would obviously give the vendor whose AHDL was selected a significant advantage. SPICE primitives are based on the common elements used by circuit designers. It is no coincidence that the underlying implementation is hidden in the simulator. This is purposely done to shield the designer from unnecessary complexity so evident in other AHDL approaches. The only difference is that SPICE is already a standard, making the simulator playing field even and its primitive language elements must be given de facto acceptance because they are the basic building blocks in electronics.

In general, currently available SPICE based simulators support items 1 through 12. Items 13 and 14 have either been added by several manufacturers, thus putting pressure on the rest to eventually comply, or are available using public domain specifications and code. While many SPICE programs offer mixed signal simulation capabilities there is only a modicum of standardization. The same is true for AHDL based products. Public domain software such as [3] and the continued acceptance of VHDL models by SPICE simulators should serve to establish a standard. Items 15-18 are either partially available or not actively being pursued.

THE MATURING OF SPICE INTO AN AHDL

SPICE has continued to mature at a pace equal to or surpassing most other simulation technologies. Unfortunately, the vast majority of authors who compare their simulators/languages to SPICE either don't review currently available technology or choose to ignore the academic and industry acceptance of SPICE. Circuit designers should be very wary of comparisons to SPICE by rival products. Most contain inaccuracies that show ignorance or outright dismissal of currently available SPICE features. In speed comparisons, most do not take advantage of modern day features as they do in their own simulators, preferring to use performance numbers based on circuits simulated at the transistor level. The vast majority ignore SPICE's proven abilities in a number of areas, as well as, major innovations that have occurred over the last few years. Today's SPICE simulators are a far cry from the 1970's SPICE2 FORTRAN versions and AHDL proponents should wake up to this fact. Table 2 shows a current snapshot of the features incorporated in today's SPICE programs.

Table 2 - Features of Today's SPICE Programs

Features	**Examples (where applicable)**	**Time Frame**
Simulator Related		
New analyses recently added	AC Sensitivity/Temperature	T
Interactive simulation control language	IsSpice4	T
Simulation Breakpoints	Stop function	T
Conditional statements	If V(3) > 50m goto end	T
Monte Carlo, Optimization, Worst Case		T
2-D/3D Finite element analysis	Transmission line problems	S
Event driven mixed mode simulation		S
Digital timing analysis	.Model And d_and Rise_delay=1n	S
Multi-threaded/Multiprocessor versions	WindowsNT	< 6 months

Modeling & Element Related

SPICE compatibility	R, L, C, Semiconductors, Macro-models	T
Compatibility: exchange of models	Op-Amp vendors	T
Piecewise linear tables (Discontinuous)	IBIS Models	T
C code subroutines	New Models (See Modeling Features)	S
Nonlinear Diff Eq.	Int./Diff. Blocks	T
In-line equations (transfer functions)	SPICE 3 B element	T
Arithmetic operators	Summers/Multipliers	T
Mathematical operators	Trig/Transcendental	T
s-parameters in and out	RF Device Modeling	F
Z/S-domain expressions	PLL/Digital Filters	T
Switches, Transmission lines, Magnetics	SPICE 3	T
Combinational logic (Boolean expressions)	V= ~ (V(1) & V(2))	S
Conditional statements (If-Then-Else)	Limiters	T
Mix digital and analog models	ADC - DAC	S
Special functions: state machine	combinational logic/unlimited states	S
Explicit differential equations in subcircuits	V=V(1)*Int(V(2))	TF*
Calculated model parameters in subcircuits	V=V(1)*@r1[p]	F
Time and temperature in subcircuits	V=V(1)*TIME	S
Event Scheduling		S
New Models	BSIM3/Lossy Line	T

In Table 2, T = Today, a condition where most SPICE simulators have this feature, S = Some, a condition where the major SPICE vendors have this feature, and F = Future, where this feature could be easily added. *While differential equations can not be typed directly into a SPICE netlist, differential equations can be modeled through the use of subcircuits that perform the integration and differentiation functions.

SPICE has been in existence for over 2 decades. In that time many changes have taken place. The code has moved from FORTRAN to C and a number of new built-in models, analyses, and functions have been added. While most of the initial changes were a result of U.C. Berkeley's contributions, the last few years have seen contributions by CAE software tool developers and other academic institutions. [3] In addition, as features have been added by one vendor, other vendors have adopted them as well. Examples include parameter passing, Monte Carlo analysis, analog optimization, behavioral modeling using equations and conditional branching, various convergence algorithms, and the incorporation of digital simulators.

Modeling Features

AHDL proponents have claimed that SPICE's reliance on macro-modeling limits its modeling capability and that adding a primitive element to SPICE is difficult. SPICE contains support for modeling on many levels. At the highest level, designers need a general expressive method for describing the function of entire circuits and systems. In SPICE this is accomplished with equations in the time, frequency or any other analogous domain (temperature, mechanical, physical) [5,6]. Common functions are usually grouped into subcircuits. More complex functions can then be created by graphically combining subcircuits. This is the method most familiar to circuit designers. While subcircuit or macro-modeling is much maligned by AHDL proponents it has several major advantages.

Conceptually, a subcircuit should be thought of as a primitive element. As stated by one AHDL supplier, the practice of subcircuit macro-modeling is identical to the creation of models by AHDL constructs [16]. Once completed the subcircuit can be looked upon as a new primitive element. "The VHDL standard, now IEEE-1076, gives engineers a means of describing the behavior of electronic systems which is totally removed from the circuitry that implements the behavior." [13] For analog designers, this represents a problem, not a solution. Circuit designers, by their nature, are not programmers. They rarely have the time or the skills to code behavioral models let alone deal with the complexities required by most AHDLs.

For the vast majority of elements and applications, however, the ability to graphically combine predefined blocks, as is the case with macro-modeling, is dramatically superior to AHDL modeling in the areas of ease of development, maintenance, portability and convergence. Analog model development is not the same as VHDL model development especially when behavioral modeling is involved. It is often the case that only the designer knows the required accuracy and fully understands the design. Therefore, the model complexity must be in the hands of the designer. Development of models by third party groups poses problems such as documentation of the model and required background to develop the model. [9] Therefore, the model development must remain with the designer and by necessity remain simple.

Traditionally, new element models could only be added to SPICE by modifying the source code or producing a subcircuit representation. In the case where the subcircuit approach is insufficient the availability of C code subroutines allows new elements to be added without having to modify the SPICE source. The C code modeling approach, which is relatively new, allows users to describe arbitrarily complex functions and behavior and greatly extend SPICE's capabilities in board-level and systems modeling. It has virtually all the capabilities offered by current AHDLs. C code modeling supports arbitrary user defined node types, as well as, access to various analog (time and temperature, differential equations) and digital (event scheduling) quantities. Adding C subroutines to SPICE is equivalent in complexity to current AHDL/VHDL model development. While not directly portable, the interface described in [3] is public domain and once adopted will make the subroutines widely accepted. In addition, use of OLE2 and dynamic link libraries (DLLs) could enhance C code model portability and REMOVE the need to recompile portions of the executable program each time a code model is added or changed.

Based on currently available simulation and modeling features it is obvious that SPICE is as viable as any proposed AHDL product for an AHDL standard.

Assertions Made By AHDL Proponents

SPICE only allows structural descriptions and circuits can only be built by connecting different elements. - False SPICE allows behavioral, conditional and structural descriptions in the main circuit and in macromodels. This, in turn, allows virtually any electronic device to be modeled as well as quantities used in other technologies (mechanical, thermal, physical etc.)

SPICE has no AHDL constructs, only basic electrical elements. SPICE simulations are limited to essentially transistor level simulation - False It is obvious from Table 2 that today's SPICE programs have most of the behavioral, conditional and mathematical features available in AHDL languages in addition to basic SPICE primitive elements. Since circuits can contain any level of modeling, they may also be simulated at any level (system, behavioral, or transistor)

ADDING SPICE BASED ANALOG EXTENSIONS TO VHDL

The following tables review some of the modeling and development issues related to incorporating analog HDL extensions into VHDL versus combining VHDL and SPICE.

Table 3 - Adding AHDL Constructs to VHDL

Disadvantages

- Requires analog circuit designers to become programmers.
- Changes the graphical block approach used in macromodeling, currently used by most analog designers, to a programming style environment. Current AHDLs take users away from the WYSIWYG design entry environment.
- Requires learning about the simulator's flow and structure (data/node types) as well as the inner workings of modeling constructs (selection of sample points for piecewise linearization, coding of partial derivatives, etc.) in order to deal with various modeling issues (convergence). While this is also true for advances in SPICE modeling (addition of C code subroutines), the vast majority of modeling can be done and has been done with the behavioral/macromodeling combination.
- Requires users to program convergence into the model rather than using blocks whose convergence is known.
- Introduces a huge learning curve to analog model developers with increased reliance on technical support for programming guidance.
- Need to add various analog concepts (timeless domains, non-binary states, differential equations, nonelectrical domains [2]) to VHDL. Requires a new class of analog signals and analog behavioral operators.
- Adding basic primitive elements to VHDL/Verilog requires experience and effort commensurate with those needed to add a model to SPICE.
- Requires coordination in an industry known for not cooperating. The popular digital simulator of the year has invariably changed whereas SPICE has remained the dominant standard for over 20 years. [4]
- Requires SPICE programs to read, parse, and execute a completely new language while providing little, if any, increased functionality.
- Requires AHDL to have SPICE model and analysis compatibility.

Advantages

- New products will be needed.
- Allows VHDL users to add analog functionality using a familiar environment.
- Provides a potential path for analog synthesis.
- Allow exchange of VHDL models with analog functions.

Table 4 - AHDL or SPICE; Which should be combined With VHDL?

Disadvantages

- For AHDL, a standard syntax must be chosen and maintained. There is currently no clear leader and no public domain precedents. Use of an alternate AHDL requires reinventing the wheel since SPICE compatibility must be maintained. [21]
- AHDL requires higher costs in terms of on-site modeling support and development
- For SPICE, additions must be made to behavioral and mixed mode capabilities. The syntax must be standardized on but public domain precedents for a standard syntax exist. [3]
- For SPICE, speed is not as big an issue as it once was. Faster computers have eased the performance issues allowing larger circuits to run quickly even at the transistor level. [8] Interactive simulation interfaces and behavioral modeling has further eased this issue.

Advantages To Using SPICE

- SPICE is already in existence and is already a standard. This makes it a natural starting point for AHDL development and addition to VHDL. The top three SPICE based simulators, Pspice, Is-Spice, and HSPICE, account for over 30,000 purchased seats (not discounted or given away to universities). Additionally, tens of thousands of circuit designers have copies of student and demo versions of various SPICE products. Most students learn and use SPICE in school, alleviating the training burden.
- Already strong support in the hardware vendor community.
- Models already being interchanged: IBIS Models, op-amp models, RF models, power models
- Interfaces exist to PCB, EMC, and signal integrity tools.
- SPICE already satisfies the vast majority of AHDL requirements.
- With the addition of C code models or minimal additions to behavioral/macro-modeling features virtually every AHDL feature is matched by SPICE.
- C code model development uses a language and tools which are already in widespread use.
- SPICE only needs increased functionality rather than an entire new product set.
- SPICE models can be easily modified by the user and vendors don't encrypt their models.
- SPICE simulates multiple domains: time, frequency, Laplace, and Z, as well as Mixed Technologies, (Non-Electrical: Thermal, Neural network, Fuzzy Logic, Physical, Mechanical [5], Hydraulic/Vacuum [6]). Advanced SPICE programs even have pre-made models for many elements in these fields [5].
- Public domain and independently funded: Berkeley SPICE, Georgia Institute of Technology (XSPICE), etc.. The EDA industry tends to migrate to open standards, of which SPICE is one of the oldest and strongest. [2]
- Operating systems and platforms: SPICE runs on virtually all computers.
- Several SPICE versions have already been linked to VHDL.
- New public domain simulation algorithms are continuously being added to SPICE. [19]
- Superior interactive interface as compared to most AHDL based simulators.
- Superior price, performance, and availability on all platforms especially PCs and Windows.

Affordable and Cost Effective Tools

SPICE is available on virtually every computer platform and has been on PCs since 1984. Its price is usually an order of magnitude or less than other AHDL tools and several freeware type versions are in widespread use. “It is apparent that in the long run UNIX may not survive as a significant player in the CAE market”. [4] Therefore, the selected AHDL must be available on PCs and Windows for widespread acceptance to occur.

With the downsizing of the aerospace industry many experienced analog designers have been forced into the consulting arena. This trend of specialization and downsizing is likely to continue. The importance of low cost tools can not be overstated. Needless to say, the 5 and 6 figure prices (not including maintenance) on most AHDL tools is hardly within the reach of most small and medium sized engineering firms.

Vendor Independent Support For and From SPICE

Like VHDL, SPICE's development has been driven by independent funding and academic research, both keys to fostering widespread use. Other AHDLs are mainly proprietary and will face serious resistance for a variety of price, performance, and compatibility reasons. In any case, due to the pervasiveness of SPICE, they will be forced to support SPICE syntax, hence, greatly diminishing their usefulness.

Like any AHDL, connections to digital simulation must be made to support mixed designs. SPICE vendors are tackling the mixed signal simulation problem like other AHDL vendors with one advantage. Most of the academic work combining various algorithms and techniques with analog simulation is being done with SPICE. [19] (For example, Cider (physically based device level simulator), Ecstasy (circuit optimization), both from U.C. Berkeley and XSPICE (12-state event driven digital simulator) from Georgia Institute of Technology) In the area of EMC simulation virtually all commercial 2-D/3D finite element simulators produce SPICE compatible output [17]. In the PCB area, many layout packages are geared to produced transmission line and parasitic data in the form of SPICE netlists. And lastly, SPICE simulators are beginning to support VHDL models.

Compatibility

In a recent survey readers listed compatibility with existing systems, design data exchange, and exchange of vendor data as number one priorities in selecting a CAD system. [18] SPICE's compliance in these key areas is well established. Today there are no less than 16 different hardware vendors supplying over 1600 SPICE models that will run on virtually any SPICE based simulator on any platform. No other analog simulation language is providing this level of compatibility. In fact, the majority of the SPICE vendors are now incorporating XSPICE extensions in their tools including: Deutsch Research (Dr. SPICE), Cad-Migos (SPICE-It!), Microcode Engineering (CircuitMaker), Interactive Image Technologies (Electronic Workbench), Visionics, and Intusoft (IsSpice4).

Modeling Ability

SPICE contains support for modeling on many levels as discussed above. The inefficiency for the majority of subcircuit based models is greatly overstated in competitive summaries, especially when today's high performance platforms are considered. With the addition of behavior modeling features and other AHDL-like constructs SPICE provides a comparable wealth of modeling features and performance.

Design Documentation

The incorporation of design documentation features could push the language complexity to astronomically unmanageable heights. This seems something much better suited to the schematic data base. In any case, both current AHDLs and SPICE programs are lacking in this area.

SUMMARY

It is clear that current SPICE simulators provide virtually all of the features that AHDL based simulators purport to have. In addition, SPICE has achieved a level of standardization that is the envy of its opponents and the measure by which other languages are judged. It seems obvious that SPICE should be the starting point for the addition of analog extensions to VHDL or for adding VHDL as an extension to the mixed mode capabilities now being incorporated into SPICE.

As it has been over the past 20 years, SPICE will continue to dominate. Not only because it is a proven, well designed, and well implemented technology, but because it grows and incorporates the best of it challengers, including current AHDL products. VHDL usage will continue to grow as well, with added features like SPICE compatibility and analog extensions. Undoubtedly both approaches will continue to advance.

[1] "SPICE3 Version 3f User's Manual", B. Johnson, T. Quarles, A.R. Newton, D. O. Pederson, A. Sangiovanni-Vincentelli, Dept. of EE and CS, U.C. Berkeley, October 1992

[2] “HDL-A Extends VHDL into the Analogue Domain”, Electronic Engineering, May 1994

[3] “Code-Level Modeling in XSPICE”, F. Cox, W. Kuhn, J. Murray, S. Tynor, Proceedings of the 1992 Intl. Syp. on Circuits and Systems, May, 1992, Georgia Institute of Technology

[4] “Directions To Watch In Design Technology”, A. Richard Newton, Department of EE&C U.C. Berkeley, 29th ACM/IEEE DAC June/92

[5] SPICE Applications Handbook, 2nd Edition, C.E. Hymowitz, L.G. Meares, Intusoft, 1990

[6] “Vacuum Pump Models for SPICE”, Technology Sources Application Note, Technology Sources Newmarket Suffolk, U.K. Tel. 638-561-460, 1991??

[7] “Development of an Analog Hardware Description Language”, C. Kurker, J. Paulos, B. Cohen, and E. Cooley, IEEE Custom Integrated Circuits conference, 1990

[8] “Examining Some SPICE Programs”, C. E. Hymowitz, M. Archambault, EE Times, June 6, 1994,

[9] “Behavioral Analog Circuit Models for Multiple Simulation Environments”, Larry Moser, IBM Corp., IEEE Custom Integrated Circuits conference, 1990

[10] “ESACAP - A PC Implemented General Purpose Circuit Simulator”, P. Stangerup, IEEE Circuits and Devices, Vol. 4, No. 4, July, 1988

[11] “Behavioral Modeling Techniques in the Analysis of a Bipolar Integrated Circuit”, Peter Denyer, Analog and Mixed Signal Conference, 1991

[12] Mast AHDL Brochure, Analogy

[13] “Progress in Developing An Analog HDL”, Stephan Ohr, Analog and Mixed Signal Conference, 1991

[14] “Analog Verilog Quasi Mixed Signal simulation Model for A/D Converter”, Michael K. Mayes, Analog and Mixed Signal Conference, 1991

[15] “Understanding Mixed-Mode Simulators”, Mark Chadwick, Analog and Mixed Signal Conference, 1991

[16] “An Introduction to Behavioral Modeling”, I. Getreau and D. Teegarden, Analog and Mixed Signal Conference, 1992

[17] “Simulation Breaks into New Fields Modeling Connectors”, N. Campbell, M. Penberth, C. Hymowitz, ECN EDA Supplement, June 1993

[18] “Trends in Use and Purchase of EDA”, ED Magazine, 6/94, based on ED 1994 EDA Market Study, presented at DAC 1994

[19] “Current State of the Art in Non-Traditional Analog Circuit Simulation”, P. Allen, IEEE Custom Integrated Circuits conference, 1992

[20] “Beyond SPICE with Saber and MAST”, H. Mantooth, M. Vlach, IEEE Custom Integrated Circuits conference, 1992

[21] “HDL Momentum Grows”, Lindsey Vereen, ASIC & EDA magazine, June 1994