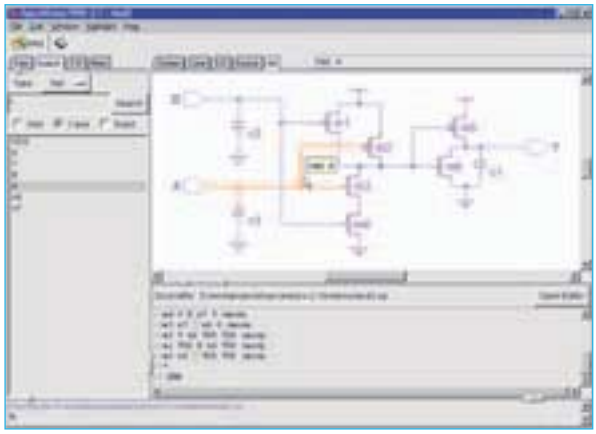
Visualization of SPICE files helps to optimize SoCs

by Gerhard Angst, Concept Engineering

SPICE text files are not easy to read and understand. SpiceVision PRO loads the SPICE file and automatically generates a circuit diagram. This familiar view helps to understand the text files and allows all members of the design team to look at the same circuit presentation.



The SpiceVision GUI shows the gate as well as its SPICE list

SpiceVision PRO (SVP) converts SPICE text files to circuit diagrams and uses cone windows ("intelligent magnifying glasses") to get a better view of selected components. Additional user programming via TCL allows for customer specific extensions like consistency checks or locating heavy loads. Selecting parts of the design leads to reduced complexity and results in faster simulation of sub-circuits. SpiceVision PRO has been designed as an interactive tool to visualize SPICE descriptions, to analyze SPICE models, to speed up debugging and help to speed up the optimization process. SpiceVision loads the SPICE file, automatically generates a circuit diagram including all connections described, using predefined images of the various SPICE elements like R, L, C or transistors. Now the designer can see a familiar view of a circuit diagram, or a cut-out view of selected elements.

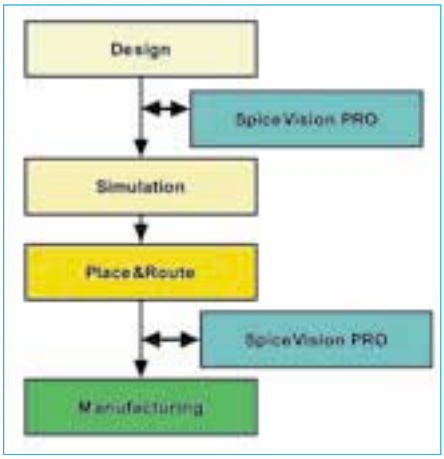
SpiceVision interprets SPICE text files generated by any design tool in all the important SPICE dialects, including SPICE2, SPICE3, HSPICE, PSPICE and CDL. Even more specialized versions are supported indirectly, since most tools generating these dialects provide an option to export data in one of the standard forms. SpiceVision flawlessly integrates with existing design and characterization flows, independent of manufacturer. Before loading a new file, the user can select the appropriate dialect or let the tool decide which to use.

The conversion from text to circuit diagram normally uses predefined symbols for the different components, like resistors, capacitors, transistors, current and voltage sources. In addition links are possible to external symbol libraries and definitions. SpiceVision's hierarchy browser can start from the top level of the selected SPICE file and move down the circuit hierarchy, displaying different levels in different windows, until the basic components like capacitors and transistors become visible.

An important capability of the tool is its search function, finding the areas of interest and displaying them on the screen. After initialization, SVP will show the search results and store them in a separate result list. This list is the basis for display. The user can select which parts of the results list he wants to see, independent of the hierarchy levels involved. Object attributes, for example input or output port definitions or types of power supplies, can be modified via pop-up menus to further increase readability. These definitions are then used by SVP to generate circuit diagrams that are even easier to read and understand.

The tool will show all the components in the part list, but simplification is possible. Components in parallel, such as capacitors, can be replaced by a single component representing the sum. If there are too many capacitors in a part of the file, the screen becomes cluttered. SVP provides a feature that selects all of them and can make them invisible. Special areas or component groups of a design, for example nets or instantiations, can be selected and saved as bookmarks for later use. Context sensitive menus allow for fast access to the relevant data.

One problem still needs to be solved: how can we work with a file that generates a display of, for example, 200,000 transistors? How can we have a high magnification display but still see modules of the design that are scattered over the circuit? The solution is the cone window, an "intelligent magnifying glass" that displays the selected fragments in a new view, either as a flat or as a hierarchical design. The same feature can be used to display critical paths that have been calculated by other tools; again only the interesting parts of the design are seen, which can be followed easily. The cone window in SpiceVision PRO provides new applications including selected simulation and communication. As soon as the designer has selected all the components that are important for him this can be stored as separate file. This new file can be printed or electronically shared between all designers with access to SpiceVision PRO and discussed on the phone with all the data restricted to the main theme. It is also possible to simulate only the selected area.



Chip-design flow using SpiceVision PRO

Having defined and saved the SPICE data of a fragment the designer can execute a quick partial simulation, either locally or remotely. Normally even small modifications in the design require the full design to be simulated. SpiceVision PRO stores any selected SPICE data or circuit fragment for use in an external simulator. One practical example: in a design of 200,000 elements only a fragment of that 200 is altered by a change is relevant. By simulating just the design fragment, simulation time is reduced by a factor of 10 to 100. Translated into hours this is the difference between spending the 10 hours of a working day waiting for a result, or just an hour (even the lunch break). Partial simulation allows more simulations to take place yet still completes the design faster.

As soon as a design has been finished, it has to be implemented in the semiconductor technology chosen: the components have to be placed and connected. The design tools used for place and route can create new SPICE files from the actual implementation, taking into account effects caused by wire length on the chip, coupling between neighbor wires etc. As these new SPICE text files are normally generated automatically, the original SPICE list of the design and the same list after place and route are extremely difficult to compare. The new list after place and route is a lot longer and hardly any place and route tools can generate a circuit diagram of the enlarged SPICE list.

Place and route tools generate new critical paths, for example those that influence and limit maximum clock frequency or impact device yield in production, but do not also generate an easily readable circuit diagram. Critical paths are extremely difficult to follow in a text list but SpiceVision can be used to create schematics, with the "intelligent magnifying glass" feature. Many new components are generated by place and route tools, but again the ability to hide or sum capacitors or other components achieves a better view of the final circuit function. In many cases SVP is the only option to get a view of the final circuit in silicon. At this state of the design a selection and display of circuit fragments connected via multiple hierarchies is even more important. The same applies to "selective simulation" as the increased files lead to even longer simulation cycles.

No engineer likes to do documentation. In many cases the stress levels are high until tape-out and, additionally, the next design is waiting. As a result the documentation is minimized. SpiceVision PRO supports this phase of the design cycle. SPICE lists that define actual design stages can be used to document designs as circuit diagrams or circuit books where the data is automatically cut into the pages required. This print and document feature is not just important at the end of a design. As the actual SPICE data are always available, any complete or any partial view can be frozen or printed at any time. This is even more attractive when combined with a revision control software package such as that available from ClioSoft. At any time, such as before and after place and route, the designer can take a snapshot to freeze the design status. This can be shared automatically within the whole design team. The snapshots provide the capability of returning to a working circuit diagram with all its full and partial views, and use this as a new starting point. As all the print files of all versions are available on the network, communication is very much simplified.

The re-use of proven parts of a design either internally or from third parties is increasing as IP is seen as the only solution to achieve more throughput with an existing design team and a limited development time budget. With SpiceVision PRO designers are able to select certain parts of the design and save these as new files, cookie-cutting the new IP out of the complete design. A full definition of the IP as a SPICE text file, together with the circuit diagram, is available immediately for filing and transfer.

SpiceVision is designed to visualize 2000 elements. In SpiceVision PRO a more flexible database imposes no limitations and additionally an application program interface allows access to the internal database. Using the TCL programming language the user can read and write to and from the database, for example to take data and generate user specific reports as additional tool functions. One known simulation problem is "floating gates" which lead to error messages. In large designs this will cost time, as the file has to be corrected by the relevant designer before a new simulation run can start. Using the SpiceVision PRO API the user can find all the floating gates and correct the design before the simulation group takes over. ■

TASKING™
TriCore
Embedded intelligence
When it comes to embedded software development TASKING tools deliver:
- Highly optimizing ISO C++/EC++ and ISO C compilers, using Altium's new VIPER-technology VX
- Supports all TriCore architecture steps and derivatives including: TC11IB, TC1765, TC1775, TC1910, TC1912, TC1920 and the new TC1130
- Easy to use, fully integrated EDE for project file management, tools & target configuration and a language sensitive editor
- Kernel aware CrossView Pro debugger (OSEK) with instruction set simulator and On-Chip-Debug support

Altium Sales and Support:
Free Call: 0800 0 258486 (from Germany)
00800 776 776 77 (German)
00800 776 776 55 (French)
00800 776 776 44 (English)
Free Fax: 0800 776 776 00
Email: tasking.sales.de@altium.com
Web: www.altium.com/tasking
Quote Value Code 1400
1609
All trademarked names are the property of the respective owners
Altium Making Electronics Design Easier™