Application Note AN-11

SPICE Models for PSPL Coaxial and SMD Components

James R. Andrews, Ph.D., IEEE Fellow & PSPL Founder

A general statement can be made that no electrical component behaves exactly the way our beginning electrical engineering textbooks would have us believe. Oftentimes, very complex circuit models are required to fully account for all of the parasitic behavior associated with what would seem to be a simple component, such as an inductor.

PSPL's coaxial and SMD components are all designed making extensive use of computer modeling tools, such as SPICE. Individual electrical components, such as inductors, capacitors, resistors, and active devices used in PSPL products are exhaustively characterized using both time domain and frequency domain measurements. SPICE .subckt models have been created by PSPL for all critical components used in our products. When designing new products, our engineers use SPICE to compute circuit performance and interactively optimize the performance of each product. We have created very exotic SPICE models for all of our products, which match very well with actual measured time and frequency domain responses.

PSPL has received numerous requests from customers for our SPICE models of our products. We have been unwilling in the past to disclose these SPICE models because they contained extremely proprietary information on how to build our products. We have now decided to create a set of totally new SPICE models that will not disclose any proprietary circuit details. These new models are phenomological-based and take the view that the component being modeled is instead a "Black Box". All information for modeling a component is taken from the publicly disclosed data found on the PSPL specification sheet. For example, for a bias tee, this would include basic capacitance, inductance, dc resistance and the time domain step response, along with the frequency domain "S" parameters of S_{21} and S_{11}. The resultant models do not model every bit of the fine structure found on the S parameters, for example, but they do model the major effects.

The actual SPICE models in net list form are available to be downloaded from our Web site: www.picosecond.com. They are located in an ASCII file called "PSPLspice.txt". As new SPICE models are created for additional products, they will be added to this file.

Example

```
.SUBCKT  BT5542   1   2   3
*  subcircuit for pspl model 5542 bias tee
*  0.22uF, 16V, 1.5mH, 100ma, 5.6 ohm
*  10 kHz to > 50 GHz  (-3dB bandwidth)
*  node 1 is AC port            node 2 is AC+DC port
*  node 3 is DC port            not valid  f > 50 GHz
C1     1  6    0.22UF           L6     9  10   50PH
Rdc    2  4    5.6              R7     10  2   20
L1     4  3    1.5MH            L7     10  2   0.1NH
R1     4  3    2K               R8     2   11  75
R2     3  5    1K               L8     11  12  0.68NH
C2     5  0    0.22UF           C8     12  0   10FF
C3     3  0    1NF              R9     2   13  500
R4     2  7    2K               L9     13  14  33NH
L4     7  8    10UH             C9     14  0   0.75FF
C4     8  0    4PF              R10    2   15  750
R5     6  9    3                L10    15  16  22NH
L5     6  9    0.39NH           C10    16  0   12FF
C5     6  9    560PF            .ENDS
```

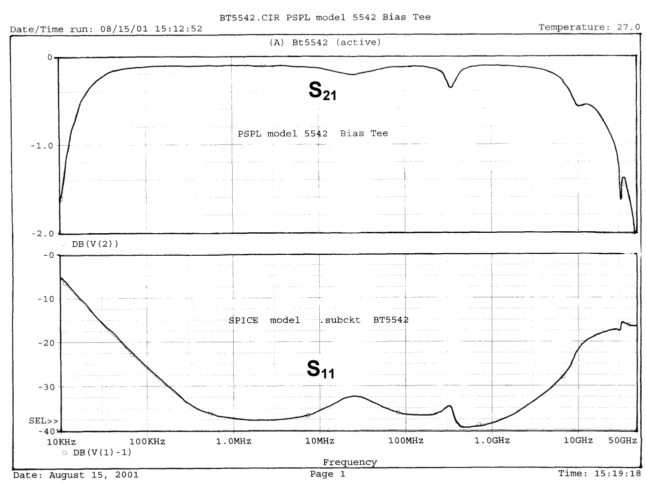

Figure 1: Computed S_{21} & S_{11} from .subckt BT5542