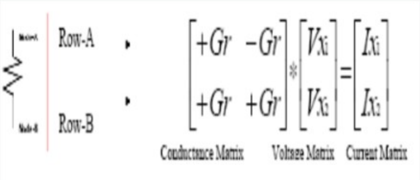
3. Spice Program Basic Structure

The system equations are created through matrices, discussed next.

<u>The system matrices</u>

To save time, Spice uses a shortcut to develop the system matrix. This shortcut of matrices is created through the "matrix construction by inspection" technique. This technique builds the system matrix with the help of a predefined element template. The template describes the position of the matrix for the 'conductance and current' values of a particular device.

A resistor template structure is shown as an example, to present the matrix structure for "conductance matrix x voltage matrix = current matrix":



(Click on image to enlarge.)

Conductance matrix:

The conductance array is the circuit's linear relationship between voltage and current for every element. Nonlinear elements such as diodes, transistors and charge-storage elements are represented in the conductance array by their equivalent linear circuit.

Voltage matrix:

The voltage matrix is presented as the node voltage of each element in the circuit. When performing analysis, Spice determines the node voltage and current with the aid of the Kirchoff's laws applied to the circuit.

Current matrix:

During Spice simulation, the branch current is defined by the current source settings.

<u>The elements of the system</u>

Each version of Spice defines its basic elements which represent the electronic circuit and system. Below are the elements of PSPICE, Device type and Letter:

- Bipolar transistor Q
- Capacitor C
- Voltage-controlled voltage source Voltage-controlled current source E
- Voltage-controlled voltage source Voltage-controlled current source G
- Current-controlled current source Current-controlled voltage source F
- Current-Controlled switch W
- Current-controlled current source Current-controlled voltage source H
- Digital input (N device) N
- Digital output (O Device) O
- Digital primitive summary U
- Stimulus devices* U STIM
- Diode D
- GaAsFET B
- Independent current source & stimulus I
- Independent voltage source & stimulus V

- Inductor L
- Coupling K
- IGBT Z
- Junction FET J
- MOSFET M
- Resistor R
- Subcircuit instantiation X
- Transmission line T
  Transmission line coupling K
- Voltage-Controlled switch S

<u>Non-linear elements</u>
These are active elements such as diodes, transistors, transistor arrays, and other charge storage elements. These elements complicate analysis because their voltage-current relationships are not always linear.

Such nonlinear and charge-storage elements must be reduced to simplified equivalent circuits before being entered in the system equations. These simplifications are required because the matrices accept linear I-V relationships only; the Spice matrix is always G x V = I. Any nonlinear functions may be expressed as series of linear approximation, and defined through a piecewise-linear curve.

<u>Linear analyses</u>
When a circuit contains linear elements, Spice uses the CAD technique of Gaussian elimination to solve the matrix.

<u>Nonlinear analyses</u>
When a circuit includes nonlinear elements, Spice uses an additional technique called the Newton-Raphson algorithm, which applies iteration to find a solution, and for which any node of the circuit is solved for its dc voltage level.

The iterations are defined by the Spice parameter of

"ITL = x"

which causes the simulation to run-up to a pre-defined limit and stop. If the dc bias in not found before the ITL limit, a non-convergence situation occurs, and an error message is generated. Upon the completion of the iteration process, the dc voltage of a node is defined by convergence process in Spice with the Newton-Raphson algorithm.