Abstracts

Frequency Domain Analysis of RF and Microwave Circuits Using SPICE

C.E. Smith. "Frequency Domain Analysis of RF and Microwave Circuits Using SPICE." 1994 Transactions on Microwave Theory and Techniques 42.10 (Oct. 1994 [T-MTT]): 1904-1909.

The SPICE circuit analysis program has limited math capability and, in general, cannot be used for RF and microwave simulation because complex arithmetic is required to compute S-parameters from node voltages. This paper presents two models of a directional coupler that can be used to obtain node voltages proportional to incident, reflected, and transmitted signals. These models are based on a previously developed multiple coupled transmission line (MTL) SPICE model. From SPICE computed node voltages, S-parameters are computed rising the math capability of the PSPICE post processor, PROBE, as an example for a low-pass filter consisting of transmission line sections. The results of this example are compared with another high-frequency circuit analysis program, TOUCHSTONE. The difference between the results of these two programs in magnitude was less than 0.005 and in phase was a few tenths of a degree. By using these couplers to simulate a network analyzer, RF and microwave analysis can be made with SPICE, which can be a cost-effective and readily available computational tool for educators and practicing engineers.

❑ Return to main document.