

11/16/2006

New Shortcut Methods for State Space Circuit Analysis – Updated M-Files.

Using state space analysis, the circuit dc, ac, and transient response can all be obtained from the same initial analysis. Hence there is an economy of effort realized that makes it worthwhile to learn state space analysis. However, conventional state space methods require an inordinate amount of circuit analysis algebra. These Word files show a shortcut method to create the circuit state space arrays, after which the analysis can proceed directly to dc, ac, and transient responses with minimal effort. The circuit transfer function can also be a part of the solution using Leverrier's Algorithm.

This method considerably reduces the amount of algebra when compared to conventional techniques as presently taught in most electrical engineering curriculums. This original method is termed the Dc Superposition (DS) method.

The author believes in teaching by example. There are many example circuits, from a component count of 3 to 22, used to demonstrate the method.

Spice-derived software packages are the de facto industry circuit analysis tools. The primary advantage of these simulation tools is that virtually no circuit analysis algebra is required. Then the question might be asked, what purpose can be served by presenting analysis techniques that minimize but do not eliminate equation writing? The pedagogical aspects aside, Spice software has a serious built-in algorithmic error that affects all Spice ac worst-case analyses. (For an example, see the file magdrvrev.doc.) Another Spice limitation is that it allows only 400 samples for Monte Carlo Analyses (MCA). From statistical confidence intervals, this is clearly insufficient.

Hence the rationale for presenting these state space methods is to provide:

- Easily programmable algorithms for correct dc and ac worst case analysis; and to allow as many MCA samples as computer memory and speed will permit.
- Simple circuit analysis methods for the working engineer if Spice is not available due to network downtime, queuing due to limited site licenses, or simply has not been purchased.
- With suitable node-list to matrix conversion software, these methods can be used as a basis for a "home grown" Spice program. \*

MATLAB output plots have been imported to these Word97 documents for illustration purposes. Some loss of quality from plots printed from MATLAB will be seen. The M-files with corresponding file names (\*.m) contain many explanatory comments, and are separate from the \*.doc files.

MATLAB Version 5.3 (R11) was used.

See dsintro.pdf next.

\* The following textbook by the author has done just that. This is a method of circuit analysis using Spice-like node lists wherein large circuits can easily be analyzed. It eliminates the cumbersome and error-prone task of writing nodal equations to create the A1 and B2 arrays. Maximum number of nodes = 89; max number of inputs = 10; max number of output nodes = no limit, up to circuit nodes: The title is "Node List Tolerance Analysis – Enhancing SPICE Capabilities with Mathcad", CRC Press, 2006. The Mathcad programs can be easily transcribed into MATLAB M-Files.

The author can be contacted via email at [bobbyd@ieee.org](mailto:bobbyd@ieee.org)