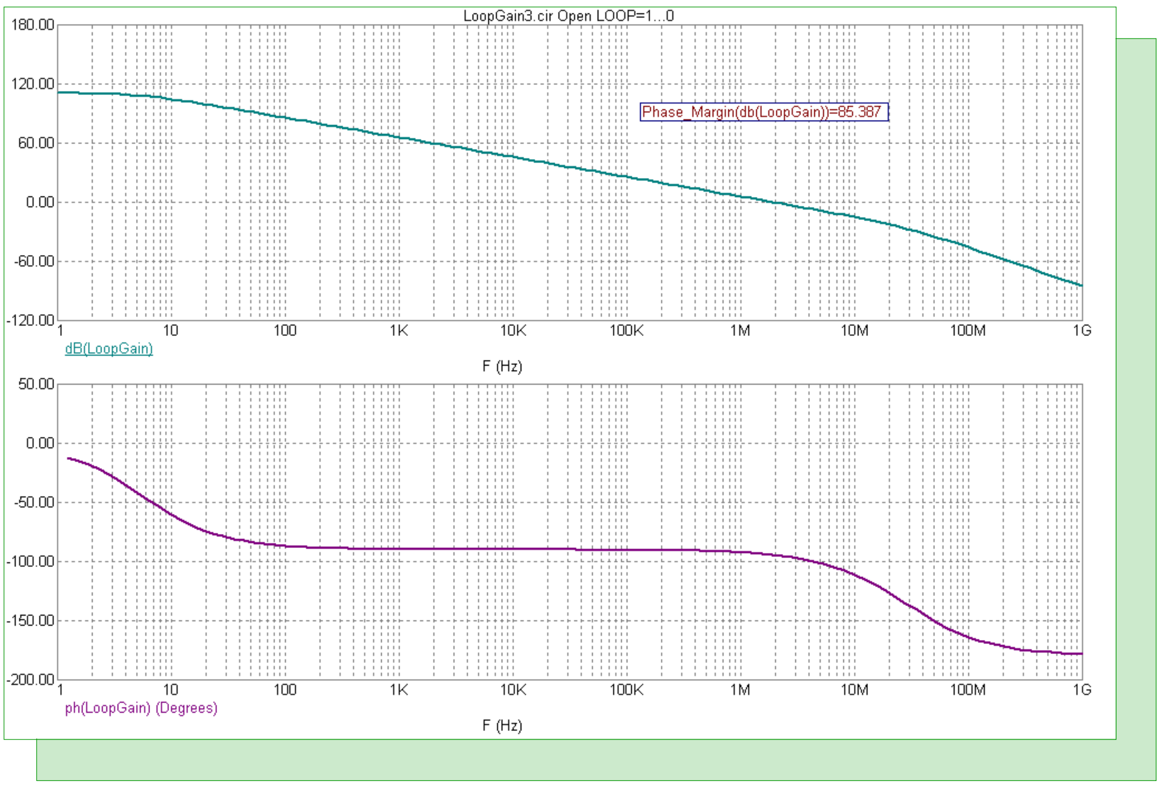
spectrum news

Applications for Micro-Cap™ Users

Spring 2011

News



Plotting Loop Gain

Featuring:

- Plotting Loop Gain Using the Tian Method
- Modeling Skin Effect in an AC Analysis
- Measuring Crest Factor

News In Preview

This newsletter's Q and A section describes the main factors that can be behind Permission denied errors. The Easily Overlooked Feature section describes the Align commands that can be used to align grid text, attribute text, and analysis text.

The first article describes using the Tian method to plot loop gain in AC analysis. The Tian method has an advantage over the Middlebrook method due to it taking into account bilateral feedback paths in its calculations.

The second article describes how to model skin effect in a lossy transmission line through the new capability that allows the use of F and S within the lossy transmission line model statement parameters.

The third article describes how to measure the crest factor of a waveform by using the performance functions available in Micro-Cap.

Contents

Book Recommendations

General 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

• *Inside SPICE-Overcoming the Obstacles of Circuit Simulation*, Ron Kielkowski, McGraw-Hill, 1993. ISBN# 0-07-911525-X

• *The SPICE Book,* Andrei Vladimirescu, John Wiley & Sons, Inc., 1994. ISBN# 0-471-60926-9

MOSFET Modeling

• *MOSFET Models for SPICE Simulation, William Liu, Including BSIM3v3 and BSIM4*, Wiley-Interscience, ISBN# 0-471-39697-4

Signal Integrity

• *Signal Integrity and Radiated Emission of High-Speed Digital Signals*, Spartaco Caniggia, Francescaromana Maradei, A John Wiley and Sons, Ltd, First Edition, 2008 ISBN# 978-0-470-51166-4

Micro-Cap - Czech

• *Resime Elektronicke Obvody,* Dalibor Biolek, BEN, First Edition, 2004. ISBN# 80-7300-125-X

Micro-Cap - German

• *Simulation elektronischer Schaltungen mit MICRO-CAP,* Joachim Vester, Verlag Vieweg+Teubner, First Edition, 2010. ISBN# 978-3-8348-0402-0

Micro-Cap - Finnish

• *Elektroniikkasimulaattori,* Timo Haiko, Werner Soderstrom Osakeyhtio, 2002. ISBN# 951-0-25672-2

Design

• *High Performance Audio Power Amplifiers,* Ben Duncan, Newnes, 1996. ISBN# 0-7506-2629-1

• *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

• *Modern Power Electronics,* Trzynadlowski, 1998. ISBN# 0-471-15303-6

Switched-Mode Power Supply Simulation

• *SMPS Simulation with SPICE 3,* Steven M. Sandler, McGraw Hill, 1997. ISBN# 0-07-913227-8

• *Switch-Mode Power Supplies Spice Simulations and Practical Designs*, Christophe Basso, McGraw-Hill 2008. This book describes many of the SMPS models supplied with Micro-Cap.

Micro-Cap Questions and Answers

Question: I installed Micro-Cap on my system. The program runs fine. However, when I exit the program, I receive error messages such as:

Permission denied 'WFB.BIN'.

What do I need to do about these errors?

Answer: The permission denied error occurs when Micro-Cap tries to perform an operation on the specified file, and the operating system states that the program does not have the proper permission to do that operation. There are two main causes of this.

1) The file or the folder that the file resides in is set to Read Only. If this option is enabled, the file is write protected so Micro-Cap is unable to modify the file. Any save operation will be prevented by the operating system.

In order to fix this, the Properties dialog box for the file or folder must be accessed through the Windows operating system. In the Properties dialog box, there should be an Attribute section that has a Read-only option. Disable this option. Depending on the configuration of your system, you may need Administrator privileges to do this.

2) Another program is already accessing the file and has placed a lock on it. In this case, you would need to find which program is accessing the file and then close the file or the program.

The reason that the Permission denied errors are occurring when the program is closing is that Micro-Cap will write to numerous files during the exit process. Micro-Cap stores the last used component and positioning information in all of the .CMP files that are loaded in the program. User preferences and settings are stored in the .DAT files, and the waveform buffer information in the WFB.BIN file is stored to the hard drive at this point.

All of these files are typically stored in the main Micro-Cap folder. If the Permission denied error occurs when exiting, make sure that the main Micro-Cap folder is not set to Read-only.

In Windows 7, Micro-Cap may also be setup to run as an administrator. This option can be enabled within the Compatibility tab of the Properties dialog box for the executable file. The Properties dialog box can be accessed by right clicking on the executable file in Windows Explorer and selecting Properties from the popup menu.

Easily Overlooked Features

This section is designed to highlight one or two features per issue that may be overlooked among all the capabilities of Micro-Cap.

Align commands
The align commands are available under the Edit menu. These lines can apply to grid text, attribute text, and analysis text. The group of commands consist of the following:

Align Left: This command aligns all selected text to the left hand edge of the control text.
Align Right: This command aligns all selected text to the right hand edge of the control text.
Align Top: This command aligns all selected text to the top edge of the control text.
Align Bottom: This command aligns all selected text to the bottom edge of the control text.

In order to use any of the Align commands, multiple text strings need to be selected. In the schematic, the strings can be a combination of attribute text and grid text.

The first text string to be selected will be the control text. The control text is denoted in the schematic and the analysis plot by a black rectangle that is displayed in the upper left corner of the text string's select region. In the figure below, the three model statements in the schematic have been selected. The N1 model statement is designated as the control text. The align commands will use this string to determine the alignment baselines.

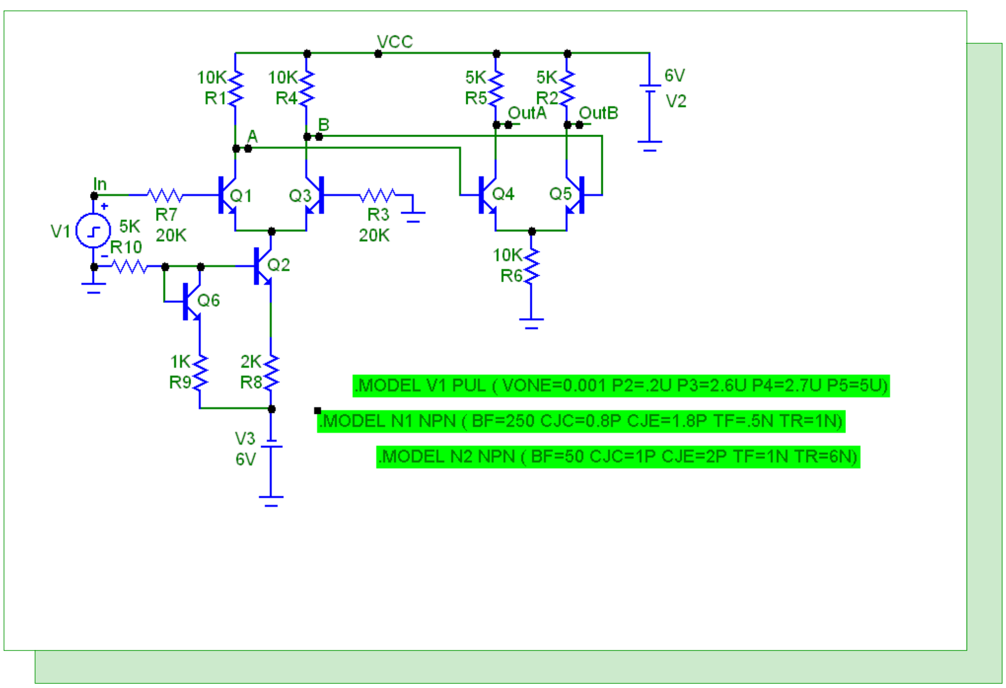


Fig. 1 - Showing the control text for the Align commands

Plotting Loop Gain Using the Tian Method

Loop gain simulations aid in determining the stability of a circuit by providing a method to measure the phase margin or the gain margin of a circuit. Previous Spectrum Software newsletter articles in the Winter 2001 and Spring 1997 issues have described methods for performing the loop gain calculation. This article will use the Tian method which is an improved technique for measuring loop gain that was originally described in Reference 1.

There are two crucial items that must be adhered to in any of these loop gain simulation methods. First, the DC operating point must not be affected by the method. Therefore, for the DC calculations, the loop must remain closed. Second, the user will need to find a critical wire that the circuit can be opened at to perform this measurement. This critical point must be located at a position where it would break all return loops in the circuit.

The Tian method uses a general two port analysis to calculate loop gain. There are at least three advantages to using the Tian method:

1) Less components need to be inserted into the circuit to calculate the loop gain.
2) The orientation of the components that make up the loop gain probe is not a factor. The orientation may be reversed and the computations will remain the same.
3) The other loop gain methods such as the Middlebrook method assume that the feedback path is unilateral so the measurement is made in only a single direction. The Tian method measures both the normal loop transmission and the reverse loop transmission. This is useful when simulating at higher frequencies such as with microwave and RF applications.

The circuit below was taken from Sedra and Smith and was also used as an example in the Winter 2001 issue.

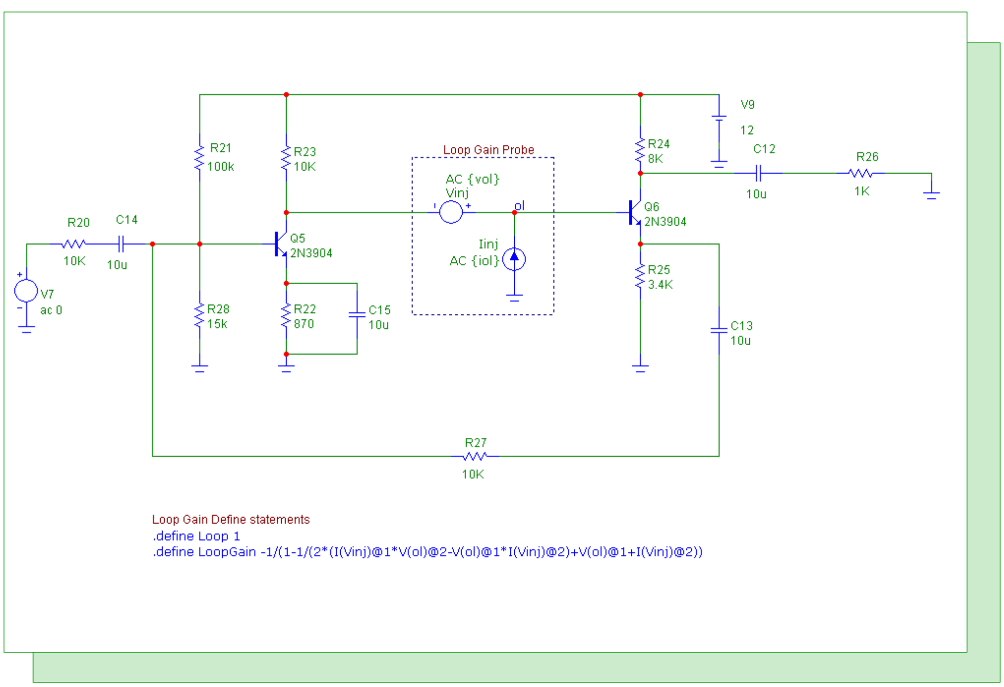


Fig. 2 - Sedra and Smith loop gain measurement circuit

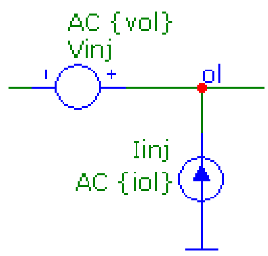
The following steps need to be taken in order to use the Tian method with the circuit.

1) Zero out the AC magnitude in existing sources
Any Voltage Source or Current Source components that are present in the circuit should have their AC Magnitude parameter set to 0 so that they create no signal during an AC analysis. In this example, the V7 voltage source is defined as:

AC 0

If there are any WAV File Source, Sine Source, or Pulse Source components in the schematic, these will need to be either disabled or deleted. The AC magnitude of these sources are fixed at 1V. Note that this only refers to the components called Sine Source and Pulse Source, and not the sine and pulse waveforms that can be defined within the Voltage Source and Current Source components.

2) Place the loop gain measurement components in the circuit
The loop gain measurement components then need to be placed at the critical point in the circuit where all return loops would be broken. The components consist of just a Voltage Source and a Current Source as shown below.



The Voltage Source should be placed in series with the loop and have its PART and VALUE attributes defined as:

PART = Vinj
VALUE = AC {vol}

The DC value of the source defaults to 0V, so during the DC operating point calculation it behaves like a short circuit and the loop remains closed. The Current Source should be placed at the plus side of the Voltage Source with its current flowing into the loop with the other side of the source connected to ground. The Current Source needs its PART and VALUE attributes defined as:

PART = Iinj
VALUE = AC {iol}

At the junction between these two sources, the node should be given the node name "ol".

3) Place the loop gain define statements in the circuit
The following two define statements need to be placed in either the schematic or one of the text pages.

```
.define Loop 1
.define LoopGain -1/(1-1/(2*(I(Vinj)@1*V(ol)@2-V(ol)@1*I(Vinj)@2)+V(ol)@1+I(Vinj)@2))
```

The Loop variable will be stepped in the AC analysis in order to calculate the loop gain. The equation in the LoopGain variable was derived by Frank Wiedmann from the Tian article. It uses the step selection operator, @, which is used to combine the waveform results from different steps. For example, the following expression:

V(ol)@2

specifies the voltage at node ol during the second step of the simulation. Should the part names of the sources or the node name at the source junction be different from those specified in the step above, then the LoopGain equation would need to be modified accordingly.

4) Place the If statement in the text page
The following If statement needs to be added to one of the text pages.

```
.if Loop==1
.define iol 0
.define vol 1
.elif Loop==2
.define iol 1
.define vol 0
.else
.define iol 0
.define vol 0
.endif
```

This If statement sets the values of the iol and vol variables depending on the value of the Loop variable. The iol and vol variables are used in the voltage and current source in the loop gain probe. When the Loop variable is set to 1, then iol will be 0 and vol will be 1. When the Loop variable is set to 2, then iol will be 1 and vol will be 0. All other values of Loop will set both iol and vol to 0.

For the Tian method, the AC analysis will go through two steps. In the first step, the voltage source is active and the current source is turned off which corresponds to when the Loop variable is set to 1. In the second step, the voltage source is turned off and the current source is active which corresponds to when the Loop variable is set to 2.

When running any other type of simulation, simply set the Loop variable to 0 in the define statement within the schematic to turn off both of these sources.

5) Setup the AC analysis and stepping of the Loop variable
In the AC analysis limits, set the Frequency Range to the range that needs to be simulated. For the Y Expression fields, the following two waveforms can be plotted.

dB(LoopGain)
ph(LoopGain)

Both expressions use the equation set by the LoopGain define statement in the schematic. The first expression plots the loop gain response in decibels. The second expression plots the phase of the loop gain response in degrees.

Next, the stepping of the Loop variable needs to be setup. In the Stepping dialog box, the symbolic variable Loop should be stepped between the values of 1 and 2. The easiest method to step the Loop variable in this manner is with the List method. Select the List option in the Method section. Set the List field to:

1,2

This will set the voltage and current sources to their appropriate values as specified within the If statement for each step of the simulation. It is important that the Loop variable value be stepped in this specific order as the LoopGain equation has been defined based on this order. The complete settings for the stepping of the Loop variable appear below.

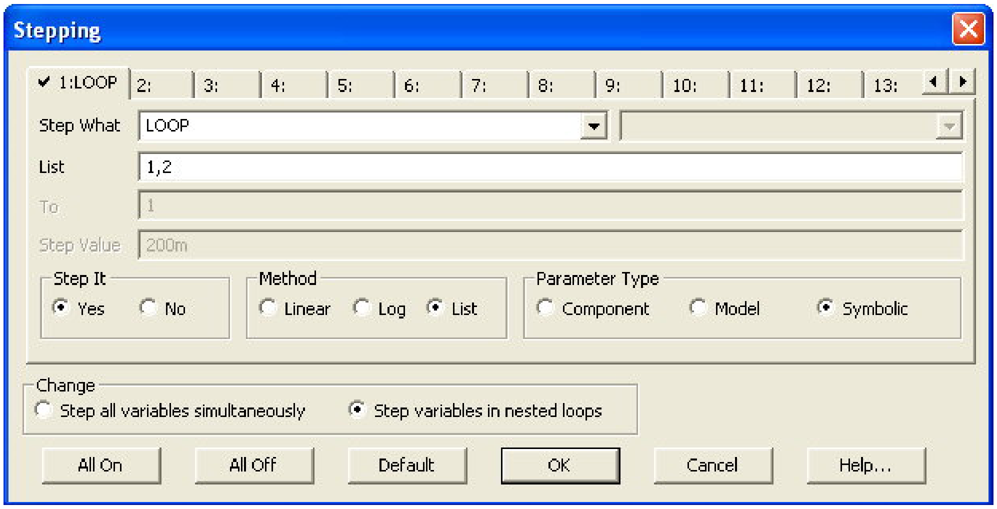


Fig. 3 - Stepping dialog box settings for the Loop variable

The AC analysis results for the Sedra and Smith circuit are shown in the figure below. A performance tag has been added to the top plot to display the phase margin of the simulation. The performance tag has been defined with the expression:

Phase_Margin(dB(LoopGain))

In order to use the Phase_Margin performance function, both the dB and phase waveforms of the expression LoopGain must be plotted. The resulting phase margin of this simulation is calculated at 89.063 degrees so the circuit is safely within the stability requirement.

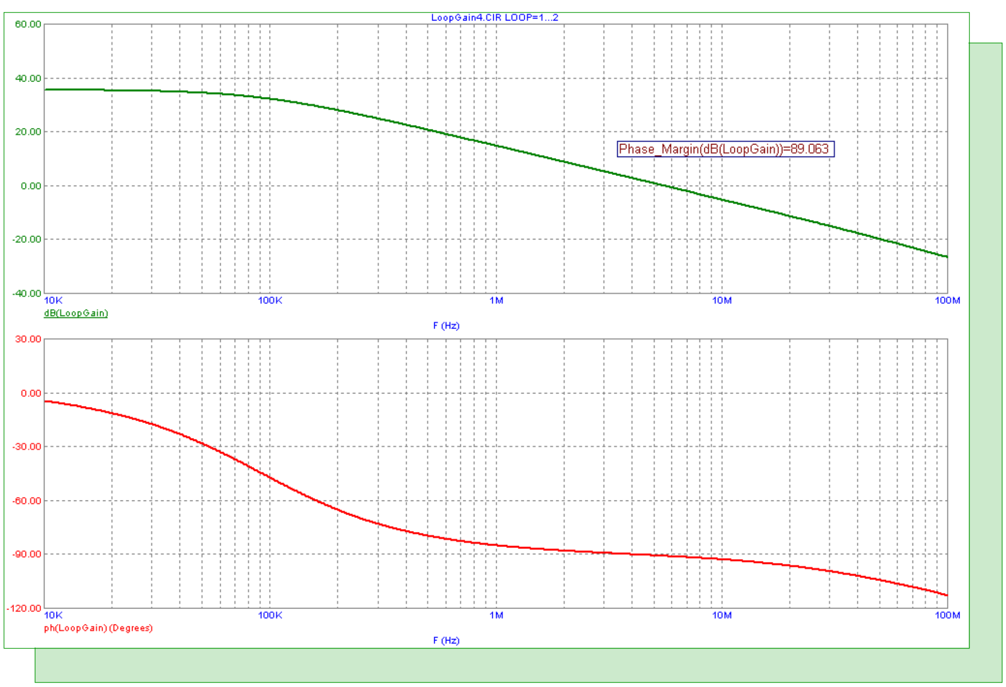


Fig. 4 - Sedra and Smith circuit loop gain analysis

A second loop gain example is shown in the figure below. This circuit is measuring the loop gain response of an OP_27 low noise precision opamp.

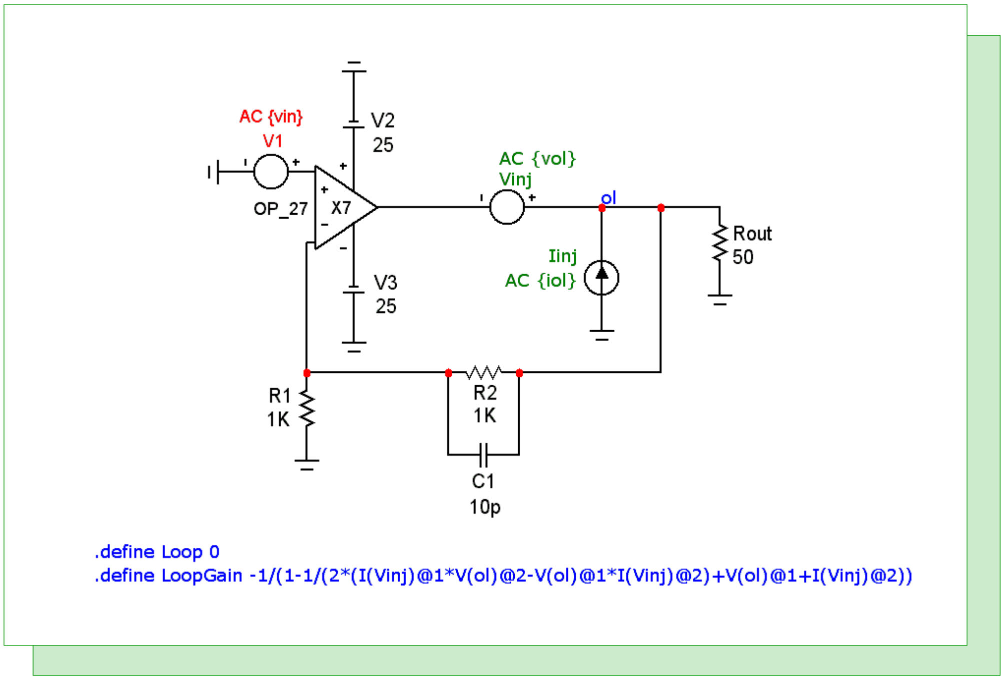


Fig. 5 - Loop gain example using the Tian method

There have been a couple of modifications from the Tian method described in the first example. First, rather than having the input source to the circuit hardcoded as a zero volt AC stimulus, the V1 voltage source instead has its VALUE attribute defined as:

AC {vin}

Then the If statement in the text area has been adapted to include the vin variable used above as follows:

```
.if Loop==1
.define vin 0
.define iol 0
.define vol 1
.elif Loop==2
.define vin 0
.define iol 1
.define vol 0
.else
.define vin 1
.define iol 0
.define vol 0
.endif
```

The If statement now also sets the AC magnitude of the input source. For the loop gain calculations (when Loop is equal to 1 or 2), the vin variable is set to 0. For any other value of Loop, the vin variable is set to 1.

The advantage to this modification is that the user no longer needs to manually edit the V1 source every time they go between an open loop simulation and a closed loop AC simulation. In this case, the Loop define statement in the schematic is as follows:

.define Loop 0

Since the Tian method needs to have the Loop variable stepped which will override the value set in the schematic, simply enabling or disabling the stepping of the Loop variable will now switch between the open loop and closed loop simulations.

In fact, with this setup, it is possible to plot both the open and closed loop response in the same simulation. In the Stepping dialog box, the List field for the Loop variable is set to:

1,2,0

The first two values set the Loop variable to 1 and 2 for the first two steps which will perform the loop gain simulation as described with the first circuit. The addition of the third value of 0 for Loop lets us plot the closed loop simulation too. Again, the order of the above values is required to get the correct plots.

The resulting open loop simulation of the OP_27 circuit is shown below. The two waveforms plotted are:

dB(LoopGain)
ph(LoopGain)

A performance tag that displays the phase margin is in the top plot. The phase margin has been calculated at 85.387 degrees which is safely within the stability requirement.

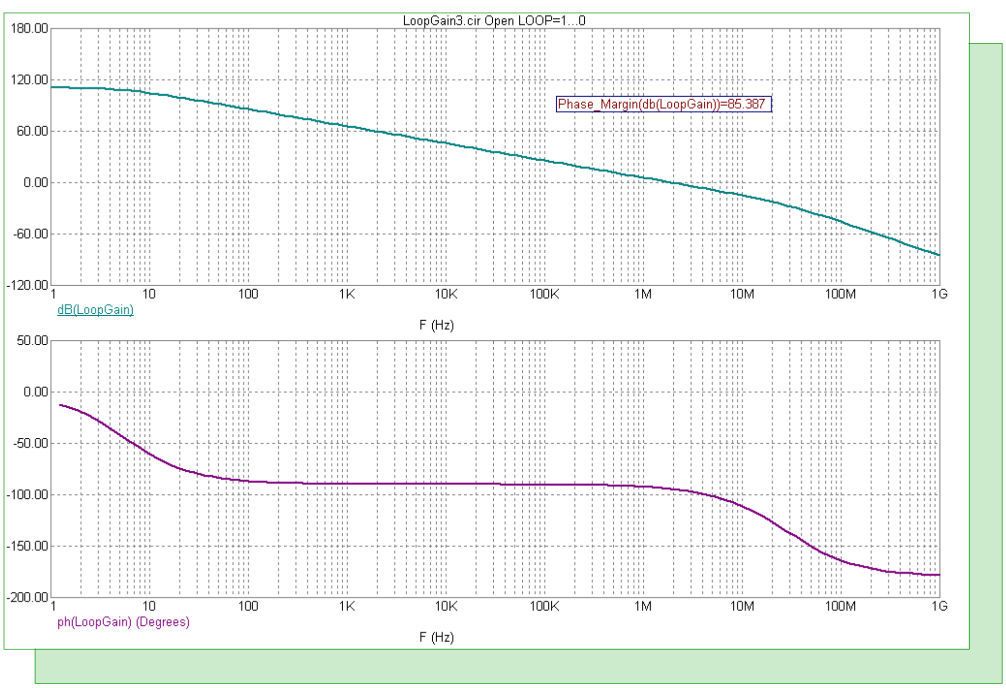


Fig. 6 - Open loop response of the OP_27 circuit

On a separate page of the AC analysis, the closed loop response is plotted using the following expressions:

dB(v(ol))@3
ph(v(ol))@3

These expressions plot the magnitude in decibels and the phase of the voltage at node ol. The step selection operator is set to plot only these waveforms from the third step of the simulation. Since the Loop variable is set to 0 for the third step, the results display the closed loop simulation. The closed loop simulation is shown in the figure below.

Fig. 7 - Closed loop response of the OP_27 circuit

Reference:
1) "Striving for Small-Signal Stability", Michael Tian, V. Visvanathan, Jeffrey Hantgan, and Kenneth Kundert, Circuits & Devices, January 2001.

Modeling Skin Effect in an AC Analysis

The lossy transmission line component can now use the F and S variables directly in the model statement of the device. Any of the four impedance parameters (R, L, C, and G) can use these variables in order to define a frequency dependent expression for an AC analysis simulation. This capability provides a simple method to model transmission line characteristics such as skin effect.

Skin effect is the phenomenon where the apparent resistance of a wire increases as the frequency increases. At DC, the charge carriers have an even distribution throughout the area of the wire. However, as the frequency increases, the magnetic field near the center of the wire increases the local reactance. The charge carriers subsequently move towards the edge of the wire, decreasing the effective area and increasing the apparent resistance.

The area through which the charge carriers flow is referred to as the skin depth. The skin depth is frequency dependent, and we will use it to calculate the AC resistance by implementing its transfer function in the R parameter of the lossy transmission line model. The first step is to compute the transfer function for the AC resistance. The AC and DC resistances are related through the effective area of the wire as follows. Refer to the figure below for a graphical representation of the variables used.

AC resistance / DC resistance = DC area / AC area

AC resistance = (DC area / AC area) * DC resistance

DC area = PI * R * R

AC area = (PI * R * R) - (PI * r * r)

r = R - e

e = (3.16e3 / (2 * PI)) * Sqrt(res / (f * km))

AC resistance = ((PI * R * R) / (PI * R * R - PI * (R - e)*(R - e))) * DC resistance

where e is the skin depth, res is the resistivity, f is the frequency, and km is the relative permeability. The AC resistance has now been reduced to depending on only the frequency variable and constants. The skin depth equation presented here is only valid for higher frequency analysis, so we will analyze it at 50kHz and higher.

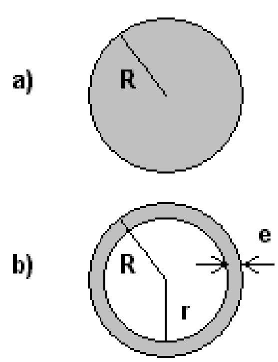


Fig. 8 - a) DC Representation of a Wire b) AC Representation of a Wire

The basic lossy transmission line model using the skin effect equation is as follows where Rad is the radius of the line, Res is the resistivity, km is the relative permeability, and DCRes is the DC resistance.

```
.MODEL TLSkin TRN (C=2n L=1.12n LEN=1
+ R={((PI*Rad**2)/((PI*Rad**2)-PI*(Rad-503.3*(sqrt(Res/(km*F))))**2))*DCRes})
```

As an example, a pair of 1 meter sections of AWG No. 22 wire will be simulated. Two transmission line models will be created, one line using copper material characteristics and the other line using aluminum material characteristics. The resulting models are as follows:

```
* Copper Line
.MODEL TLCopper TRN (C=2n L=1.12n LEN=1
+ R={((PI*.322m**2)/((PI*.322m**2)-PI*(.322m-503.3*(sqrt(1.75e-8/(1*F))))**2))*.0537})

* Aluminum Line
.MODEL TLAlum TRN (C=2n L=1.12n LEN=1
+ R={((PI*.322m**2)/((PI*.322m**2)-PI*(.322m-503.3*(sqrt(2.83e-8/(1*F))))**2))*.0868})
```

The transmission lines have been setup in the configuration shown below with source and load resistances of 50 ohms.

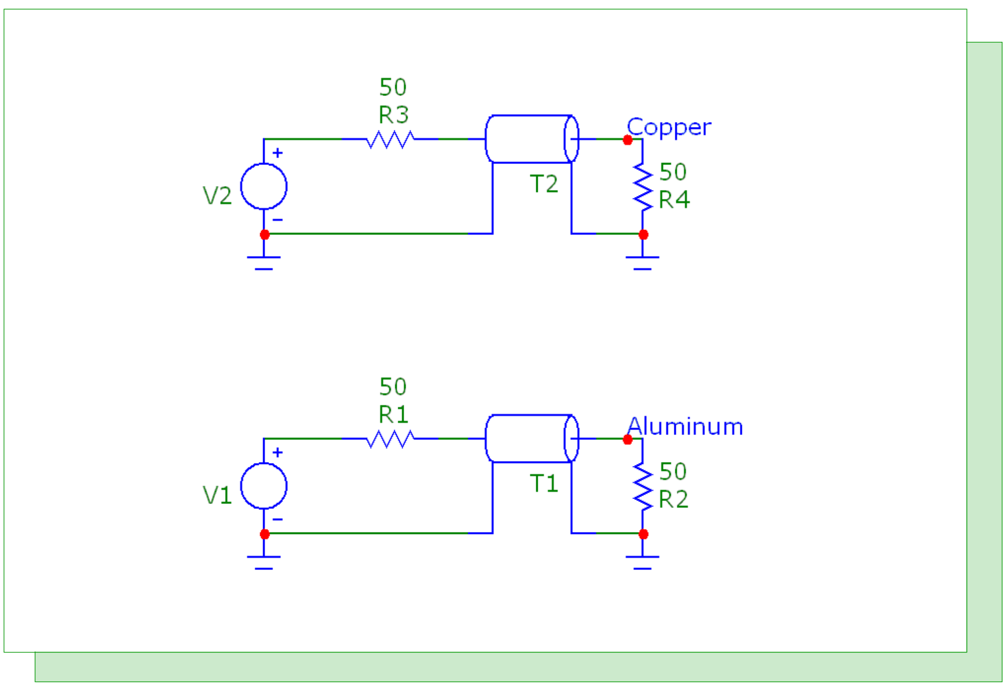


Fig. 9 - Skin effect example circuit

The resulting AC analysis is shown in Figure 10. The top plot displays the output voltage of each tranmission line in decibels. As can be seen, at higher frequencies the aluminum transmission line produces a higher AC resistance and a greater loss.

The bottom plot displays the actual resistance value being used for each transmission line in the simulation. The T1.R expression plots the R parameter for the T1 component which is the aluminum line. The T2.R does the same for the copper line.

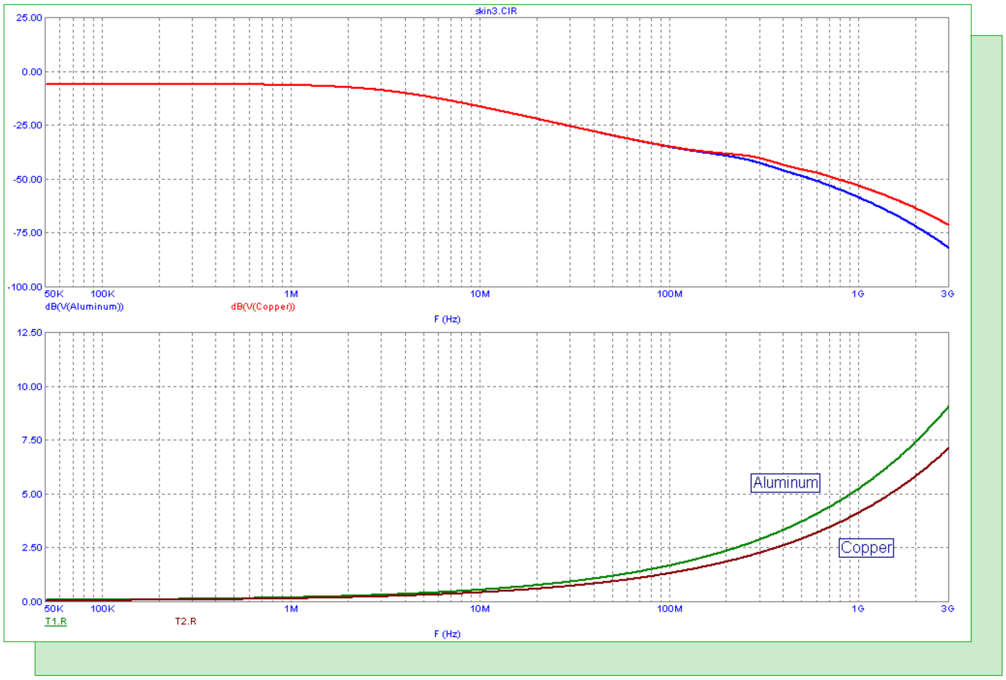


Fig. 10 - Skin effect AC analysis

The use of the F and S parameters in the transmission line is only meaningful in an AC analysis. For any other analysis, these variables are set to a value of 0. For the skin effect model, that would produce a divide by 0 problem. In this case, the transmission line models can be set through the following If statement.

```
.if analysis==_AC
.MODEL TLCopper TRN (C=2n L=1.12n LEN=1
+ R={((PI*.322m**2)/((PI*.322m**2)-PI*(.322m-503.3*(sqrt(1.75e-8/(1*F))))**2))*.0537})
.MODEL TLAlum TRN (C=2n L=1.12n LEN=1
+ R={((PI*.322m**2)/((PI*.322m**2)-PI*(.322m-503.3*(sqrt(2.83e-8/(1*F))))**2))*.0868})
.else
.MODEL TLCopper TRN (C=2n L=1.12n LEN=1
+ R={((PI*.322m**2)/((PI*.322m**2)-PI*(.322m-503.3*(sqrt(1.75e-8/(1*Freq))))**2))*.0537})
.MODEL TLAlum TRN (C=2n L=1.12n LEN=1
+ R={((PI*.322m**2)/((PI*.322m**2)-PI*(.322m-503.3*(sqrt(2.83e-8/(1*Freq))))**2))*.0868})
.endif
```

If the analysis type is AC, then the first pair of models is used. For any other analysis, the second pair of models is used. The only difference between the two sets is that in the second pair the F variable has been replaced with a variable called Freq. The Freq variable would need to be set through a define statement such as:

```
.define Freq 1Meg
```

In this case, the resistance value of the transmission lines when operating at 1MHz would be used in the other analyses.

Measuring Crest Factor

The crest factor measurement, also known as the peak-to-RMS-ratio or peak factor, provides a method to measure the quality of a signal. It is a ratio of the peak value of a waveform to the RMS value of the same waveform. It is used in a variety of engineering areas such as determining the quality of an AC power waveform, determining the headroom in an audio system, and measuring how much impacting is occurring in a mechanical system.

The crest factor can be calculated easily in Micro-Cap by using the performance functions. While there is not a single performance function that will measure the crest factor directly, an expression that consists of the High_Y and RMS performance functions can make the calculation. The following performance function expression returns the crest factor:

High_Y(Expr,1)/RMS(Expr,1,TSTART,TMAX)

where Expr is the waveform that the measurement is being made on. The High_Y performance function returns the maximum value of the specified waveform which provides the peak value. The RMS performance function returns the RMS value of the specified waveform over the requested interval. In this case, the RMS is operating over the entire transient simulation due to the use of the keywords TSTART and TMAX for the range.

This crest factor expression can be used within performance tags, analysis text, performance plots, and 3D plots. Performance plots and 3D plots are only available if stepping is used. For a single waveform, either the performance tag or the analysis text needs to be used.

To test this expression, a simple circuit has been created. Two voltage source components are in the schematic. One creates an ideal triangle waveform at a node called Triangle. The other voltage source component produces an ideal sine waveform at a node called Sine.

For this example, the crest factor measurement will be calculated within analysis text. The analysis text provides the capability to include the output of an expression within a text string. To include an expression when placing text into an analysis plot, enable the Formula option that is in the Text page of the Analysis Text dialog box. The delimiter field specifies the delimiters that will be used in order to declare the portion of the text string that is to be used as a formula. The default delimiters are the square brackets, [ ]. For example, the following string would be entered to calculate the crest factor for the triangle waveform.

Crest Factor = [High_Y(v(Triangle),1)/RMS(v(Triangle),1,TSTART,TMAX)]

When the Formula option is enabled with the square bracket delimiters for this string, the following portion of the string will be treated as a formula.

High_Y(v(Triangle),1)/RMS(v(Triangle),1,TSTART,TMAX)

The text displayed in the plot would combine the text string outside of the delimiters with the value calculated from the expression such as:

Crest Factor = 1.732

Each time an analysis is run, the formula within the analysis text will be recalculated and the text display will be updated.

The analysis in the figure below shows the transient simulation results with the triangle and sine waveforms being run over a single period. Each waveform has two crest factor measurements in its plot. The crest factor is displayed in both its standard ratio value and its value in decibels. The analysis text used in the triangle waveform plot is:

Crest Factor = [High_Y(v(Triangle),1)/RMS(v(Triangle),1,TSTART,TMAX)]
Crest Factor (dB) = [dB(High_Y(v(Triangle),1)/RMS(v(Triangle),1,TSTART,TMAX))]

The analysis text used in the sine waveform plot is:

Crest Factor = [High_Y(v(Sine),1)/RMS(v(Sine),1,TSTART,TMAX)]
Crest Factor (dB) = [dB(High_Y(v(Sine),1)/RMS(v(Sine),1,TSTART,TMAX))]

As can be seen in the figure, the crest factor calculations in the plot match exactly with the theoretical values for the types of waveforms simulated. The triangle waveform has a crest factor of 1.732 (4.771dB), and the sine waveform has a crest factor of 1.414 (3.01dB).

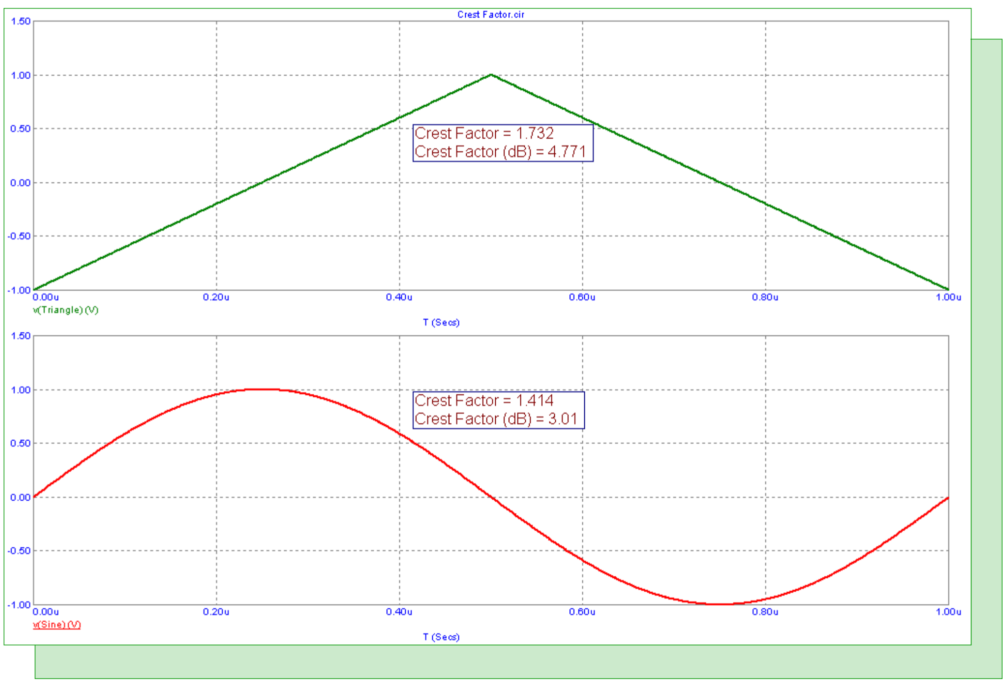


Fig. 11 - Crest factor measurements

Product Sheet

Latest Version numbers

Micro-Cap 10 .... Version 10.0.4
Micro-Cap 9 .... Version 9.0.8
Micro-Cap 8 .... Version 8.1.3
Micro-Cap 7 .... Version 7.2.4

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
User Group .... micro-cap-subscribe@yahoogroups.com