To search page

1973

SPICE developed (University of California, Berkeley, U.S.A.)

~ Integrated Circuit ~

Circuit simulation refers to simulating analog operation of an electronic circuit with a computer, and an EDA tool for executing circuit simulation is called a circuit simulator.

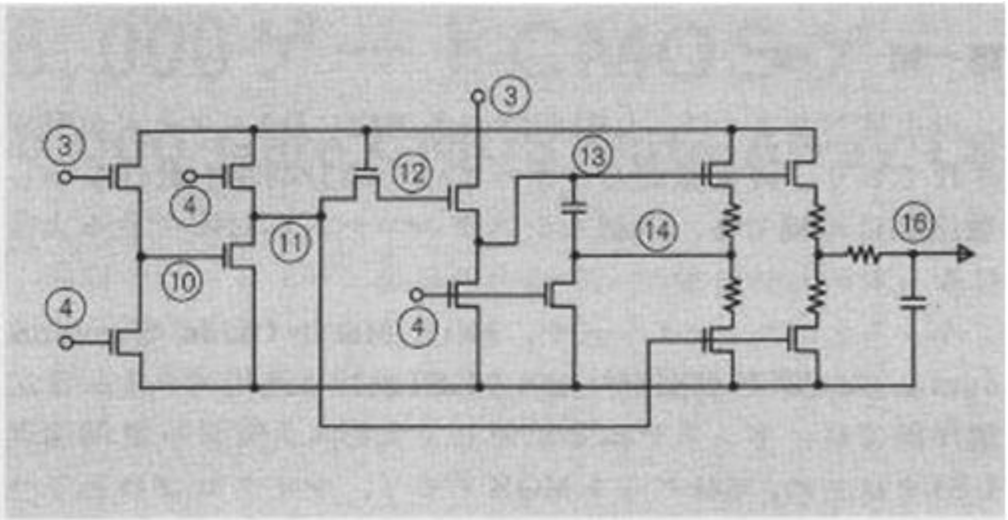
Sprouting of circuit simulation was in the 1960s. Initially, TAP (IBM in 1962) and CIRCUS (BOEING in 1965) were developed using an algorithm called state variable analysis method which formed the voltage and current of each terminal of a branch (minimum unit of circuit expression) as an equation. Also, ECAP (1965 IBM) and CANCER (1971 UCB) were developed which adopted a nodal analysis method which formed the unknown voltages of nodes (signal lines) as an equation.

SPICE (Simulation Program with Integrated Circuit Emphasis) is a circuit simulator developed by the University of California at Berkeley in 1973, and it was made by improving CANCER. The electronic circuit to be simulated is a combination of passive elements (such as resistors and capacitors), active elements (diodes, transistors, etc.), transmission lines and various power supplies. The analysis methods include transient analysis, DC analysis, small signal AC analysis, noise analysis and so on. Originally developed for the purpose of designing electronic circuits for use in integrated circuits, SPICE has become an indispensable tool in the design and verification of printed circuit boards incorporating high-speed logic LSIs.

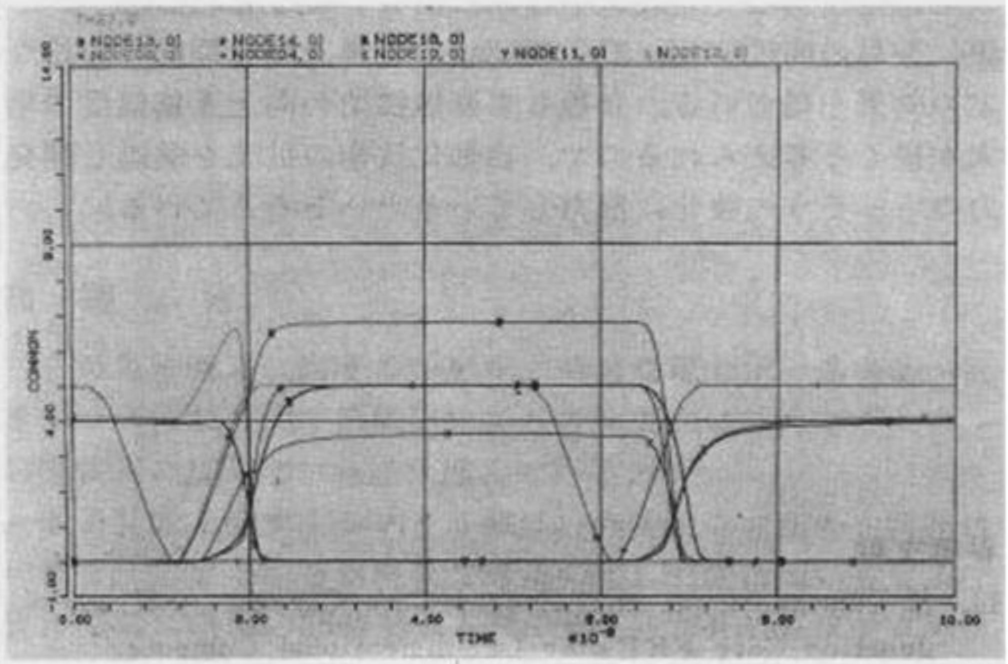
SPICE has SPICE2 (SPICE2 G6 in 1981) which was written in Fortran language, and a new SPICE3 (SPICE3 A7 in 1986) which was rewritten in C language. Since these source codes are open to the public, many commercial software (HSPICE, PSPICE etc.) and free software (Ngspice etc.) have been derived. Commercially available products include a link with a schematic editor, a man- machine interface in the pre-post parts, enhancement of device libraries, linking with a digital simulator, and optimization functions are expanded.

The main method of circuit simulator is a combination of formulation method of circuit equation called "modified nodal method" and "device model" using analysis formula. In the modified nodal method, a matrix equation based on "Kirchhoff's current-voltage law" is set with the node voltage in the circuit and the current flowing in the voltage source element as variables. Device models are also used to calculate the current flowing in each transistor. The device model is based on measured electrical characteristics (current voltage characteristics and capacitance voltage characteristics) and uses an analytical expression developed based on theoretical or experimental considerations.

The circuit equations thus created are nonlinear simultaneous ordinary differential equations, which are discretized and linearized and analyzed by a direct method matrix solving method. It is known that this method is a solution with extremely small theoretical error in numerical calculation. Any kind of device and circuit configuration can be applied generically. On the other hand, the applicable circuit scale is limited to about 20,000 to 30,000 elements. Also, since analytical expressions representing device models are generally complicated and highly nonlinear, there is a case where circuit simulation does not converge unless an appropriate initial value is given.



(a)Circuit to be simulated



(b)Simulation result

Example of circuit simulation
By inputting a circuit diagram(a) to be simulated, an output wave forms are given as shown in (b). This can be proceeded on TSS terminal in an Interactive manner

Fig. Circuit simulation example around 1980