Create S-Parameter Subcircuits for Microwave and RF Applications

by John S. Gerig Wideband Associates

The article "Obtain S-Parameter Data from Probe" (April 1994 issue of *The Design Center Source),* described useful subcircuits which permit both the transmission (S_{21} and S_{12}) and the reflection (S_{11} and S_{22}) parameters of a given circuit to be calculated and displayed using Probe. In microwave and RF design work, a PSpice subcircuit whose S-parameters are defined as *tables* is also useful; the data corresponds to that provided by many manufacturers for microwave transistors and other microwave devices. [See, for instance, the Hewlett-Packard *Communications Components Designer's Catalog*.]

The **S-Parameter Subcircuit**

A 2-port S-parameter subcircuit can be easily implemented as shown within the dashed lines of Figure 21. The voltage-controlled voltage sources—E11, E21, E12, and E22—use the frequency-response table feature provided with PSpice Analog Behavioral Modeling. Figure 21 also shows the setup for measuring S_{11} and S_{21} for this subcircuit. Node 3 is typically grounded, and nodes 1 and 2 correspond to ports 1 and 2.
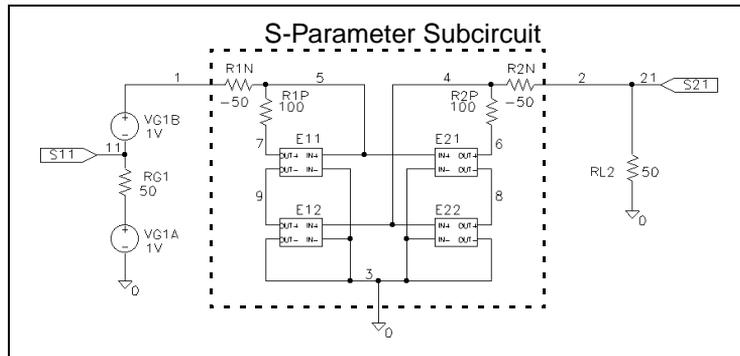
Figure 21 *S-parameter subcircuit schematic also showing setup for evaluating S_{11} and S_{21}. Z_0=50 ohms is assumed.*

As noted in the April article (repeated in the theory sidebar at left), the S_{ij} coefficients are the dimensionless ratios of scattered wave amplitudes, *b1* and *b2*, to incident wave amplitudes, *a1* and *a2*; i.e., $S_{21}=b2/a1$. The wave amplitudes are usually normalized so that their squared magnitudes measure power. In the present application, however, it is convenient to normalize *a1* or *a2* to 1 volt; the corresponding S-parameters then become the voltages at certain nodes.

The voltage generator at the left side of Figure 21 is set up as an open-circuit voltage of 2 in series with a generator impedance of Z_0 ohms. This delivers a voltage wave *a1=1* volt to a matched or Z_0 ohm load. By implementing the 2-volt generator as two 1-volt generators—VG1A and VG1B in series with the generator resistance RG1 as shown—we achieve the particularly simple result that the voltage at node 11 is the reflected or scattered wave *b1*. Since *a1* equals one, this is S_{11} itself. This can be verified by inspection. If the generator is terminated at node 1, then the voltage at node 11 is +1 for an open circuit, 0 for a matched load, and -1 for a short circuit. This is in agreement with equation (8) of the April article (see side bar).

Similarly, the voltage at node 21 to the right of Figure 21 is simply $S_{21} = b2/a1$. Thus, for example, the Probe expressions VDB(21) and VP(21) will display the dB amplitude and phase of S_{21}.

Note *In the case of S-parameters S_{12} and S_{22} associated with a2=1, the test generators and termination of Figure 21 must be transposed. See the example schematic in Figure on page 65; the nodes are labelled 22 and 12, accordingly.*

E11 and E21, when connected to matched terminations, deliver the scattered waves *b1* and *b2* resulting from the incident wave *a1* modified by appropriate response tables. Thus the required control input to these generators is the voltage *2*a1*—the total voltage generated by VG1A and VG1B in series. By introducing the negative resistance, *R1N=-Z_0*, between nodes 1 and 5, we can force node 5 to this voltage since the voltage drop across R1N due to any input current exactly cancels the drop across RG1. When E11 and E12 are turned off, the further addition of *R1P=2*Z_0* then offsets R1N and produces the required Z_0 input impedance at port 1.

E12 and E22 similarly generate the appropriate scattered waves for an incident wave *a2* at port 2.

Converting Manufacturer's Data

It is completely feasible, though tedious, to prepare table-based subcircuits by manually editing S-parameter data published by the manufacturer. However, the manufacturer's data almost always presents S-parameters in terms of magnitude and wrapped phase, whereas the frequency-response tables used in PSpice Analog Behavioral Modeling (version 6.0 and earlier) require decibel levels and unwrapped phase in degrees. (PSpice version 6.1 can optionally accept magnitude in raw form and unwrapped phase in radians.)

Phase wrapping refers to reducing the phase data to principal values in the range -180 to +180 degrees. This creates a problem for Analog Behavioral Models which linearly interpolate the phase data. For example, if the unwrapped phase data steps from -175 to -195 degrees, the wrapped version would step from -175 to +165 degrees. In the unwrapped case, the interpolated midpoint is a realistic -185 degrees; in the wrapped case, however, the nonphysical interpolated value is -5 degrees.

The S2P2LIB1 Conversion Program

It is much easier to convert the manufacturer's S-parameter data to PSpice-compatible form using a program that can interpret S2P files. Figure 22 shows the logic to do so; this program—S2P2LIB1—is written in QuickBasic.

```
'S2P2LIB1-- converts S2P tables to
+ PSpice-compatible frequency-response tables.
'Written in QuickBASIC (v4.5). See 'HelpMessage'
+ SUB for additional comments.
DECLARE SUB ParseDataLine ()
DECLARE SUB HelpMessage ()
DIM D(16), M(50, 9)
COMMON SHARED D(), M(), L$
False = 0: True = NOT False
DataLine = 0: EODat = False
C$ = COMMAND$
'C$ = "10236N.S2P"  'Decomment and edit in uncompiled version.
IF UCASE$(RIGHT$(C$, 4) = ".S2P" THEN  'Plausible command$ found.
 ufn$ = LEFT$(C$, LEN(C$) - 4)
ELSE
 CALL HelpMessage: SYSTEM
END IF
OPEN C$ FOR INPUT AS #1
OPEN ufn$ + ".LIB" FOR OUTPUT AS #2
PRINT #2, ".SUBCKT " + ufn$ + " 1 2 3; Port1 Port2 Common"
PRINT #2, "*Subcircuit generated by S2P2LIB1.EXE on " + DATE$

WHILE NOT (EOF(1) OR EODat)
LINE INPUT #1, L$
L$ = LTRIM$(L$)         'Get rid of any leading spaces.
IF LEFT$(L$, 1) = "!" THEN PRINT #2, "*" + RIGHT$(L$ + " ", LEN(L$))
IF LEFT$(L$, 1) = "#" THEN HDR$ = UCASE$(L$): PRINT #2, "*" + L$
IF VAL(L$) > 0 THEN
 InTheData = True
 ParseDataLine
 DataLine = DataLine + 1
 FOR k = 1 TO 9: M(DataLine, k) = D(k): NEXT
END IF
IF VAL(L$) = 0 AND InTheData = True THEN EODat = True
WEND
CLOSE #1
PRINT #2, " ": PRINT #2, "R1N 1 5 -50": PRINT #2, "R1P 5 7 100"
PRINT #2, "R2N 2 4 -50": PRINT #2, "R2P 4 6 100"
S$(1) = "*S11 FREQ DB PHASE": S$(2) = "*S21 FREQ DB PHASE"
S$(3) = "*S12 FREQ DB PHASE": S$(4) = "*S22 FREQ DB PHASE"
E$(1) = "E11 7 9 FREQ {V(5,3)}=": E$(2) = "E21 6 8 FREQ {V(5,3)}="
E$(3) = "E12 9 3 FREQ {V(4,3)}=": E$(4) = "E22 8 3 FREQ {V(4,3)}="
G$ = "_+_(###.##gHz_,+###.##_, +###.##)"   'Format for gHz data
M$ = "_+_(#####.##MHz_,+###.##_, +###.##)" 'Format for MHz data
IF INSTR(HDR$, "MHZ") THEN P$ = M$ ELSE P$ = G$

FOR P = 1 TO 4        'Build S11, S21, S12, and S22 blocks in sequence.
PRINT #2, " ": PRINT #2, S$(P): PRINT #2, E$(P)
```

Figure 22   *QuickBasic logic for S2P2LIB1 program.
(Continued on the next page)*

```
   Offset = 0: PrevPh = 0 'Clear variables used to unwrap phase.
   FOR F = 1 TO DataLine  'Successive frequency values.
   Ph = M(F, 2 * P + 1)   'Current phase data.
   IF ABS(Ph-PrevPh)>180 THEN Offset=Offset-360*SGN(Ph-PrevPh)
   PrevPh = Ph
   UWP = Ph + Offset                'UnWrapped Phase.
   DB=20*LOG(M(F,2*P))/LOG(10): 'Convert magnitude to DB values.
   PRINT #2, USING P$; M(F, 1); DB; UWP
   PRINT ".";  'Something to look at.
   NEXT F: NEXT P
   PRINT #2, " ": PRINT #2, ".ENDS"
   PRINT "Finished. Result saved in file " ufn$ ".LIB"
   CLOSE #2
   END

   SUB HelpMessage
    PRINT"WBA 7/94. This utility converts an"
    PRINT"S-parameter file *.S2P having"
    PRINT"a magnitude and angle (degrees) format"
    PRINT"into a PSPICE subcircuit"
    PRINT"file *.LIB which uses the Analog Behavioral"
    PRINT"Model option."
    PRINT"The source file should be in the current directory."
    PRINT"A 50 ohm reference impedance is assumed."
    PRINT"gHz frequency units are assumed unless the"
    PRINT""*.S2P files contains"
    PRINT"a header record beginning with '#' and followed by 'MHz'"
    PRINT" "
    PRINT"USAGE: S2P2LIB1 *.S2P "
   END SUB

   SUB ParseDataLine
   k=1: D$=""
   FOR CH=1 TO LEN(L$)+1
   CH$=MID$(L$+ " ", CH, 1)
   IF CH$ <> " " THEN D$ = D$ + CH$: SP = 0
   IF CH$=""AND SP=0 THEN D(k)=VAL(D$): k=k+1:SP=1:D$=""
   NEXT

   IF k<>10 THEN
    PRINT"Error: didn't find 9 numbers parsing the "
    PRINT"following data line:"
    PRINTL$
    PRINT"Check *.S2P file": END
   END IF
   END SUB
```

Figure 23 *QuickBasic logic for S2P2LIB1 program continued from prior page.*

S2P2LIB1 assumes that the S-parameter data is in magnitude and phase format, and that the reference impedance is 50 ohms. (Although other formats are allowed in S2P files, they are less frequently encountered.) Default frequency units are GHz, though MHz is also accepted if the header line ('#' in column one) contains "MHZ." S2P comment lines ('!' in column one) are converted to PSpice comment lines ('*' in column one). Noise data is ignored.

Example: AC Analysis of the 10236N S-Parameter Model

Figure 24 lists an abridged version of the 10236N subcircuit which implements the S-parameter model for an Avantek/Hewlett-Packard ATF10236 transistor. (The frequency-response tables have been truncated to conserve space.) Instead of manually setting up the
S-parameter tables, the full-table version of this subcircuit can be automatically created from its corresponding S2P file (see Figure 24) using the S2P2LIB1 conversion program described above. Simply type

```
S2P2LIB1 10236N.S2P
```

at the command line prompt. This produces the unabridged 10236N subcircuit model which implements the circuitry shown in Figure 21 in both the forward and reversed directions. The subcircuit model is saved to the "10236N.LIB" file which can then be used in a PSpice simulation.

```
!  ATF-10236 S AND NOISE PARAMETERS
!  Vds=2V Id=25mA
!  LAST UPDATED 06-28-89

!FREQ  S11      S21      S12      S22
!GHZ MAG   ANG   MAG  ANG   MAG  ANG   MAG   ANG

0.5  .97  -20   5.68 162   .023 76    .47   -11
1.0  .93  -41   5.58 143   .050 71    .45   -23
2.0  .77  -81   4.76 107   .086 51    .36   -38
3.0  .59  -114  4.06 80    .120 35    .30   -51
4.0  .48  -148  3.51 52    .149 18    .23   -67
5.0  .46  166   3.03 26    .172 3     .10   -67
6.0  .53  125   2.65 1     .189 -14   .09   48
7.0  .62  96    2.22 -20   .191 -28   .24   55
8.0  .71  73    1.75 -39   .189 -41   .37   51
9.0  .75  54    1.47 -55   .184 -46   .46   42
10.0 .78  39    1.28 -72   .180 -59   .51   34
11.0 .82  26    1.04 -86   .179 -71   .54   26
12.0 .84  12    .95  -101  .177 -82   .54   17

!FREQ Fopt GAMMA OPT  RN/Zo
!GHZ  dB  MAG  ANG   -

0.5  0.4  .94  16    0.78
1.0  0.5  .87  36    0.63
2.0  0.6  .73  74    0.33
4.0  0.8  .45  148   0.15
6.0  1.0  .42  -137  0.12
8.0  1.3  .49  -80   0.45
12.0 1.6  .65  -20   1.16
```

Figure 24   *S2P file used to derive the 10236N S-parameter subcircuit model shown in Figure 23.*

Schematic Setup:

Editor's Note: To benefit users running the Design Center with Schematics, MicroSim added this section to the author's original manuscript.)

1   *Within the Symbol Editor*, create a 10236N symbol similar to that shown in Figure 25. This symbol will represent the circuitry shown within the dashed lines of Figure 25.
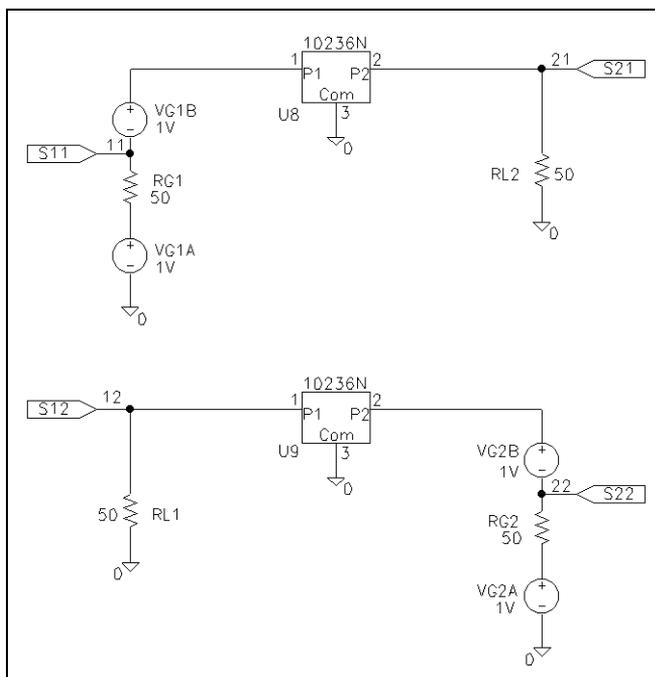
Figure 25   *Schematic setups for evaluating the S-parameters of the 10236N.*

2   Define symbol attributes (Part/Attributes), including the MODEL attribute to associate the 10236N subcircuit model with the symbol:

MODEL = 10236N
TEMPLATE = X^@REFDES %P1 %P2 %COM @MODEL

Note   The TEMPLATE value is a one-line string; do not insert newline characters when entering this value.

3   Save the symbol to a new Symbol Library file using Part/ Save Changes and File/Save As. Be sure to select Yes when asked to add this to the list of configured files for use with Schematics. (Alternatively, you can save the symbol to an existing library file using Part/Save to Library.)

4   Within the Schematic Editor, configure the model file, "10236N.LIB," for use with your example schematic (Analysis/Library and Include Files). Use the Add Library

button if you wish to have it available to the current schematic only; otherwise use Add Library* for global availability.

5 Place VSRC (or VAC voltage source), GLOBAL (global port), R (resistor), AGND (analog ground), and 10236N symbols to create the test schematic shown in Figure 25 on page -65. Define the voltage sources so that AC magnitude equals 1; define the resistors equal to 50 ohms.

6 Enable and set up the AC sweep analysis under Analysis/ Setup with the following characteristics:

AC Sweep Type	Linear
Total Pts	100
Start Freq.	0.5E9
End Freq.	12E9

Note *If you expect to run a series of simulations which test different S-parameter subcircuits, you can create one generic S2P symbol rather than a custom symbol for each model (as was demonstrated for the 10236N). To do so, create a MODEL attribute for the S2P symbol and leave it undefined (blank). Then, from within the Schematic Editor, you can change the model reference for the S2P part instance in your schematic. Simply select the S2P instance, select Edit/Model, select Change Model Reference in the dialog, then type in the name of the S-parameter subcircuit to be used in the next simulation.*

7 Circuit File Setup: Create a circuit file as shown in Figure 26. The "10236N.LIB" file is referenced using a .LIB statement (.INC will also work). Subcircuit declarations, XQF and XQR, can then reference the 10236N subcircuit model. The .AC statement defines the parameters for the AC sweep analysis as described earlier for the schematic setup.

```
SPARTST1.CIR S-Parameter Demo 10236N.LIB
*JG 7/25/94 This file illustrates a simple method for analyzing
* the S-parameters of a subcircuit. The subcircuit has been
* created from a standard S2P
* S-parameter file using the utility S2P2LIB1.EXE.

.OPT NOMOD RELTOL=.0002
.AC LIN 100 .5E9 12E9

.LIB 10236N.LIB  ;Subcircuit 10236N for ATF10236 transistor

*Separating the conventional 2 volt EMF generator into two 1 volt
*generators (VG1A and VG1B) connected by the RG1=Z0 as shown,
* produces a voltage at node 11 equal to S11. S21 appears at node 21.
VG1A 101 0 AC 1
RG1 101 11 50
VG1B 1 11 AC 1
XQF 1 21 0 10236N
RL2 21 0 50

*The following makes S22 appear at node 22, and S12 at node 12.
VG2A 201 0 AC 1
RG2 201 22 50
VG2B 2 22 AC 1
XQR 12 2 0 10236N
RL1 12 0 50

.PROBE V(11) V(12) V(21) V(22)
.END
```

Figure 26   *Circuit file to simulate the 10236N subcircuit and analyze its S-parameters.*

After having simulated the circuits with PSpice, Probe can be used to display the various S-parameters in any desired form. Sample results are shown in Figure 27.
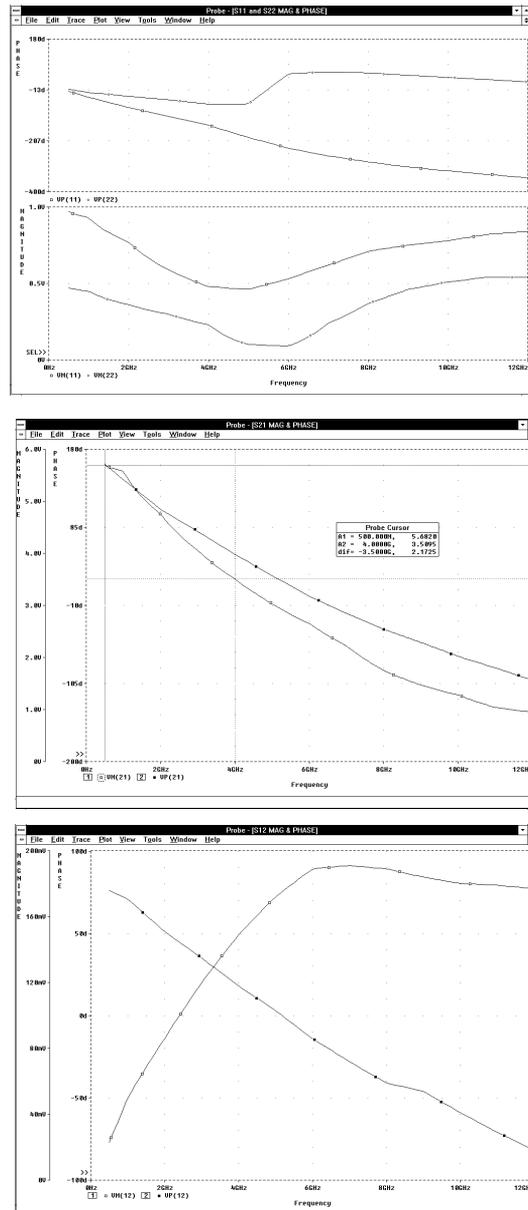
Figure 27 *Simulation results showing S-parameter magnitude and phase.*

If the AC sweep analysis is modified to reproduce the frequency steps occurring in the original S2P file, and appropriate print commands are added for VM(11), VP(11), etc. (using VPRINT1

pseudocomponents in the schematic or .PRINT statements in the circuit file), the resulting PSpice output file (".out") should numerically reproduce the original data.

Transient Analysis Considerations

Data for an S-parameter model is usually measured in the frequency domain, and normally represents only the linear or small-signal behavior of the device under specified DC operating conditions. Effects in a nonlinear transistor or diode are not modeled. The frequency-response table option in Analog Behavioral Modeling allows transient analysis, but the cautions discussed in the PSpice user's guide should be reviewed.

To Download Files from the BBS

The files referenced in this article are available on the MicroSim bulletin board in one self-extracting file named "sparam.exe." This file contains the conversion program, S2P2LIB1 as a DOS executable (".exe") and as uncompiled QuickBasic source (".bas"). This file also contains the unabridged version of "10236N.LIB," the S2P file—"10236N.S2P," and the corresponding symbol file—"S2P.SLB" (which can be imported into Schematics from within the Symbol Editor using Part/Import and Part/Save to Library).

To download the self-extracting file, select [T]ech Support from the main menu, [6] File Transfer, [1] Download User Requested Files, and indicate that you wish to download "sparam.exe." The BBS number in the U.S. is (714) 830-1550 (14.4k-1200, N-8-1).

Biography: John S. Gerig is a consultant specializing in microwave circuit design, frequency filters, and frequency synthesizers. He owns the company, Wideband Associates and may be reached by phone at (703) 391-*5619 or FAX at (703) 391-0318.*

OrCAD

| HOME | OrCAD LAYOUT | OrCAD CAPTURE/CIS | PSPICE | CONTRIBUTE | DISCUSS |

Using PSpice | Application Notes | Downloads | Models | Movies & Demos | Books & Articles | FAQs

PSPICE

Engineers present tools and techniques to help you use PSpice more effectively.

Using PSpice
Basic product answers and multi-media overviews.
Go there>>

Application Notes
Application engineers have collected a variety of technical "how-to's" to help you optimize PSpice.
Go there>>

Downloads
Make sure your version of PSpice is up to speed with these downloads.
Go there>>

Models
See if the model you're looking for is listed here.
Go there>>

Movies & Demos
A highly informative way to learn about Windows-based PCB design.
Go there>>

Books & Articles
Build your own reference library.
Go there>>

FAQs
Quick answers to popular questions.
Go there>>

Application Note

Obtain S-Parameter Data from the Probe Window
Source: The Design Center Source newsletter, April 1994
Revised by: Bill Janewicz, April 1999
Edited by: Coby Bassett, May 1999

RF and microwave circuit designers frequently express the input and output characteristics of circuits in terms of scattering parameters (s-parameters). By adding two subcircuits, s-parameter data can be printed to the output file or displayed in the Probe window. The method presented here is qualified for two-port networks, but the concept can be extended for n-port networks.

Theory
S-parameters measure the ratio of the incident and reflected signal. The incident signals are defined as $a1$ and $a2$. The reflected signals are defined as $b1$ and $b2$. The incident and reflected signals are related to voltages and currents at ports 1 and 2 by

$$\tag{1}$$

$$b1 = \frac{V1 - Z_0 I1}{2\sqrt{Z}} \tag{2}$$

$$a2 = \frac{V2 + Z_0 I2}{2\sqrt{Z}} \tag{3}$$

$$b2 = \frac{V2 - Z_0 I2}{2\sqrt{Z}} \tag{4}$$

The scattered waves are related to the incident waves by the following set of linear equations:

$$b1 = S_{11}a1 + S_{12}a2 \tag{5}$$

$$b2 = S_{21}a1 + S_{22}a2 \tag{6}$$

or, in matrix form as

$$\begin{bmatrix} b1 \\ b2 \end{bmatrix} = \begin{bmatrix} S_{11} & S_{12} \\ S_{21} & S_{22} \end{bmatrix} \begin{bmatrix} a1 \\ a2 \end{bmatrix} \tag{7}$$

The Sij coefficients are dimensionless ratios; for most applications, the characteristic impedance of the system, $Z0$, is 50 ohms. $S11$ is the input reflection ratio and is defined as the ratio of the input port reflected wave to the input incident wave. If the incident wave at the output, $a2$, is set to zero, then the equations reduce to $b1 = S11a1$ and $b2 = S21a1$. Using the defining equations, this reduces to

$$S_{11} = \frac{b1}{a1} = \frac{V1 - Z_0 I1}{V1 + Z_0 I1} = \frac{Z - Z_0}{Z + Z_0} = 2\left(\frac{Z}{Z + Z_0}\right) - 1 \tag{8}$$

where $V1/I1$ is the input impedance Z. Similarly, $S21$ is the forward transmission ratio and is defined as the ratio $b2/a1$. If the input and output load impedances of the circuit are the same, then $S21$ is the voltage measured at the output multiplied by 2. If the incident wave at the input is set to zero, then the equations reduce to $b1 = S12a2$ and $b2 = S22a2$.

Defining the Subschematics

To make all of the necessary measurements, two subschematics are required. These subschematics are shown in **Figure 1** and **Figure 4** and can be created by drawing the schematic in Capture and creating a hierarchical part for each. For the purpose of this article, these custom parts are named XMITS and REFLECTS, respectively. See Creating and Editing Parts in the Capture User's Guide for reference to creating hierarchical parts.

The XMITS circuit shown in **Figure 1** is used to measure the forward, $S21$, and reverse, $S12$, transmission coefficients. Since the output load matches the input load, the transmission coefficients are the output voltage multiplied by 2. The E device, E1, has a gain of 2. The port labeled CKT is used to connect to the external circuit. The port, S_TR, is a hidden pin (see **Figure 2**); if it is left unconnected in a schematic, Capture will generate a unique net. Alternatively, a specific net can be named for the connection by editing the "PSpiceDefaultNet" attribute value (see **Figure 3**) for the XMITS part instance; this way, the S_TR pin will have a known label ($s21$ in this case) when analyzing simulation results within the Probe window.
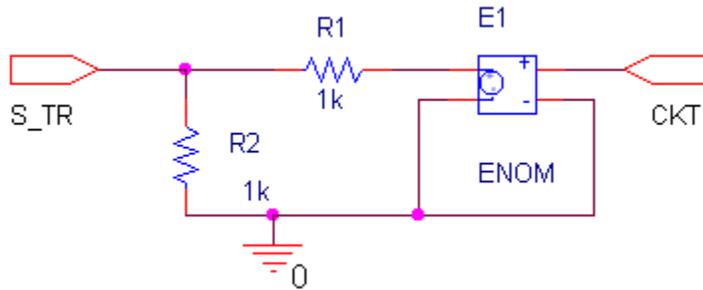
Fig 1. Transmission coefficients measurement subschematic

Fig 2. Capture Part for the XMITS subschematic
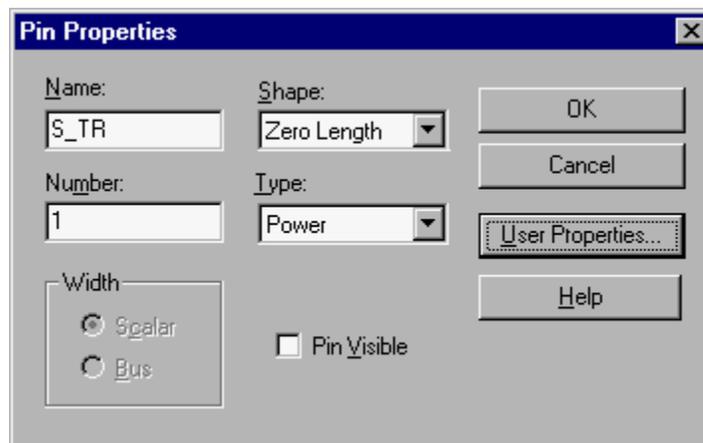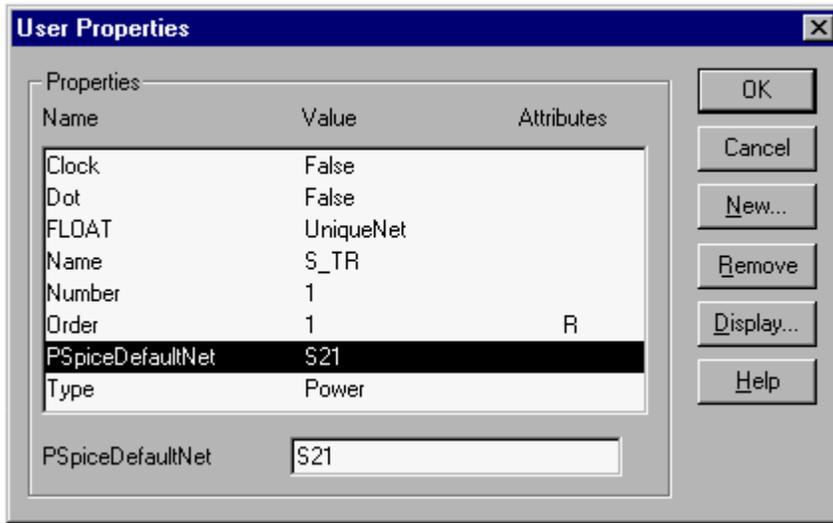
Fig 3. Pin Properties for the XMITS part showing hidden S_TR pin

The REFLECTS circuit shown in **Figure 4** is used to measure the input, *S11*, and output, *S22*, reflection coefficients. The reflection coefficients are the input voltage multiplied by 2 minus AC unity. The E device, E1, has a gain of 2. As in the transmission coefficients measurement circuit, the interface pin labeled CKT is used to connect to the external circuit. The pin, S_RE, is a hidden pin like S_TR described above, (see **Figure 5**). The REFLECTS symbol also has a *DCbias* property. On active circuits, the DC level can be set on voltage source, V1, by changing the *DCbias* property for the REFLECTS part instance in Capture. By default, this property is set to zero.
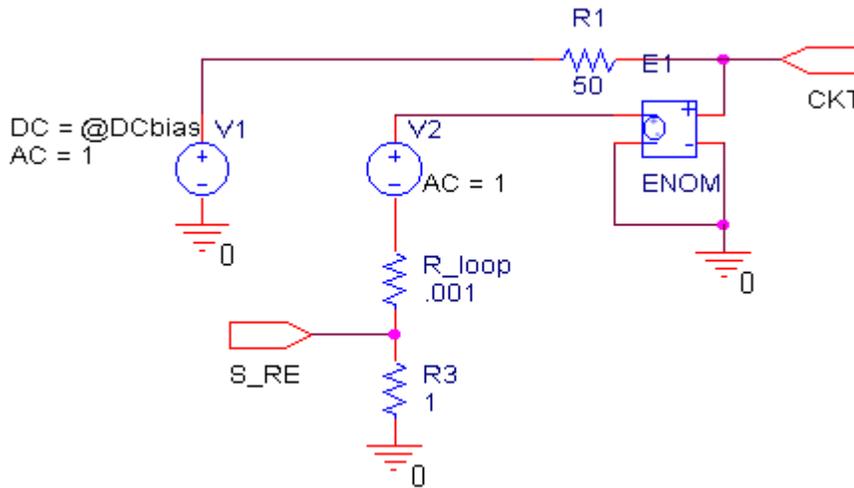
Fig 4. Reflection coefficients measurement subschematic
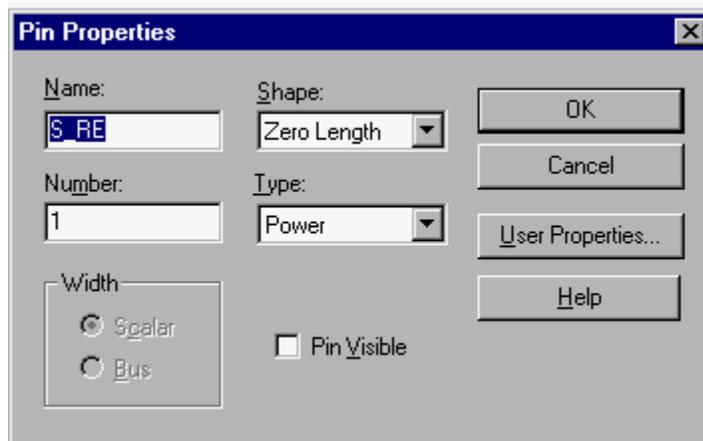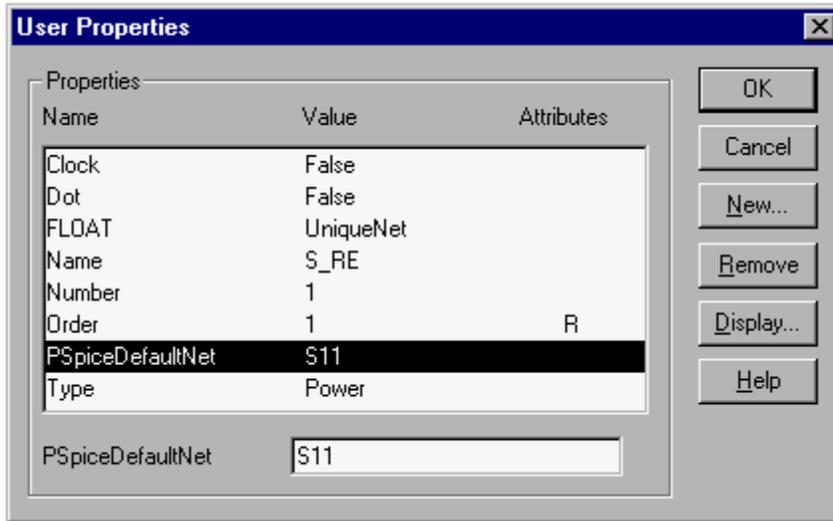
Fig 5. Pin Properties for the Reflects symbol showing hidden S_RE pin

Using the Subschematics

The subschematics can be used for both passive and active circuits. The circuit shown in **Figure 6** is a 4th-order Butterworth bandpass filter with a center frequency of 250 MHz. The first circuit measures *S11* and *S21*. The second circuit measures *S12* and *S22*. For simulation, the AC analysis settings are 500 linear points from 200 MHz to 350 MHz. The results of the analysis are shown in **Figure 7**.
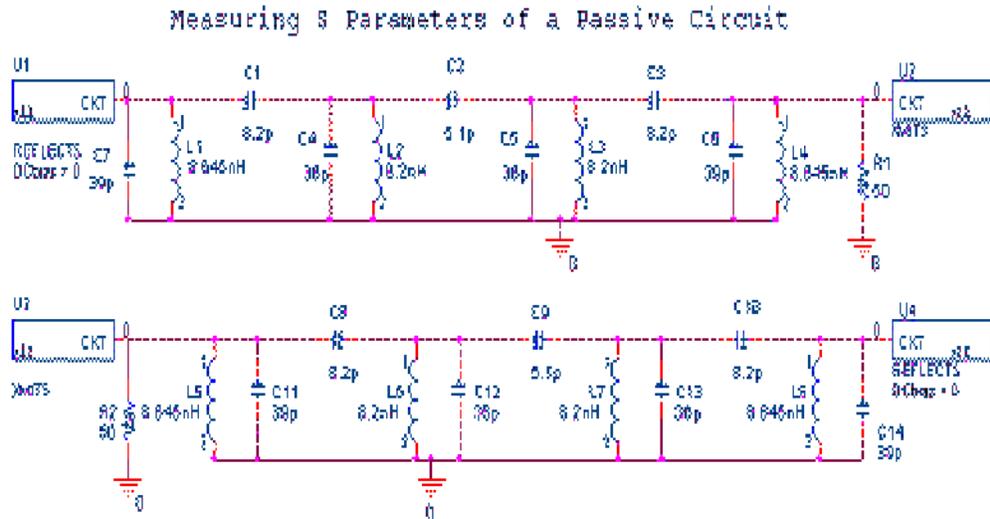
Fig 6. Bandpass filter example
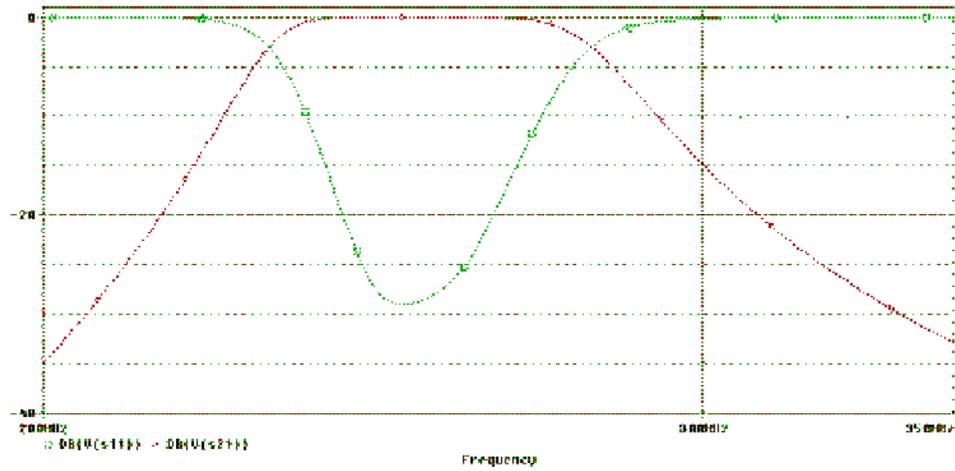
Fig 7. S11 and S21 for Filter Example

It is also possible to measure the s-parameters of an active circuit. To illustrate this, we have chosen to measure the s-parameters of the RF transistor, MRF5711/MC. **Figure 8** shows the circuits for this example. The transistor is biased for a VCE of 6.0 volts and an IC of 5.0 mA. The current is set by the current source at the emitter of the transistor. The DC bias is set by V1 in the circuit that measures $S11$ and $S21$, and by the *DCbias* attribute in the reflection measurement subcircuit that measures $S12$ and $S22$. For simulation, the AC analysis settings are 100 points per decade from 200 MHz to 2 GHz. The results shown in **Figure 9**, **Figure 10**, and **Figure 11** are expressed as magnitude and phase. This is typically the way most manufacturer's data sheets show the s-parameters.
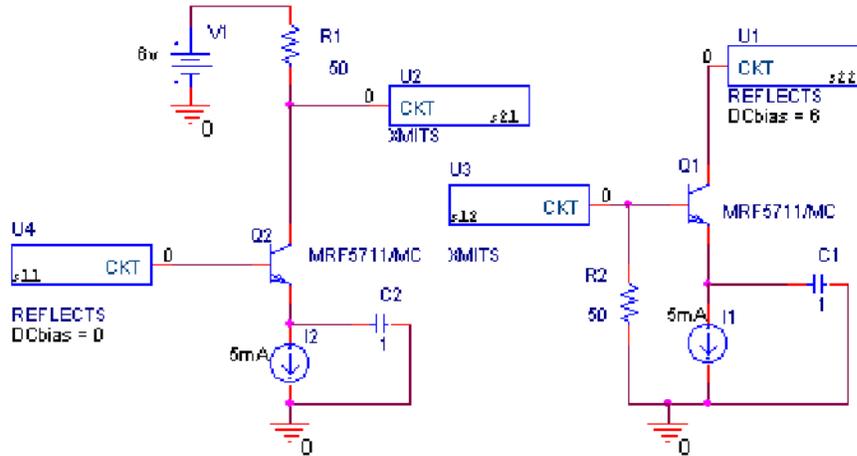
Fig 8. Transistor example
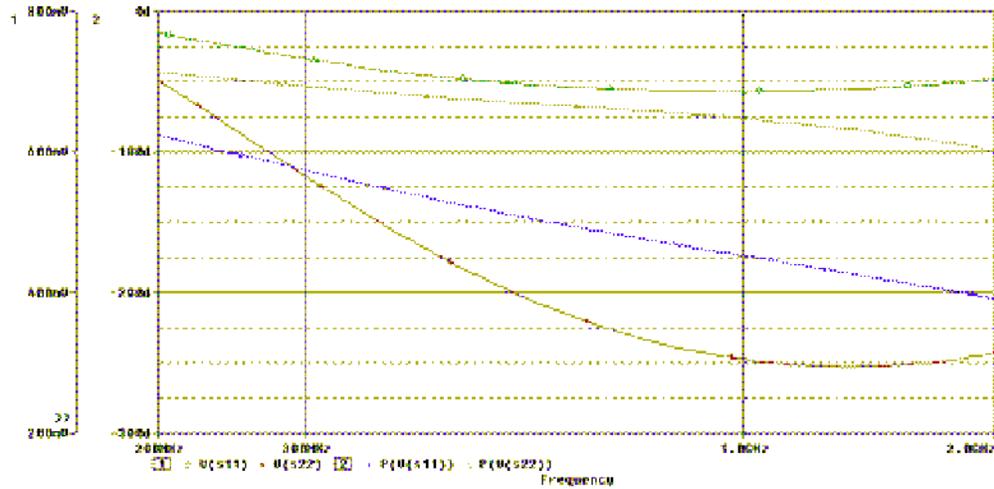
Fig 9. S11 and S22 magnitude and phase
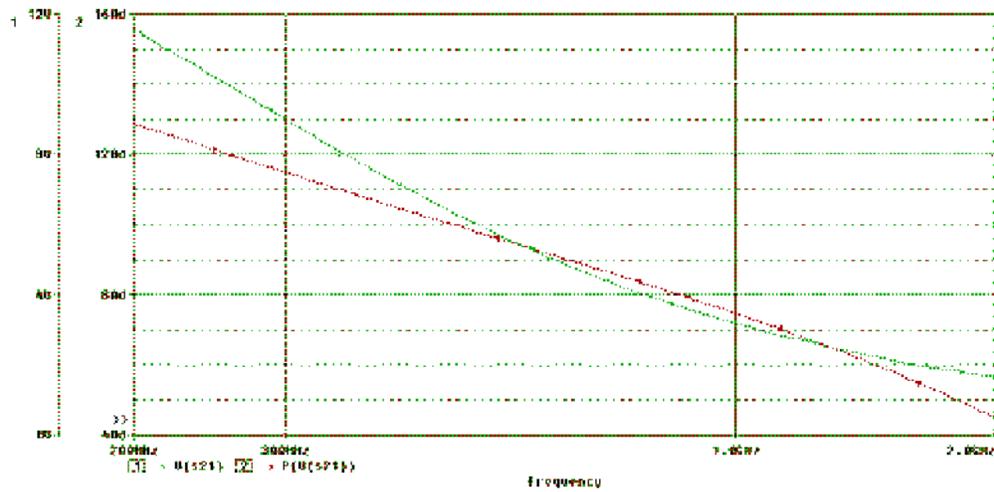
Fig 10. S21 magnitude and phase
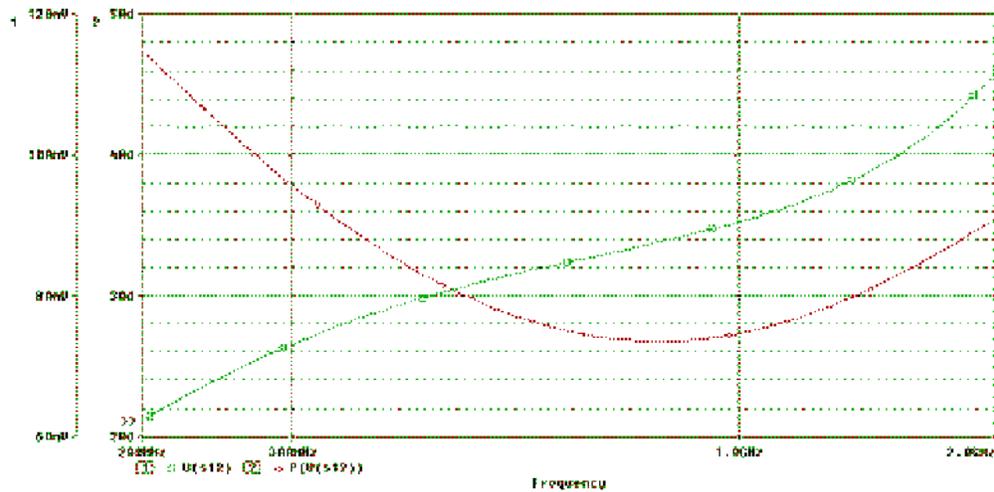
Fig 11. S12 magnitude and phase

TriQuint SEMICONDUCTOR®

Application Note

Robert Hickey
11/12/97

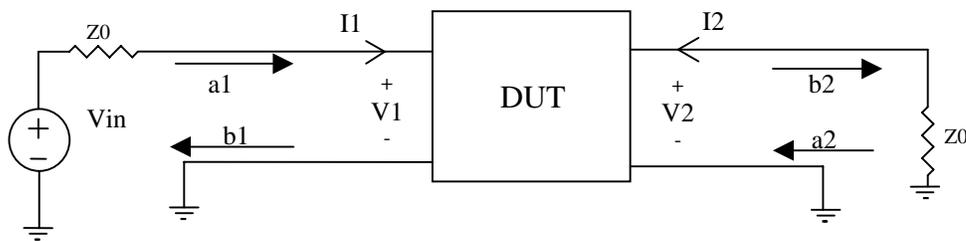## Extracting Scattering Parameters from SPICE

Scattering, or S parameters are easily generated by an AC analysis in SPICE. S parameters are defined with respect to incident (a1 and a2) and reflected (b1 and b2) voltage waves defined by the diagram below and by:

$$a1 = (V1 + Z0 * I1) / (2 * SQRT(Z0)) \qquad \text{(Eq. 1)}$$
$$b1 = (V1 - Z0 * I1) / (2 * SQRT(Z0)) \qquad \text{(Eq. 2)}$$
$$a2 = (V2 + Z0 * I2) / (2 * SQRT(Z0)) \qquad \text{(Eq. 3)}$$
$$b2 = (V2 - Z0 * I2) / (2 * SQRT(Z0)) \qquad \text{(Eq. 4)}$$

The scattering matrix relates the incident and reflected waves.

$$\text{(Eq. 5)} \qquad \begin{vmatrix} b1 \\ b2 \end{vmatrix} = \begin{vmatrix} S11 & S12 \\ S21 & S22 \end{vmatrix} \begin{vmatrix} a1 \\ a2 \end{vmatrix}$$

$S11 = b1/a1$ for the case when $a2 = 0$. a2 is the wave reflected from the load back toward the output port and is zero when the load is the impedance used to define the S parameters. Hence,

$$\text{(Eq. 6)} \qquad S11 = \frac{V1 \quad - \quad Z0*I1}{V1 \quad + \quad Z0*I1}$$

Noting that $Z1 = Z_{in} = V1/I1$, we have the familiar

$$\text{(Eq. 7)} \qquad S11 = \frac{Z1 \quad - \quad Z0}{Z1 \quad + \quad Z0}$$

Let the Voltage Source be 2 Volts. This value simplifies calculations without compromise, for an AC run in SPICE is a small signal analysis. The voltage at the network input is then found from a voltage divider as:

$$V1 = (2 * Z1) / (Z1 + Z0) \qquad \text{(Eq. 8)}$$

Solving for Z1 in terms of V1 results in:

$$Z1 = (V1 * Z0) / (2 – V1) \qquad \text{(Eq. 9)}$$

Substituting this in Eq. 7 yields the simple result:

$$S11 = V1 – 1 \qquad \text{(Eq. 10)}$$

This is more intuitive than many S parameter results. If a 2 volt source is used at the input, and if the input is perfectly matched, V1 will be 1 Volt. Subtracting 1 from that leaves S11 of zero, a "proper" result for a perfect impedance match. In SPICE, 1 is easily subtracted from V1 by attachment of a 1 Volt AC generator to the V1 node. The voltage at node 11 below is S11 in both magnitude and phase.
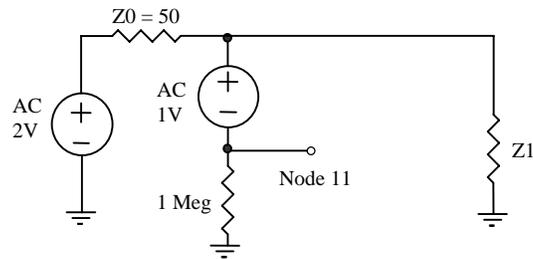
S21, the forward scattering parameter, is defined as S21=b2/a1. Using equations 1 and 4, this becomes

$$S21 = (V2 - I2*Z0) \; / \; (V1 + I1*Z0) \qquad (Eq.\ 11)$$

S21 is defined by Eq. 11 for the case when there is no reflection from the load, setting a2 = 0. This produces I2 = -V2/Z0, and Eq. 11 reduces to
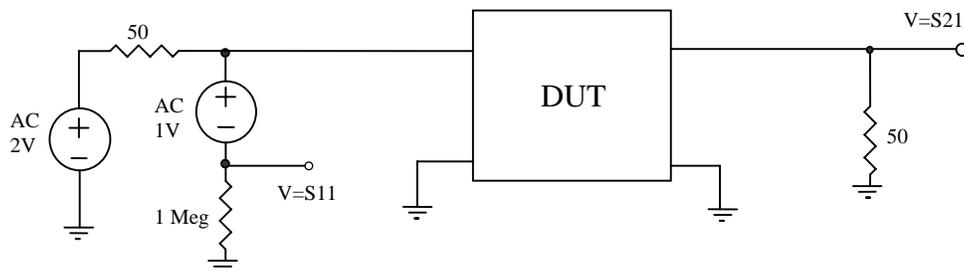
$$S21 = (2*V2) / (V1 + Z0 + I1) \qquad (Eq.\ 12)$$

Consider our special case for a 2 volt driving generator. I1, the input current, is related to the input voltage, I1 = V1 / Z1. Substitution into Eq. 12 yields

$$S21 = (2*V2/V1) / (1 + Z0/Z1) \qquad (Eq.\ 13)$$

Equation 9 relates Z1 to V1; substitution simplifies to the almost trivial result that

$$S21 = V2 \qquad\qquad (Eq.\ 14)$$

Hence, S21 and S11 are calculated in SPICE from

S12 and S22 are calculated with similar ease with