In a Nutshell: How SPICE Works

By Dr. Colin Warwick, Agilent Technologies, Inc.

Intended audience: This article won't help software engineers who have to *implement* circuit simulators: they invest time with the classic textbooks[1]. But I believe that textbooks are over kill for SPICE *users* and that a shorter, simpler explanation is a better investment of time, hence this article.

Let's start simply with a onetime step (i.e. DC) solution of a circuit that consists of two unknown node voltages, V_1, V_2, a ground node V_0, three known ohmic conductances, $G_{xy} = 1/R_{xy}$ (where $I_{xy} = G_{xy} (V_y - V_x)$ and x and y are the node indices), and three known current sources (Fig. 1).

You can solve a circuit using either Kirchhoff's current law or voltage law or both. These laws are named after the German physicist Gustav Robert Kirchhoff (1824–87)[2]. SPICE is a modified nodal solver and uses the current law: the sum of the currents into each node is zero. We'll talk about what the 'modified' bit means in a future 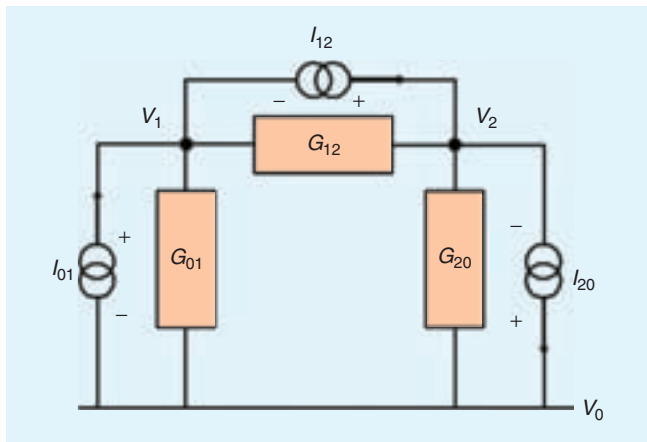article on 'super nodes.' We'll also postpone a discussion about when Kirchhoff's laws break down for a future article (hint: Faraday's law trumps Kirchhoff's law).

The nodes are joined by branches, so the other ingredients are the *branch constitutive equations* of the components that join them; for example V = IR if it's an ohmic resistor, V = L dI/dt for an inductor, etc. In this simple example, we have three simultaneous equations, one each from node 0, 1, and 2:

$$G_{01}(V_1 - V_0) - G_{20}(V_0 - V_2) + I_{20} - I_{01} = 0$$
$$G_{12}(V_2 - V_1) - G_{01}(V_1 - V_0) + I_{01} - I_{12} = 0$$
$$G_{20}(V_0 - V_2) - G_{12}(V_2 - V_1) + I_{12} - I_{20} = 0$$

… with three unknowns, V_0, V_1, and V_2.

The same equations can be rearranged into matrix form; in this case the augmented (or indefinite) node conductance matrix relates the voltage and current vectors:

$$\begin{bmatrix} (G_{01} + G_{20}) & -G_{01} & -G_{20} \\ -G_{01} & (G_{12} + G_{01}) & -G_{12} \\ -G_{20} & -G_{12} & (G_{20} + G_{12}) \end{bmatrix} \begin{bmatrix} V_0 \\ V_1 \\ V_2 \end{bmatrix}$$
$$= \begin{bmatrix} I_{20} - I_{01} \\ I_{01} - I_{12} \\ I_{12} - I_{20} \end{bmatrix}$$

Note the 'pattern of four' that each conductance (e.g. G_{01} highlighted below) impresses into the conductance matrix (Table 1).

In SPICE parlance, making this 'pattern of four' impression is called 'stamping the matrix.' Conveniently, this 'stamping' generalizes for any number of nodes and two terminal components. In a future article, we'll show how a small modification to this

Fig. 1.

[1] For example "Computer Methods for Circuit Analysis and Design" by Kishore Singhal and Jiri Vlach
[2] The *ch* in Kir*ch*hoff is pronounced like the *ch* in the Scots' word lo *ch*.

TABLE 1.

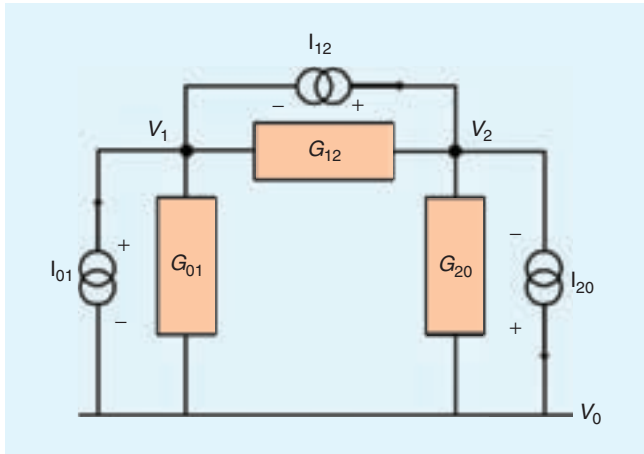|	Column x	Column y
Row x	$+G_{xy}$	$-G_{xy}$
Row y	$-G_{xy}$	$+G_{xy}$

Fig. 2.

method allows us to 'stamp in' a three- or four-terminal component like a voltage-controlled current source (and hence deal with transistors).

Pairs of nodes with no physical branch element connecting them have a conductance of 0. In practical circuits, the average number of non-zero conductance components per node is only $\sim3-4$, whereas the number of nodes can be quite large: hundreds, thousands, even millions. Thus, practical circuits have sparse, not full, conductance matrices: SPICE can make use of the efficiency of a sparse matrix solver.

An n + 1 by n + 1 augmented matrix has rank *n*. (In our simple example n = 2.) The normal (or definite) conductance matrix can be obtained simply by selecting a datum node (e.g. define node 0 to be 0V) and deleting its row and column. The matrix equation $GV = I$ can then be solved for the column vector of voltages $([V_1; V_2])$ by matrix inversion: $V = G^{-1}I$ (Fig. 2).

Before we go on to see how SPICE deals with time-stepping ('transient analysis'), with reactive, four-terminal, and non-linear elements, and with shorts and voltage sources, let's do a quick self-test:

Which of these components *cannot* be modeled using a pure nodal circuit solver?
- Capacitor
- Non-linear resistor where $I = k_1V + k_2V^2 + k_3V^3$
- Open circuit
- Voltage source
- BSIM4 MOSFET model
- Voltage-controlled current source

To see the answer we need to explore beyond conductances and current source, and look at other analyses and components.

Voltage Source and Other Infinite Conductances

It turns out that pure nodal analysis can't handle voltages sources because they have infinite conductance. So the answer to our pop quiz is *voltage source*.

SPICE avoids the infinite conductance problem by modified nodal analysis (MNA). Nodes joined by infinite conductances are considered "super-nodes" whose constituent node voltages V_x and V_y move up and down in lock step. When SPICE creates a super-node the two individual KCL equations are eliminated and replaced by one KCL equation that sums all the currents into both nodes (into the blue dashed oval in Fig. 3) plus one
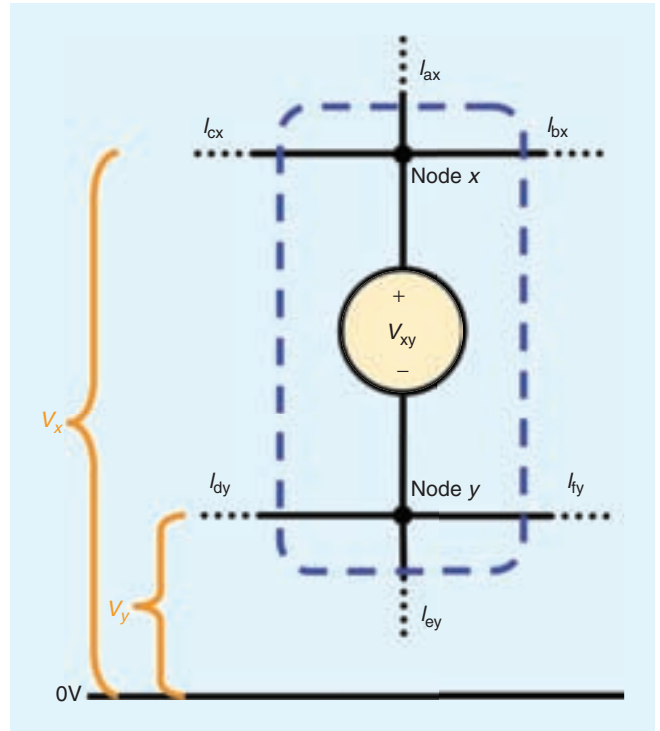

Fig. 3.

internal constitutive equation, namely $V_x = V_y + V_{xy}$, where V_{xy} is the known value of the voltage source. Shorts and current sinks (i.e. the input ports of current controlled sources) can be treated similarly.

By the way, MNA isn't the only solution to the infinite conductance problem. For example, Hachtel et al. proposed a sparse tableau method of where both branch currents and node voltages are considered unknown, both KCL and KVL equations are formulated. You then have to pick out a set of linearly independent equations.

Three and Four Terminal Devices

How about more than two terminals? It turns out that when the current source "stamped" its pattern-of-four into the conductance matrix, it was a special case of a more general four terminal "stamp." The general case is the *trans*-conductance (Fig. 4).

The "pattern of four" is simply displaced off-diagonal in the conductance matrix (Table 2).
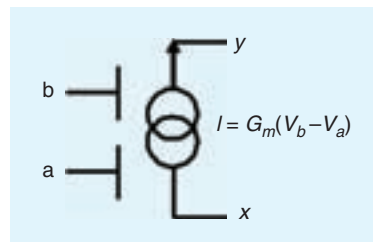

Fig. 4.

TABLE 2.				
	Column a	**Column b**	**Column x**	**Column y**
Row a			$+G_m$	$-G_m$
Row b			$-G_m$	$+G_m$
Row x				
Row y				

TABLE 3. BRANCH CONSTITUTIVE EQUATIONS FOR VARIOUS COMPONENTS AND VARIOUS ANALYSES IN SPICE

Element	Mode	Branch constitutive equation
Resistor $G = 1/R$	All	$I = GV$
Capacitor C	DC	$V = ?, I = 0$
	Transient	$I = C\dfrac{dV}{dt}$
	AC	$I = j\omega CV$
Inductor L	DC	$V = 0, I = ?$
	Transient	$I = \dfrac{1}{L}\int V dt$
	AC	$I = \dfrac{V}{j\omega L}$
Voltage Source	All	$V = V, I = ?$
Current Source	All	$V = ?, I = I_s$
Voltage Controlled Current Source	All	$V_{out} = ?, I_{out} = g_m V_{in}$
Voltage Controlled Voltage Source	All	$I_{out} = ?, V_{out} = A V_{in}$
Current Controlled Current Source	All	$V_{out} = ?, I_{out} = A I_{in}$
Current Controlled Voltage Source	All	$I_{out} = ?, V_{out} = I_{in} R_m$
Mutual Inductor M	DC	$V_1 = V_2 = 0, I = ?$
	Transient	$I_1 = \dfrac{1}{L_{11}}\int V_1 dt + \dfrac{1}{M}\int V_2 dt$
		$I_2 = \dfrac{1}{L_{22}}\int V_2 dt + \dfrac{1}{M}\int V_1 dt$
	AC	$I_1 = \dfrac{V_1}{j\omega L_{11}} + \dfrac{V_2}{j\omega M}$
		$I_2 = \dfrac{V_2}{j\omega L_{22}} + \dfrac{V_1}{j\omega M}$

Non-linear, Time-domain ("Transient") Analysis

For transient analysis, components are represented in differential or integral form. See the table below. SPICE performs a numerical ordinary differential equation (ODE) solution. Non-linear elements are solved by an iterative method (e.g. Newton-Raphson) at each time step. An initial guess at the node voltages is created (usually all zeros). The slope and intercept of the tangent to the actual I-V curve is used to calculate a linear approximation of the non-linear element. The linear approximation (a conductance and a current source) is inserted into the conductance matrix as a proxy for the real device. The solution of the linear proxy yields a better guess at the voltage vector. A new set of conductance/current source proxies is calculated using tangents at the new voltages. This is repeated until — hopefully! — convergence is reached for that time step.

SPICE uses variable time steps. The initial voltage vector guess for each time step is the converged solution of the previous step. If the time step causes accuracy problems, SPICE backtracks by disregarding that calculation and taking a small step from the previous time point.

AC Analysis

For DC analysis, reactive elements — inductors and capacitors — are treated as shorts and opens, respectively. For AC analysis, complex admittance is used in place of real conductance. For example, admittance of a capacitor and inductor are $j\omega C$ and $1/j\omega L$, respectively. Again see Table 3 at left.

SPICE has its limitations. If there is a changing magnetic flux through a given mesh, Faraday's Law of magnetic induction $\nabla \times E = -B$. affects the branch equations and breaks KVL by making the electric field non-conservative and the voltage undefined. At that point you need to switch to an EM solver, which is a topic for another day.

Colin Warwick is signal integrity product manager at the EEsof division of Agilent Technologies. Prior to joining Agilent, Colin was with Royal Signals and Radar Establishment in Malvern, England, then Bell Labs in Holmdel, NJ, and most recently at The MathWorks in Natick, MA. He completed his bachelor, masters, and doctorate degrees at the University of Oxford, England. He has published over 50 technical articles and holds thirteen patents. **EMC**