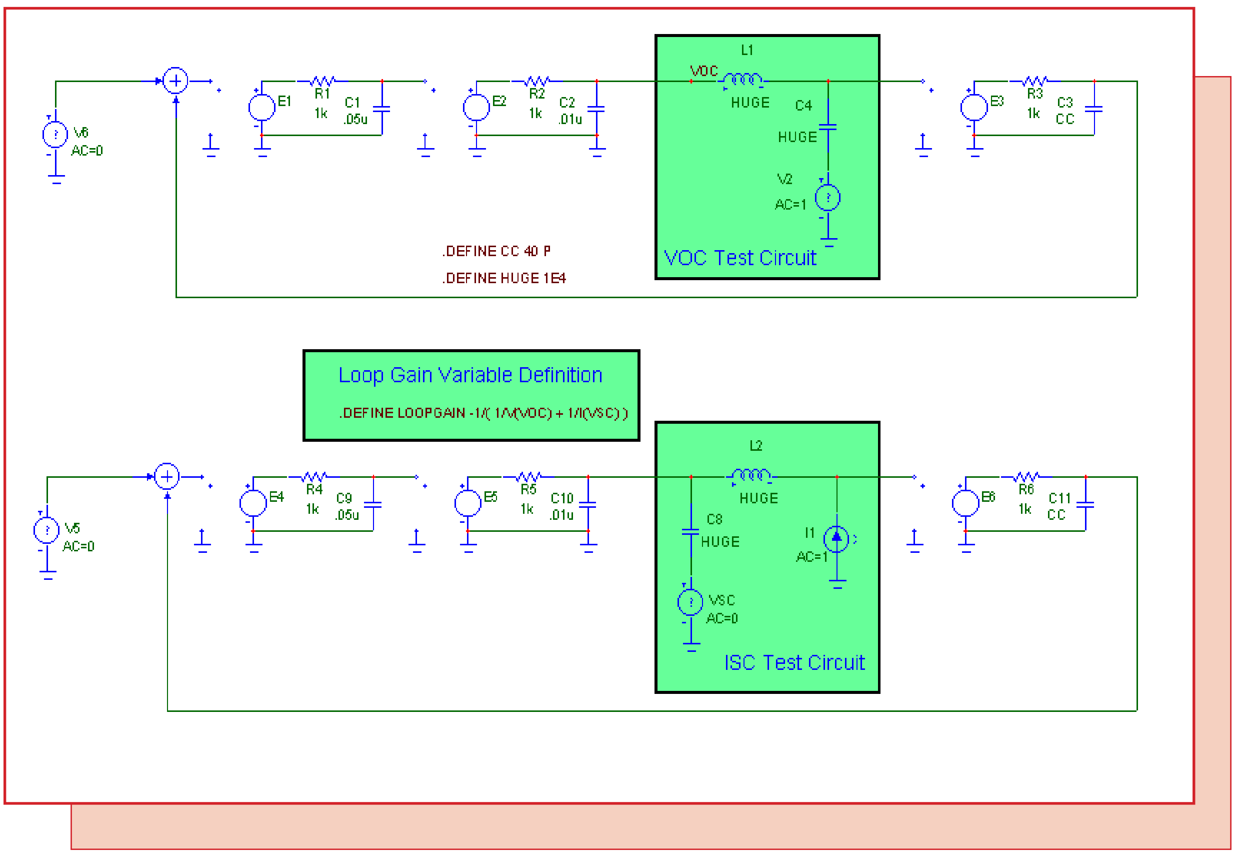
spectrum news

Applications for Micro-Cap™ Users

Winter 2001

Measuring Loop Gain and Phase Margin



Featuring:

- Plotting Loop Gain and Phase Margin
- Current-limited Power Supply Model
- Measuring S-Parameters
- Converting S-Parameters to Y-Parameters

News In Preview

This newsletter describes how to model some common components.

The first article describes how to measure loop gain and its phase margin and how to use phase margin to assess loop stability.

The second article shows how to create a current-limited power supply using an NTIOFV source.

The third article describes how to measure S-parameters.

The fourth article describes how to convert S-parameters to Y-parameters. It then describes how to use the Y-parameters in a circuit to model the original two-port the S-parameters were created for.

Contents

Book Recommendations

Micro-Cap / SPICE
- *Computer-Aided Circuit Analysis Using SPICE*, Walter Banzhaf, Prentice Hall 1989.
 ISBN# 0-13-162579-9
- *Macromodeling with SPICE*, Connelly and Choi, Prentice Hall 1992.
 ISBN# 0-13-544941-3
- *Semiconductor Device Modeling with SPICE*, Paolo Antognetti and Giuseppe Massobrio
 McGraw-Hill, Second Edition, 1993. ISBN# 0-07-002107-4
- *Inside SPICE-Overcoming the Obstacles of Circuit Simulation*, Ron Kielkowski,
 McGraw-Hill, First Edition, 1993. ISBN# 0-07-911525-X
- *The SPICE Book,* Andrei Vladimirescu, John Wiley & Sons, Inc., First Edition, 1994.
 ISBN# 0-471-60926-9
- *SMPS Simulation with SPICE 3,* Steven M. Sandler, McGraw Hill, First Edition, 1997.
 ISBN# 0-07-913227-8
- *MOSFET Modeling with SPICE Principles and Practice,* Daniel Foty, Prentice Hall, First Edition,
 1997. ISBN# 0-13-227935-5

German
- *Schaltungen erfolgreich simulieren mit Micro-Cap V,* Walter Gunther, Franzis', First Edition, 1997. ISBN# 3-7723-4662-6

Design
- *Microelectronic CircuitsHigh Performance Audio Power Amplifiers,* Ben Duncan, Newnes, First Edition,
 1996. ISBN# 0-7506-2629-1

References

- *Microelectronic Circuits.,* Adel Sedra, Kenneth Smith, Fourth Edition, Oxford, 1998

High Power Electronics
- *Power Electronics,* Mohan, Undeland, Robbins, Second Edition,
 1995. ISBN# 0-471-58408-8

RF Electronics
- *Microwave Circuit Design*, Vendelin, Pavio, and Rhoda, First Edition,
 1990. ISBN# 0-471-60276-0

Micro-Cap 6 Questions and Answers

Question: When I run AC analysis in a circuit with more than one source, how can I tell which source is providing the AC stimulus?

Answer: In AC analysis each waveform source contributes AC signals as follows:

Pulse Source:	Fixed at 1
Sine Source:	Fixed at 1
V (SPICE voltage source):	Specified by user, defaults to 0
I (SPICE current source):	Specified by user, defaults to 0
User Source	Value of expression in selected user file
Battery or Fixed Analog	Fixed at 0

Typically, only one source is used to provide an AC stimulus and the AC value is set to 1.0. All others are usually set to zero.

Question: How do I plot or print energy?

Answer: Energy is obtained by integrating one of the power expressions. For example, suppose you wanted to see the energy stored in an inductor versus time. Here is the expression you'd use:

Since PS(L1) is the power stored in the inductor L1 we need only integrate to get energy like this:

SUM(PS(L1),T)

SUM(PS(L1),T) integrates the expression P(L1) over time to produce the energy stored in L1.

To see the energy stored in the entire circuit, use the PST (Power Stored Total) variable like this:

SUM(PST,T)

To see the energy dissipated in the entire circuit, use the PDT (Power Dissipated Total):

SUM(PDT,T)

To see the energy generated by all the sources in the circuit, use the PGT (Power Generated Total)

SUM(PGT,T)

Question:
How do I add a new NMOS model with my own parameters?

Answer: First select the DNMOS component for placement from the Component menu / Analog Primitives / Active Devices / DNMOS. Then click in the schematic to place the part. When the Attribute dialog box comes up, enter any model name you wish, like IRF711A. This instantiates a model with default parameters. To edit them manually click on the Edit button and edit the parameters as needed. If you already have a model statement for the part, paste it into the text area before placing the part and the program will pick it up and place its name in the Attribute dialog box's model list. Click on the name to select the part and its model.

If you select a model name from the list, then edit one or more of its parameters, its modified model parameters become local to the circuit file.

Question:
How do I handle a model parameter that uses an expression?

Answer: MC6 can handle formulas or expressions in model parameters if the formula is enclosed in braces (e.g. VTO={2.5-TEM/100}), and if the variables used have been created with a .define statement. MC6 cannot handle simulation variables like TEMP (operating temperature) that are only known when the simulation is run. Future versions will be able to do this but MC6 cannot. If you have a model that includes a parameter formula that uses say the TEMP simulation variable, the way to handle it is this. Change the variable name in the formula to TEM. Then add a .define command like this:

.DEFINE TEM 27

Here TEM acts as an alias for the expected temperature of the run.

Each time you run an analysis at a new temperature, edit the command statement to the new temperature value. Of course, you cannot step temperature during the analysis this way, as that would require multiple values of temperature.

Easily Overlooked Features

This section is designed to highlight one or two features per issue that may be overlooked because they are not made visually obvious with an icon or a menu item.

Select Toggle with Space Bar
When drawing a circuit, one frequently switches between Component or Text mode while placing parts or text and the Select mode to move or edit the component or text. You can do this easily by pressing the space bar. It toggles between the current mode and the Select mode. If you are in Text mode and have just finished entering a piece of node text, but it didn't fall exactly where you wanted it, press the space bar to temporarily change to Select mode, drag the text to where you want it, then hit the space bar again to restore Text mode. This applies regardless of whether the mode is Component, Line, Text, Info, or any other of the drawing modes.

Other helpful accelerator keys for the common modes are as follows:

Mode	Key
Select	CTRL+E
Text	CTRL+T
Component	CTRL+D
Wire	CTRL+W
Info	CTRL+I
Help	CTRL+H

Plotting Loop Gain and Phase Margin

Circuit stability

What is stability? How do you determine if a circuit is stable? A circuit is unstable if it spontaneously starts oscillating in the presence of noise alone. Consider a typical, three-pole circuit show below.

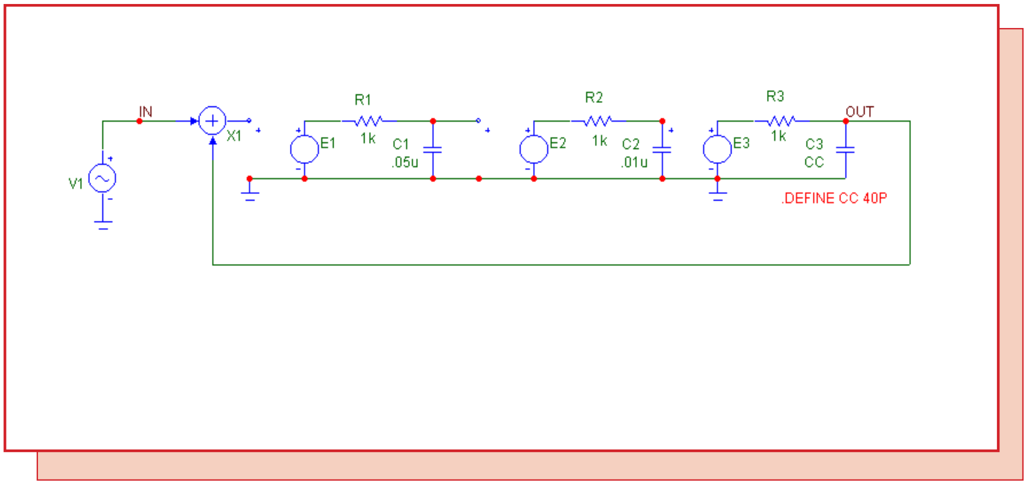


Fig. 1 - A three-pole circuit with feedback

If you run this circuit in transient analysis, it will produce the following plot.

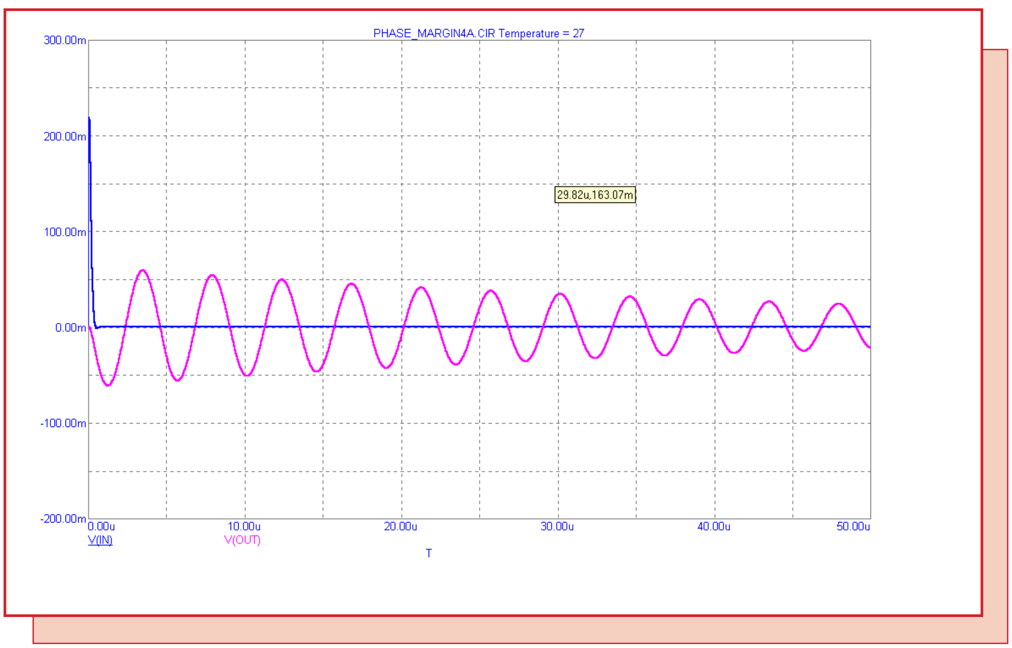


Fig. 2 - With CC=40pf the circuit is (barely) stable

In this circuit the input is a one volt damped sine wave programmed to simulate a noisy spike. When the value of the last capacitor, CC, is 40pF, the circuit is stable as can be seen from the damped oscillation on the output.

What happens when we changed CC?. If the value of CC is changed from 40pf to 80 pf, the result is much different. As Figure 3 shows, the circuit now produces undamped oscillation.

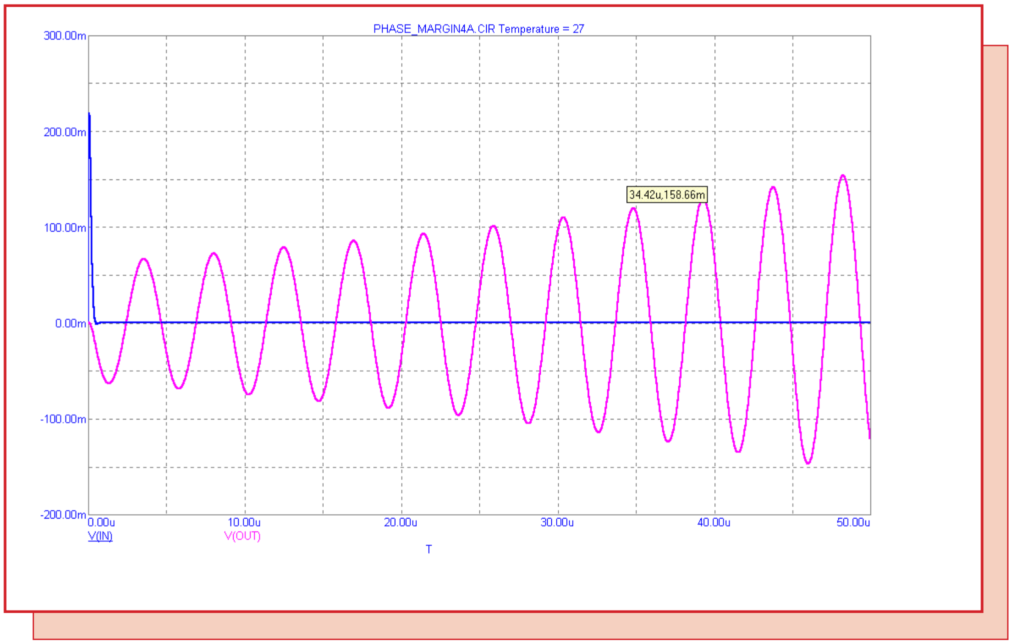


Fig. 3 - With CC=80pf the circuit is unstable

This is one way to test for circuit stability. The other way is to examine the loop gain and use the phase margin test.

Loop Gain

Knowing the loop gain of a closed-loop feedback system is essential to understanding its stability properties. It is easy to plot open loop gain. You simply place an AC source at the input and plot the signal value at the output. But what about loop gain? How do you get that? First, a little review.

The general feedback circuit sketched below produces a loop gain of **A*β**.

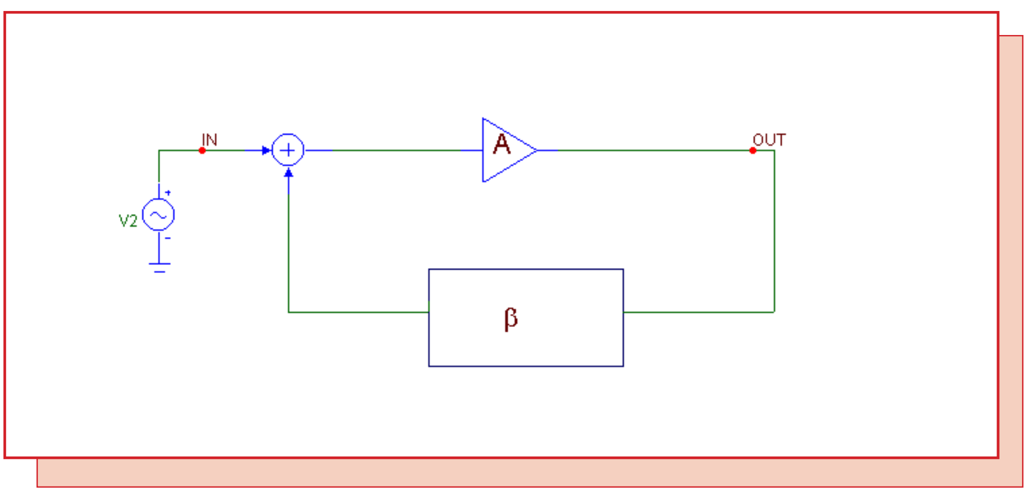


Fig. 4- A general circuit with feedback

Stability Requirement

Stability requires that the loop gain phase angle should be greater than -180 degrees when the loop gain is 1.0.

Phase margin is the amount by which the phase angle exceeds -180 degrees when the loop gain is precisely 1.

Phase margin = Phase Angle - (-180) = Phase Angle + 180.

Phase margin measures the circuit's stability, or in other words, its susceptibility to oscillations. If the phase margin is, say 10 degrees, the circuit will not oscillate and any noisy disturbance on the input will produce, at most, a damped oscillation. If the phase margin is exactly 0, the circuit will oscillate at a steady amplitude. If it is, say, -10 degrees, the circuit will break into undamped oscillations which eventually will be limited to whatever the power supplies dictate. In general designers strive to achieve the following.

Phase Margin Rule
Keep the phase margin greater than 45 degrees.

As circuits age, their components will vary, possibly pushing a marginal circuit into instability. Phase margin quantifies the safety margin the circuit has to such variations.

Breaking the loop:
To measure loop gain and its phase we must break the loop. The loop must be broken only in an AC sense, as the DC or biasing circuitry must be allowed to operate. The trick to doing this is to strategically place large valued inductors and capacitors to close the loop for DC (low frequency bias) signals and open the loop for AC (higher-frequency) signals. Rosenstark (1986) developed the following formula:

Loop Gain = -1 / (1/Toc+1/Tsc)

Toc = open circuit voltage gain = V(VOC) / 1 = V(VOC)

Tsc = short circuit current gain = 1/I(VSC) / 1 = 1/I(VSC)

All we need do is measure Toc and Tsc and plot the Loop Gain function. The following circuit

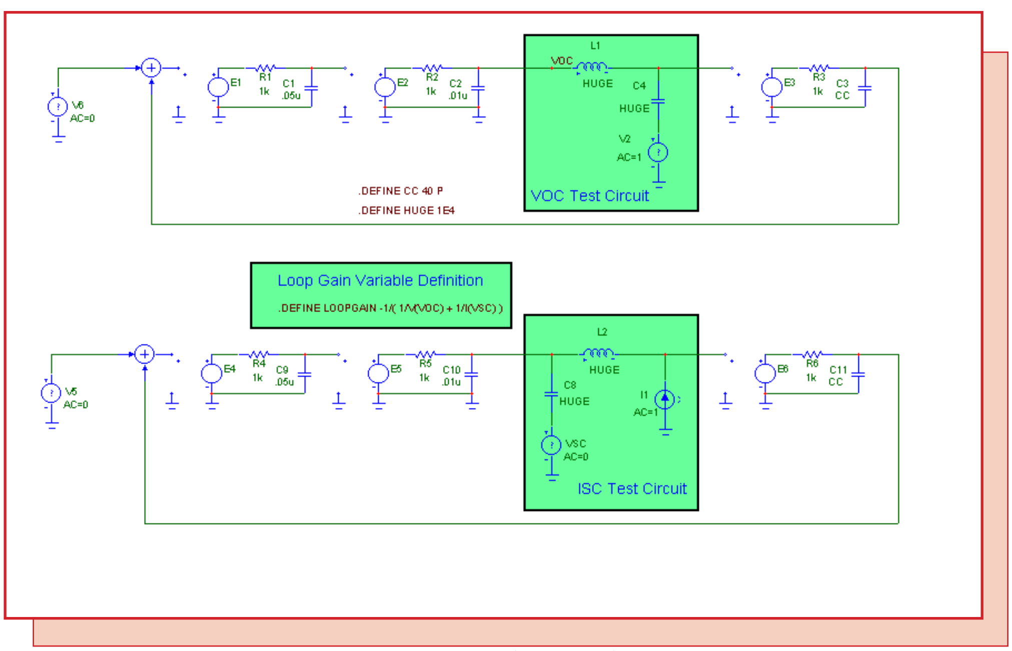


Fig. 5- Plotting loop gain

uses two copies of the three-stage feedback circuit of Figure 1. One copy is used to measure Toc, the loop voltage gain and the second circuit measures the Tsc, the short circuit gain. The symbolic variable LOOPGAIN embodies the loop gain formula and provides a convenient plot variable. This is the general procedure for testing loop gain on an arbitrary circuit:

1) Set the AC magnitude of all of the original sources to 0. Copy the circuit to be tested (DUT) to the clipboard. (CTRL+A then CTRL+C).

2) Make a new circuit and paste two copies of the DUT (CTRL+V) from the clipboard to the new circuit. The first copy is for measuring Toc and the second copy is for measuring Tsc.

3) Break the loop of the first copy and insert a large AC loop breaking inductor (L1) and a large capacitor (C4) that AC 'shorts in' the 1.0 VAC input test source (V2). Add the Voc text label the point where the open circuit voltage will be measured.

4) Break the loop of the second copy and insert a large AC loop breaking inductor (L2) and a large capacitor (C8) that AC 'shorts in' the 0.0 VAC output current measuring source (VSC). Add the 1 Amp AC input current source (I1).

5) Add the LOOPGAIN define statement text as shown in Figure 5.

6) Run AC and plot dB(LOOPGAIN) on plot 1 and PH(LOOPGAIN) on plot 2.

7) When the run is done, invoke the GoTo Y command (SHIFT + CTRL+Y), type in 0 in the Value box. Click on the Left button, then on the Close button. The left cursor will be positioned on the 0 dB point of the gain curve and at the corresponding frequency point on the phase curve.

8) Read the phase in the "PH(LOOPGAIN)" plot as -178.37 degrees. The circuit is barely stable with less than 2 degrees margin of safety.

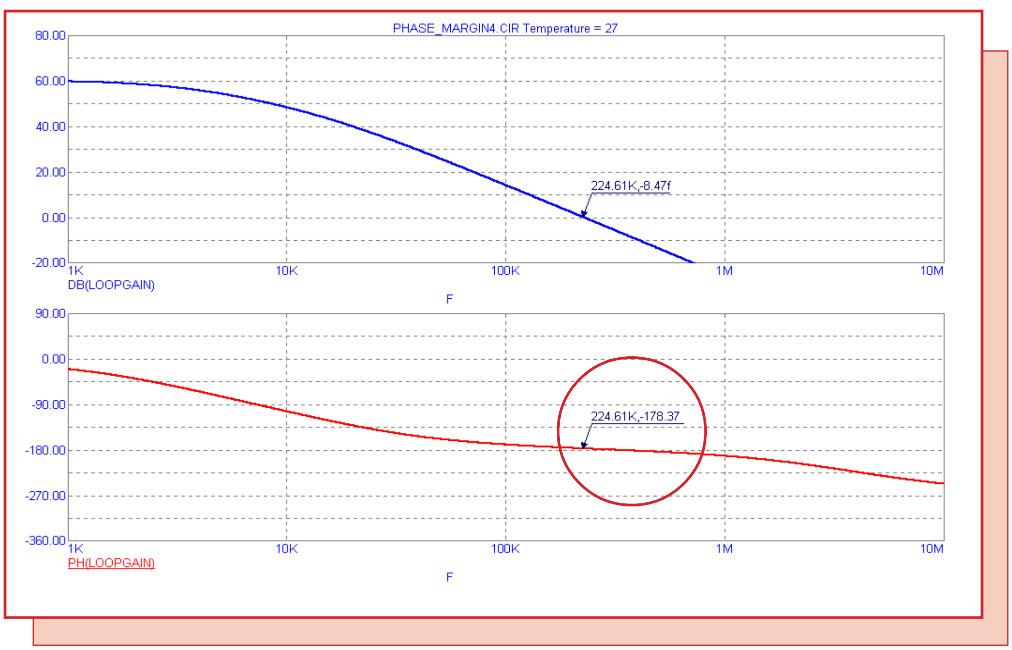


Fig. 6 Phase margin wiith CC = 40pf = 180 - 178.37 = 1.73 degrees = barely stable

What happens when we increase CC to 80 pF? Here is what the plot looks like:

Fig. 7 Phase margin with CC = 80pf = 180 - 181.55 = -1.55 degrees = unstable

The phase margin is -1.55 degrees so the circuit is unstable. By experimenting with the CC value you can determine that the actual crossover from stability to instability in this circuit occurs when CC is between 60pF and 61 pF.

Helpful Hints

Where do you break the loop? In this circuit it does matter much, but in general the loop should be broken such that the V(VOC) measurement can be made where the circuit has a low output impedance.

The Sine and Pulse and User sources all have a fixed AC magnitude of 1.0, so you cannot use them at the usual input since their AC magnitudes cannot be set to 0. Use the SPICE V source as we've done in this example.

In many circuits, the short-circuit current gain, Tsc, will be extremely large when the input impedance is extremely large. If it is known to be very large, as it is in this example, 1/Tsc may be small in comparison with 1/Toc. If so, you can skip the Tsc test circuit entirely. Then you would change the LOOPGAIN symbolic statement to reflect the zero value of Tsc as follows:

.DEFINE LOOPGAIN -V(VOC)

Do not use too large a value for the HUGE inductors and capacitors. If it is too small, it distorts the low frequency loop gain plot. This is mostly an esthetic problem as the phase margin measurement is made at higher frequency. If HUGE is too large it may cause numeric problems in the matrix solver. A value of 1E2 to 1E4 works well. A value of 1E9 (suggested in one reference) will cause numeric problems with most analog simulators.

Here is another example similar to the one in Sedra and Smith (pg 739).

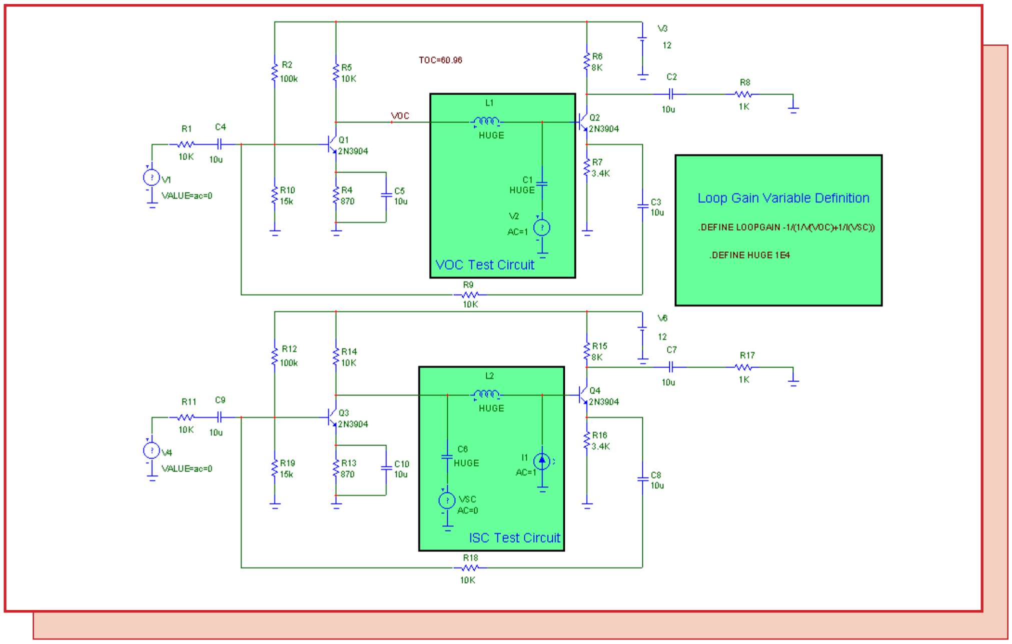


Fig. 8- Loop gain of the Sedra Example

And here is a plot of its loop gain and its phase.

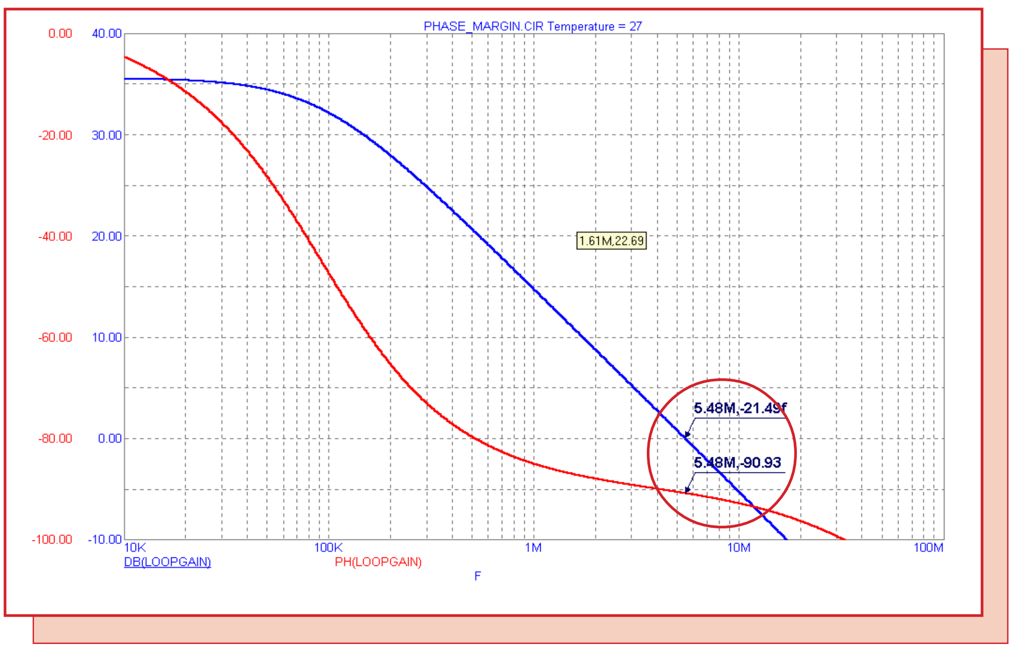


Fig. 9 Phase margin of the circuit = 180 - 91 degrees = 89 degrees = stable

This circuit has only one inverting stage so its phase shift is safely within the stability requirement. Its phase margin measures about 89 degrees.

Current-limited Power Supply Model

Ever wanted to make a power supply that was current limited? Here is an easy way to generate a current-limited voltage source.

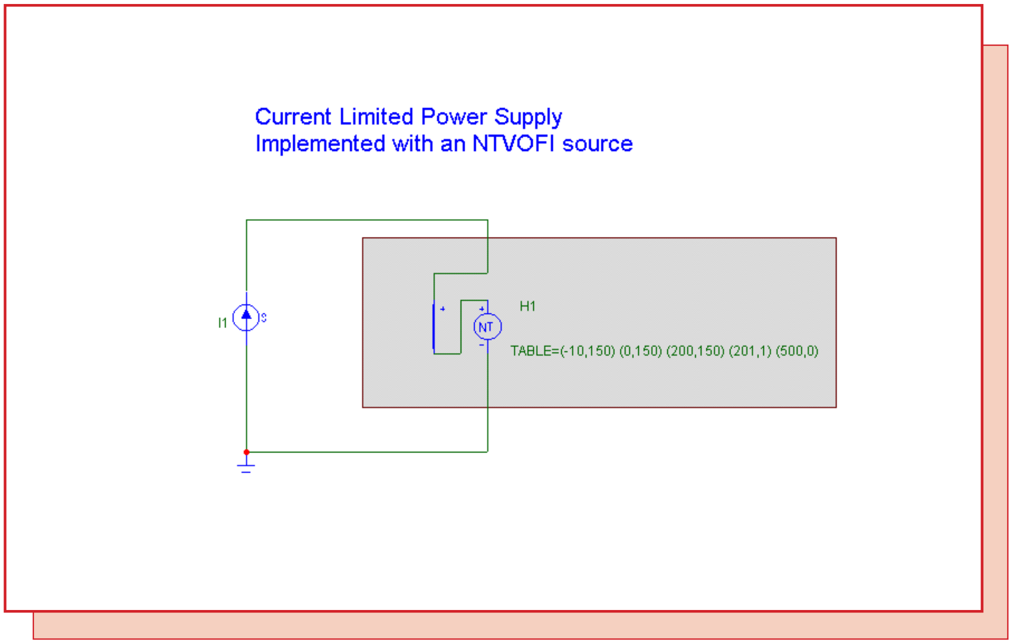


Fig. 10- The current-limited voltage source circuit

The source is implemented with an NTIOFV source. This source has its voltage controlled by the current flowing through its input terminals according to a tabular list. You control the degree of current limiting by constructing the appropriate current, voltage data pairs in the table. Here is a plot of the output voltage versus the input current. Notice that it exactly matches the values specified in the TABLE attribute.

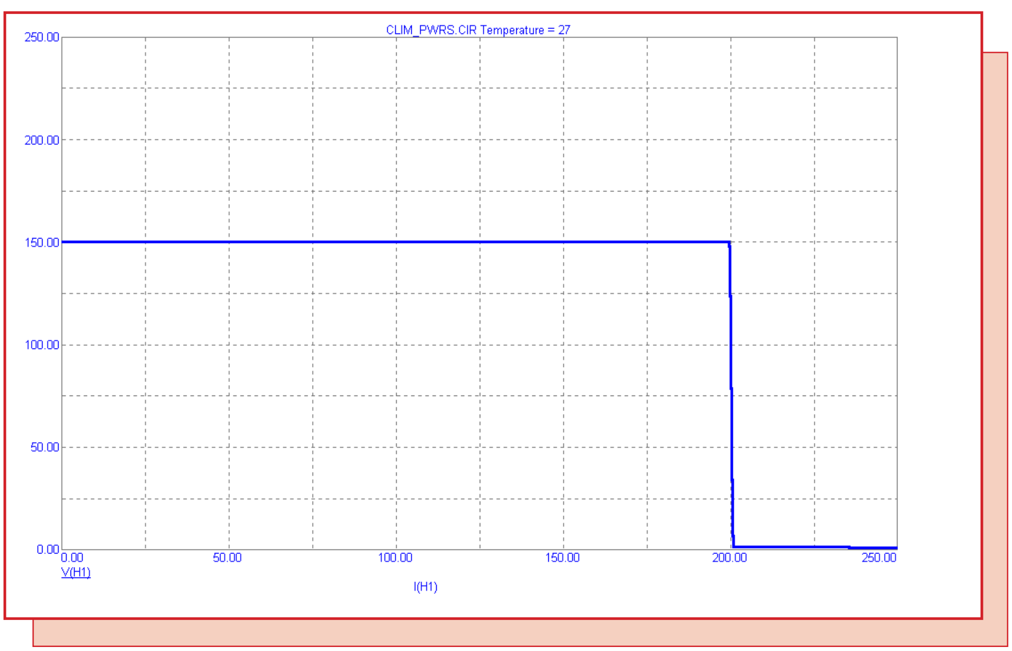


Fig. 11- The output current vs. voltage characteristic

Measuring S-Parameters

In this section we show how to measure the S-parameters of any two-port network. S-parameters are defined with respect to the two-port network of Figure 12 below as:

$$b1 = S11*a1 + S12*a2 \qquad (1)$$
$$b2 = S21*a1 + S22*a2 \qquad (2)$$

Assuming that R0 is purely real,

a1 = Normalized incident voltage at port 1=$(V1+R0*I1)/(2*\mathrm{sqrt}(R0))$ (3)
a2 = Normalized incident voltage at port 2=$(V2+R0*I2)/(2*\mathrm{sqrt}(R0))$ (4)
b1 = Normalized reflected voltage at port 1=$(V1-R0*I1)/(2*\mathrm{sqrt}(R0))$ (5)
b2 = Normalized reflected voltage at port 2=$(V2-R0*I2)/(2*\mathrm{sqrt}(R0))$ (6)

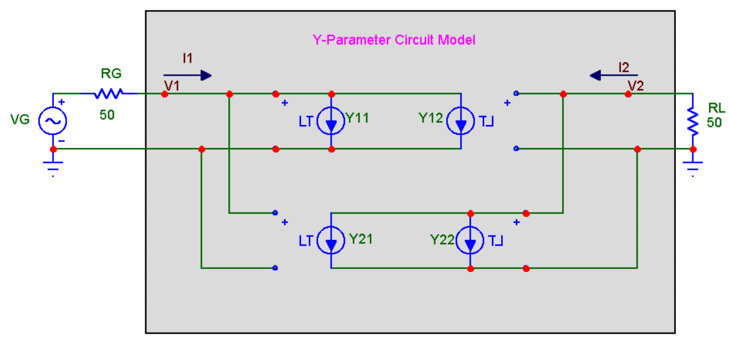


Fig. 12- The Y-Parameter equivalent circuit

S11 is measured when a2=0, so

$$S11=b1/a11=(V1-R0*I1)/(2*\mathrm{sqrt}(R0))/(V1+R0*I1)/(2*\mathrm{sqrt}(R0))=(V1-R0*I1)/(V1+R0*I1) \qquad (7)$$

$-VG + R0*I1 + V1 = 0$ (From Kirchoff's Law at port 1) (8)

$S11 = 2*V1-1$ (Solving (7) and (8)) (9)

Similarly, we can solve for the remaining S12, S21, and S22 to get

$S21 = 2*V2$

$S12 = 2*V1$

$S22 = 2*V2-1$

Measurement procedure

1) Set up the test circuit with bias circuitry to properly bias the device. Add large (10-100) value inductors or capacitors as needed to provide the necessary short or open circuits for the biasing circuitry. Make sure that the isolating inductors and capacitors do not affect the high frequency measurements. The inductors should be in series with the bias circuitry and the capacitors should be in series with the input and output signal paths.

2) To measure S11 and S21, drive the input port through a resistor of R0 with a source having an AC magnitude of 1.0. Place an R0 resistor across the output port. R0 is normally 50 ohms.

3) To measure S22 and S12, drive the output port through a resistor of R0 with a source having an AC magnitude of 1.0. Place an R0 resistor across the input port.

4) Add the text labels "IN" and "OUT" at the input and output nodes.

5)Add the following define commands to the circuit.

```
.define S11 2*V(IN)-1
.define S21 2*V(OUT)
.define S12 2*V(IN)
.define S22 2*V(OUT)-1
```

6)Run AC analysis. To plot the magnitude and phase of S11, plot S11 and PH(S11). Plot the other parameters similarly.

Here for example, is the Siemens BFG194 device, together with its package parasitics, embedded in the circuitry needed to measure the S11 and S21 parameters.

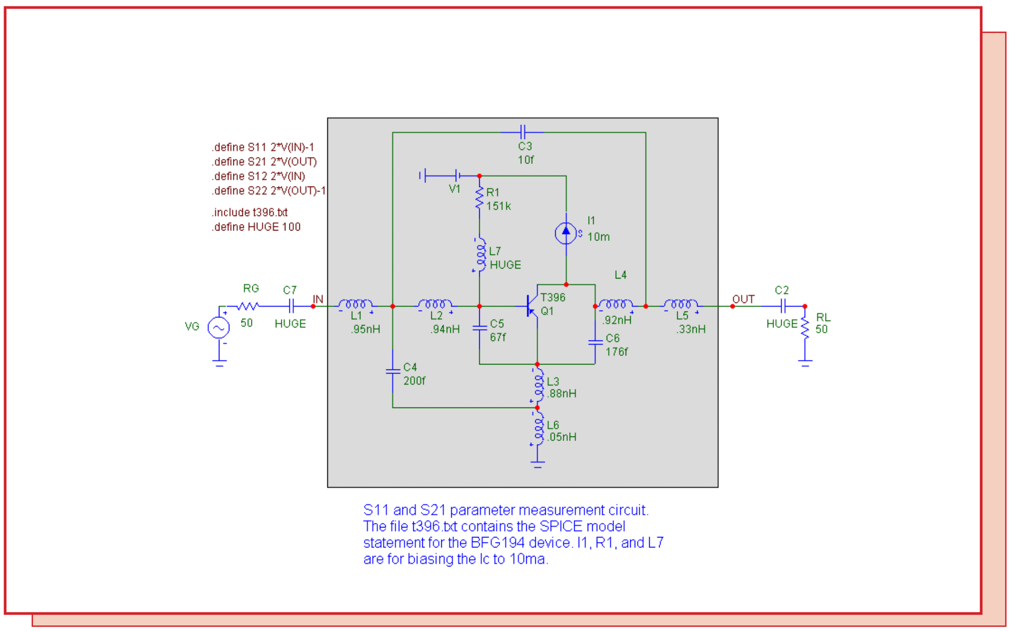


Fig. 13- Test circuit for measuring S11 and S21 on the BFG194

Figure 14 shows the plot of the S11 and S21 parameters for the BFG194 device.

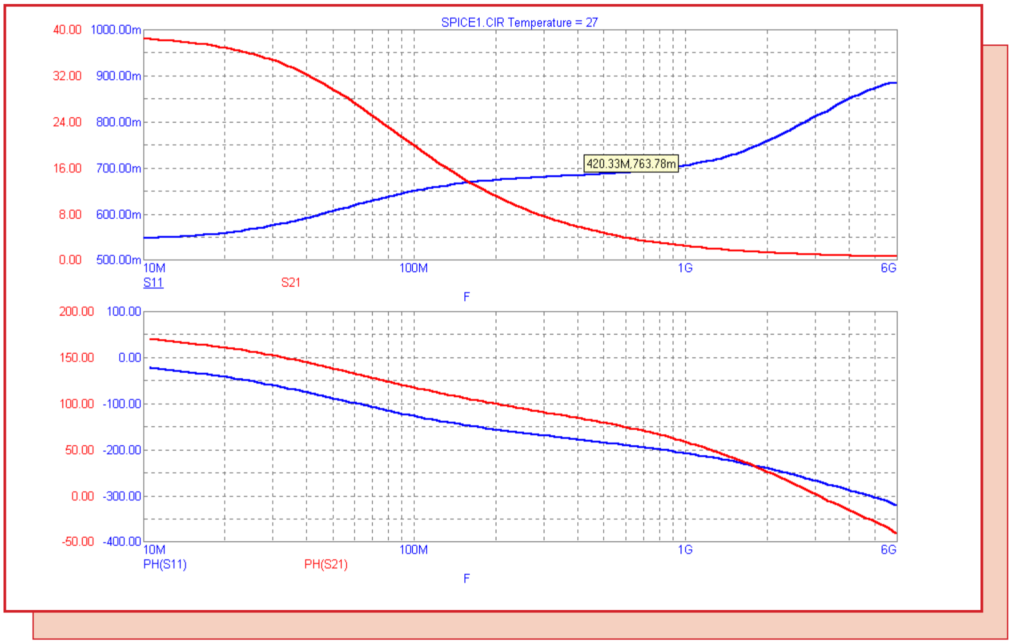


Fig. 14- Plotting S11 and S21

Here is the same Siemens BFG194 device, together with package parasitics, embedded in the circuitry needed to measure the S22 and S12 parameters.

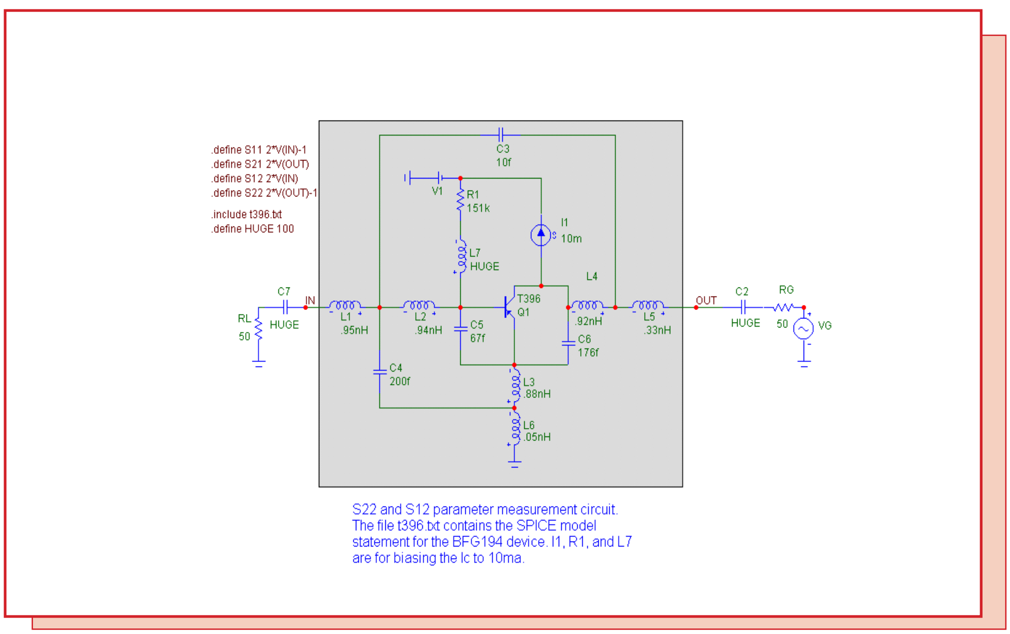


Fig. 15- Test circuit for measuring S22 and S12 on the BFG194

Figure 16 shows the plot of the S22 and S12 parameters for the BFG194 device.

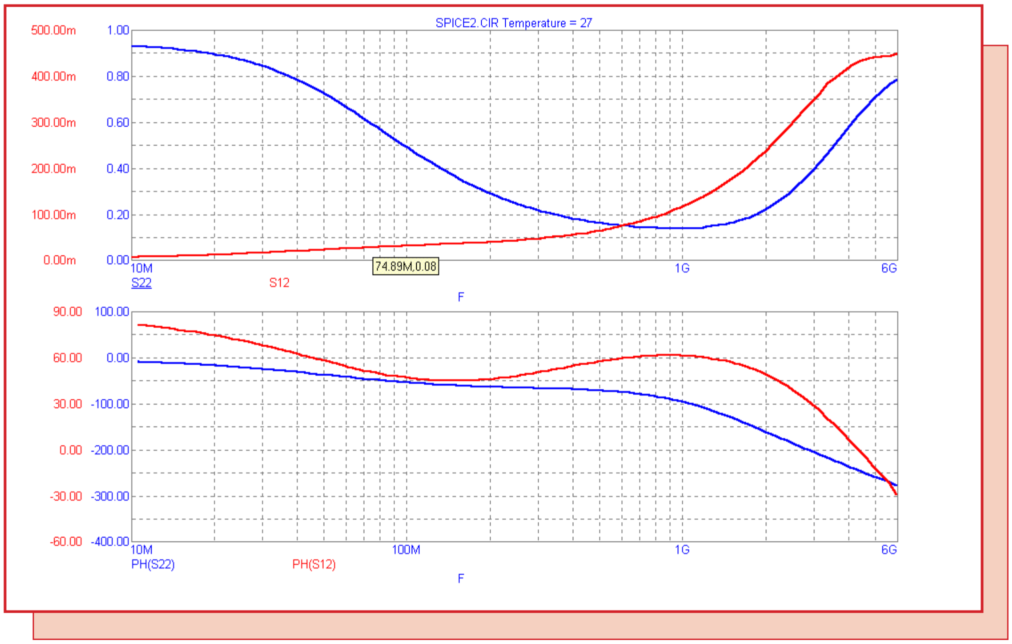


Fig. 16- Plotting S22 and S12

While the procedure for measuring S-parameters is quite straightforward it requires a SPICE or other nonlinear model for the device. What if you don't have such a model? Well then perhaps you can use S parameters directly. Manufacturers frequently provide measured S-parameters for RF devices. Using these supplied S-parameter tables is the topic of the next article.

Converting S-Parameters to Y-Parameters

Manufacturers of RF devices often supply S-parameters for their parts in the form of tables of magnitude and phase versus frequency. Micro-Cap 6 cannot directly import S-parameter tables, so how can you use them? The answer is to convert the S-parameter equations to Y-parameter equations and then implement them with Laplace table sources. Y-parameters may be calculated directly from S-parameters using the following standard formulas (See "Microwave Circuit Design", by Vendelin, Pavio, and Rhoda page 16).

D = ((1+S11)*(1+S22)-S12*S21)

y11 = ((1-S11)*(1+S22)+S12*S21) / D
y12 = -2*S12 / D
y21 = -2*S21 / D
y22 = ((1+S11)*(1-S22)+S12*S21) / D

All we need to do is to read in the S-parameters from a file, calculate the equivalent Y-parameter table values at each frequency using the formulas above, and write a file with four tables of values, one each for y11, y12, y21, and y22. The table file is then imported into a circuit like Figure 17 that implements the Y-parameter equations with Laplace table voltage-controlled current sources.

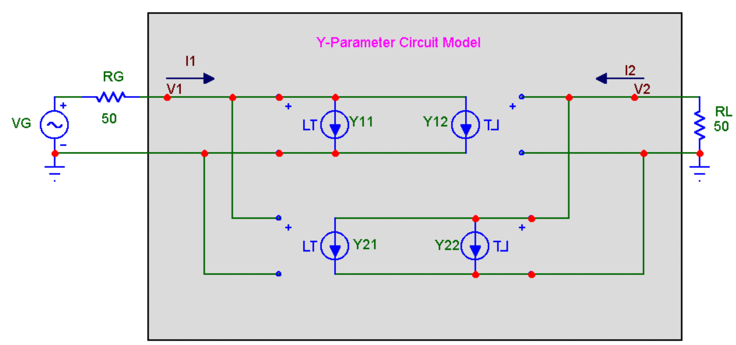


Fig. 17- The Y-Parameter equivalent circuit

This equivalent circuit implements the standard Y-parameter equations:

I1 = y11*V1 + y12*V2

I2 = y21*V1 + y22*V2

where V1 and I1 are the input port voltage and current and V2 and I2 are the output port voltage and current.

While the conversion formulas look simple, they require complex arithmetic. To ease the conversion, we have made a DOS program called STOY(S to Y converter) available for download from our web site at www.spectrum-soft.com.

STOY operation
The program is run with command line arguments that list the input and output file names:

stoy infile > outfile

for example,

stoy GG10V10M.STP > 194.OUT

STOY input file format
The program reads in an ASCII text file containing the S-parameters in standard tabular value format as follows:

Frequency S11 Mag S11 Ph S12 Mag S12 Ph S21 Mag S21 Ph S22 Mag S22 Ph

where Mag is the magnitude and Ph is the phase in degrees. These can optionally be the real and imaginary parts. A header line of the following form specifies the format.

[K | M | G | T]Hz S [MA | RI] R[Z0]

K, M, G, or T specify optional frequency units.

S identifies the data as S parameters.

MA specifies the S-parameter data pair format as magnitude, angle, while RI specifies the S-parameter data pair format as real, imaginary.

Z0 specifies the characteristic impedance value.

For example, here is a fragment of the input file for the BFG194 device from Siemens.

```
! SIEMENS Small Signal Semiconductors
! BFG194
! Si PNP RF Bipolar Junction Transistor in SOT223
! VCE = -10 V     IC = -10 mA
! Common Emitter S-Parameters:                August  1996
# GHz  S  MA  R  50
!  f       S11           S21           S12           S22
! GHz    MAG   ANG     MAG   ANG     MAG   ANG     MAG   ANG
 0.010  0.5524  -14.8  24.054  172.0  0.0064   85.3  0.9481   -7.0
 0.020  0.5479  -29.1  23.311  165.6  0.0127   78.0  0.9393  -13.9
 0.050  0.5716  -66.6  20.450  146.2  0.0290   63.9  0.8185  -31.5
...
 4.000  0.8441   67.5   0.535  -44.3  0.3928  -14.3  0.5992   90.5
 4.500  0.8478   58.3   0.476  -52.0  0.4157  -25.2  0.6579   73.1
 5.000  0.8611   50.1   0.448  -57.6  0.4250  -37.1  0.7108   57.5
 5.500  0.8600   42.7   0.427  -63.7  0.4407  -48.6  0.7486   42.1
 6.000  0.8579   35.6   0.405  -71.5  0.4307  -61.4  0.7538   28.4
```

STOY output

STOY generates an output file that contains define statments for each of the four Y sources. The output file typically looks like this:

```
.define y11
+ (1.000000e+007,-44.567845,10.243495) (1.250000e+007,-44.439330,12.559271)
(1.500000e+007,-44.299133,14.804465) (1.750000e+007,-44.148469,16.975683)
 ...
.define y12
+ (1.000000e+007,-81.407425,-88.961998) (1.250000e+007,-79.433630,-89.671276)
(1.500000e+007,-77.825623,-90.140868) (1.750000e+007,-76.468959,-90.474666)
...
.define y21
+ (1.000000e+007,-9.907279,-362.261998) (1.250000e+007,-9.935632,-362.475424)
(1.500000e+007,-9.963956,-362.690248) (1.750000e+007,-9.992250,-362.906476)
...
.define y22
+ (1.000000e+007,-66.016592,-699.839215) (1.250000e+007,-66.212254,-692.815564)
(1.500000e+007,-66.275002,-685.578696) (1.750000e+007,-66.199028,-678.352899)
...
.define s11
+ (1.000000e+007,-5.154927,-14.800000) (2.000000e+007,-5.225974,-29.100000)
(5.000000e+007,-4.858156,-66.600000)
...
.define s21
+ (1.000000e+007,27.623746,172.000000) (2.000000e+007,27.351218,165.600000)
(5.000000e+007,26.213866,146.200000)
...
.define s12
+ (1.000000e+007,-43.876401,85.300000) (2.000000e+007,-37.923926,78.000000)
(5.000000e+007,-30.752040,63.900000)
...
.define s22
+ (1.000000e+007,-0.462917,-7.000000) (2.000000e+007,-0.543914,-13.900000)
(5.000000e+007,-1.739626,-31.500000)
```

Using the STOY output

When the 194.OUT file generated by STOY is imported into a circuit with a statement like

```
.import 194.out
```

the symbolic variables, y11, y12, y21, and y22 defined by these statements are available for use in the circuit. In the sample circuits to follow we have set the VALUE attribute of each source to one of these four variable names. Definitions of s11, s12, s21, and s22 are also included so that a quick comparison can be made between the measurements of the S-parameters on the Y-equivalent circuit and the original S-Parameters can be compared.

Figure 18 shows how to use the equivalent Y-parameter circuit to model a two-port whose parameters are imported from a file (194.out) generated by the STOY program. This is a very general circuit that can model any two-port. You need only change the name of the input file.

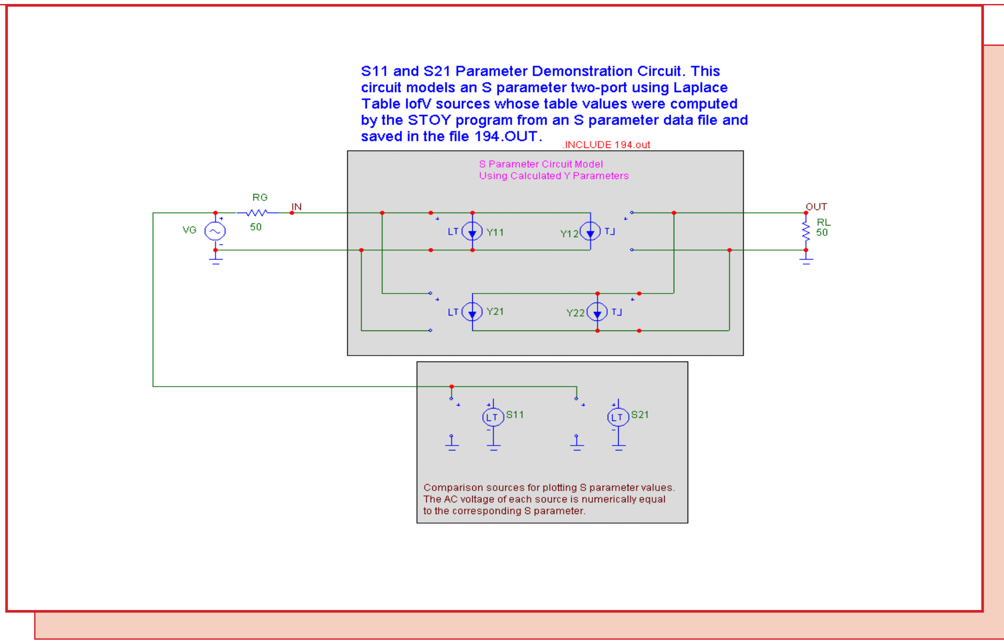


Fig. 18- Using the Y-parameter equivalent circuit

Here is a plot of the measured S11 and S21 parameters, together with a comparison plot of the original S11 and S21 parameters.

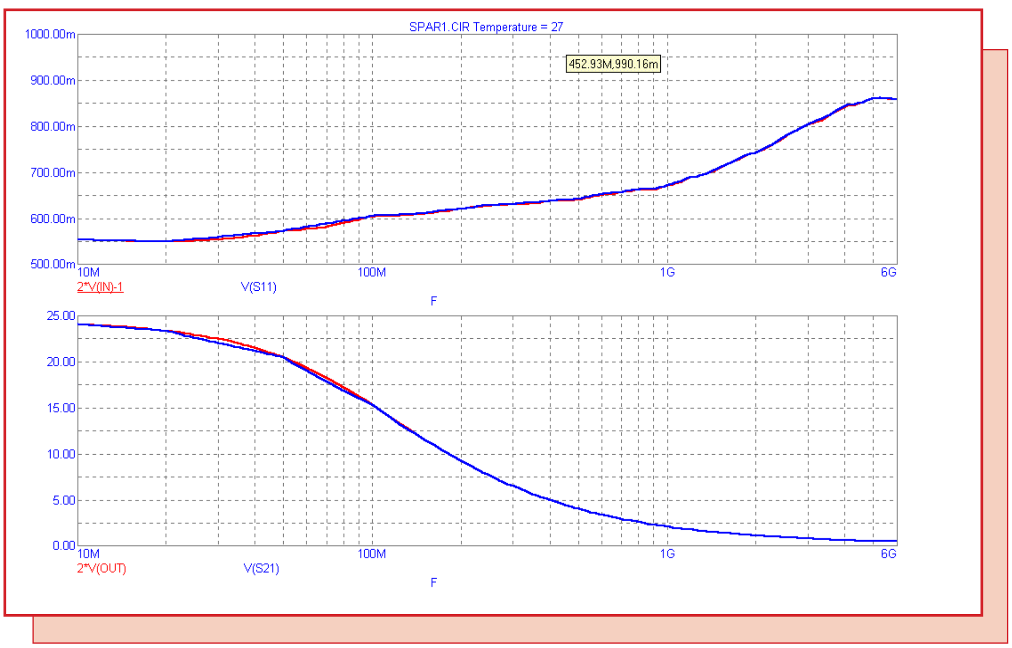


Fig. 19- Measured and original S11 and S21 parameters

Figure 20 shows the equivalent Y-parameter set up to measure the S22 and S12 parameters.

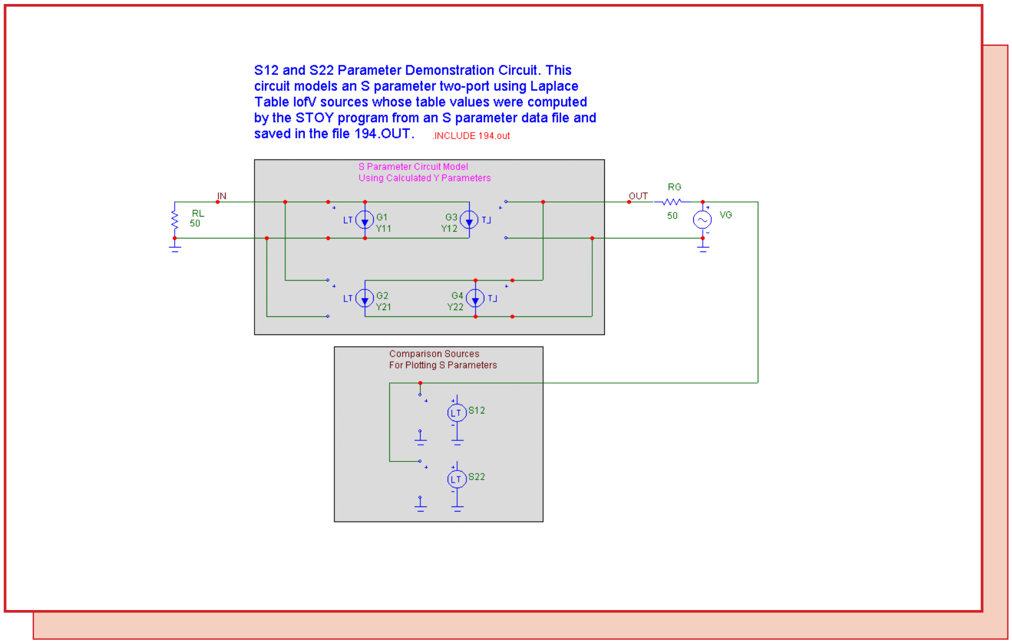


Fig. 20- The Y-parameter equivalent circuit set up for measuring S22 and S12

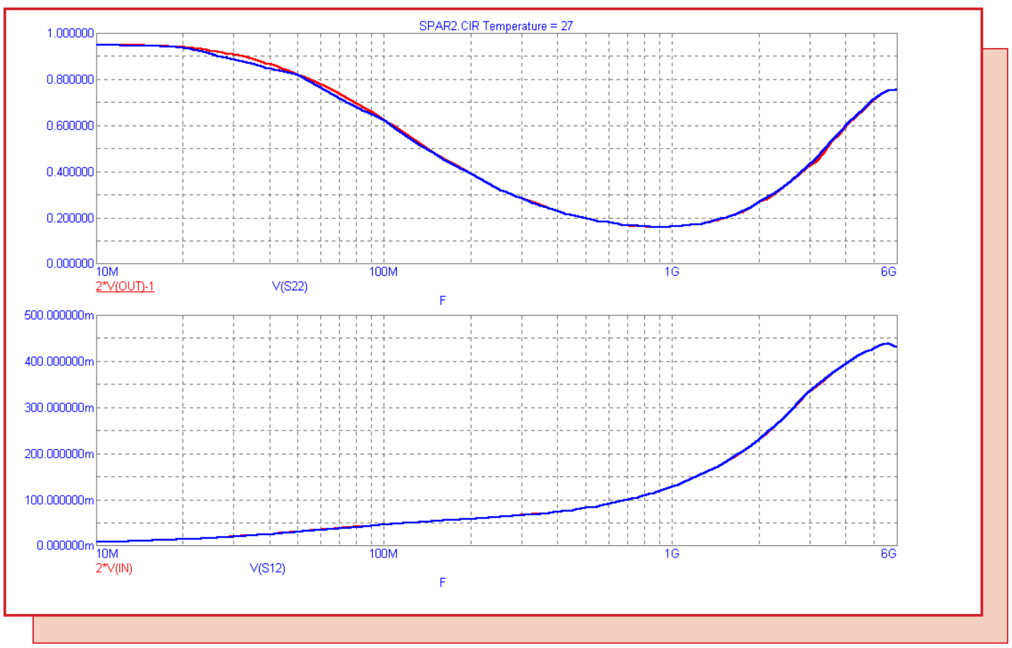


Fig. 21- Measured and original S22 and S12 parameters

Figure 21 shows a plot of the measured S22 and S12 parameters, together with a comparison plot of the original S22 and S12 parameters.

In general, the fit between measured and modeled values is exact at the original data points, whereas logarithmic interpolation used between data points produces some slight discrepancies.

Product Sheet

Latest Version numbers

Micro-Cap 6 .... Version 2.0.0
Micro-Cap V .... Version 2.1.2

Spectrum's numbers

Sales .... (408) 738-4387
Technical Support .... (408) 738-4389
FAX .... (408) 738-4702
Email sales .... sales@spectrum-soft.com
Email support .... support@spectrum-soft.com
Web Site .... http://www.spectrum-soft.com