# Taking the scary out of Spice

*How can you convert Spice text files to circuit diagrams and still bypass the problems of working with hundreds of thousands of transistors? Gerhard Angst of Concept Engineering explains.*

SPICE text files, generated by many eda tools, are the standard way of describing the lowest level of behaviour of digital and analogue circuits both during circuit design and during verification after place and route. The description details the analogue behaviour of the transistors and other components that make up digital circuits and gate functions. Unfortunately SPICE files are not easy to read let alone understand.

During an SOC design SPICE text files are generated to describe, for example, the inner working of a gate. The designer can use transistor geometries and capacitive loads as a basis to judge maximum clock frequency behaviour or power consumption. But a text file of SPICE values is very difficult to read and to interpret, as all the relationships and connections have to be put in context from a long list. Fig. 1 shows a partial SPICE list at the bottom of the screen.

SpiceVision PRO (SVP) has been designed as an interactive tool to visualize SPICE descriptions, to analyse SPICE models, to speed up debugging and the optimisation process. The tool loads the SPICE file and automatically generates a circuit diagram using predefined images of the various SPICE elements, (such as R, L, C or transistors). This means that all members of a design team – wherever located – can look at the same circuit representation and discuss the details normally hidden somewhere in the SPICE text.

## Dealing with dialects

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.

The conversion from text to circuit diagram normally uses pre-defined symbols for different circuit components. The 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 initialisation, 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 increase readability further. These definitions are then used by SVP to generate circuit diagrams that are even easier to read and understand.

The tool will show all of 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.

One problem still needs to be solved: how can we work with 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?
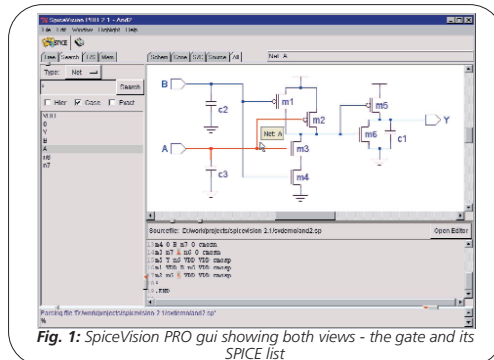


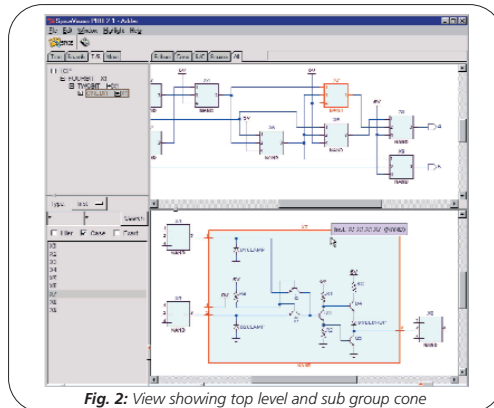*Fig. 1: SpiceVision PRO gui showing both views - the gate and its SPICE list*



*Fig. 2: View showing top level and sub group cone*

The answer is to use a so-called cone window, an "intelligent magnifying glass" that displays the selected fragments in a new view, either as a flat or as a hierarchical design (see fig. 2). 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 and can be followed easily.

The cone window provides new applications including selected simulation and communication. As soon as the designer has selected all of the components that are important for him this information can be stored as separate file. It is also possible to simulate only the selected area.

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. SVP stores any selected SPICE data or circuit fragment for use in an external simulator.

For example, take a design with 200,000 elements that has only 200 elements altered following a design change. 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.

## Place and route

As soon as a design is complete, it has to be implemented in a semiconductor technology. The design tools used for place and route can create new SPICE files from the actual implementation and take into account the effects of wire length on the chip or coupling between neighbour wires for example.

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 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 its core window. 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.

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.

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.

While SpiceVision is designed to visualize 2,000 elements, SVP has a more flexible database that imposes no limitations. It also has an api that allows access to the internal database. Using tool command language (tcl) 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". These lead to error messages, which in large designs costs time as the file has to be corrected by the relevant designer before a new simulation run can start. However, using the SpiceVision PRO api the user can find all of the "floating gates" and correct the design before the simulation group takes over.

Converting SPICE text files to circuit diagrams and using cone windows to get a better view of selected components marks an industry first. Additional user programming via tcl enables customer specific extensions such as consistency checks or locating heavy loads. Selecting parts of the design leads to reduced complexity and results in faster simulation of sub-circuits.

**Concept Engineering**                    *Write in number 392*