

Using IsSPICE and MathCad

Author: Paul McGinnis, EIT
5151 McFadden Ave.
Huntington Beach CA 92649-1236
714-893-4163

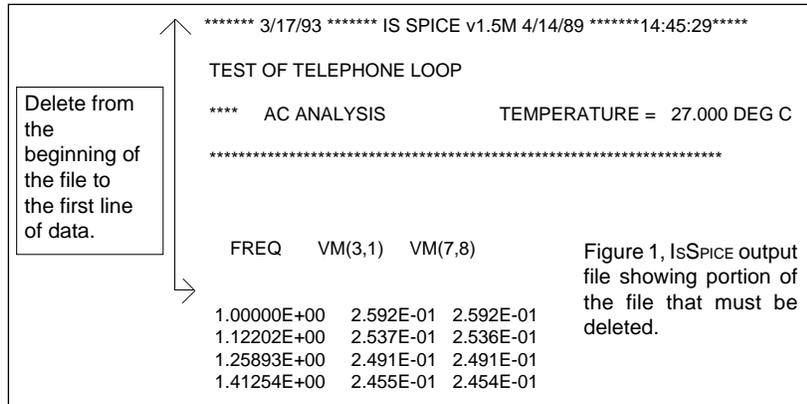
Many engineers use both SPICE and Mathcad (from Mathsoft). And while most SPICE vendors include a SPICE graphical post processing tool, engineers may be using the Mathcad environment for other parts of the design process. It is often desirable to be able to import SPICE output data into Mathcad so that all the results can be combined into one report.

SPICE is a circuit analysis program, originally developed at the University of California at Berkeley, that can simulate a wide variety of electronic applications. The SPICE program used in this case study is a popular reasonably priced version called IsSPICE and it is available from Intusoft. It is currently the only SPICE program to support all Macintosh and PC platforms. Mathcad is general purpose program used for calculating and graphing mathematical equations.

The process for including IsSPICE output in Mathcad are as follows:

- 1) Make a copy of the IsSPICE .OUT (output) file you are interested in importing into Mathcad.
- 2) Open the copy of the IsSPICE output file with a text editor. What you need to do is to strip off all of the text and leave just the tabular numerical data generated by the .PRINT statement. Note: you should not use the .PLOT statement as this will only generate line printer plots and not numerical data. Delete all lines up to and including the blank line before the numerical data starts. See Figure 1 for an illustration.

This would include all of the data under the IsSPICE banners such as: Input Listing, Element Node Table, Small Signal Bias Solution, and Operating Point Information. It will also include the banner, labeled AC Analysis, DC Analysis, Transient Analysis, Noise Analysis, or Distortion Analysis, depending on which analysis was run and the .PRINT column header names just above the tabular data.



You will also need to delete all material after the end of the numeric data, including blank lines. This includes the run times and any accounting options that may be listed. You need to make sure that the last piece of output data has either a single CR/LF (Carriage Return/Line Feed) if you are using a PC, or a single CR if you are using a Macintosh, at the end of the last line.

What you want is a table of numbers, with no extra text.

- 3) You will need to bring this table of data into Mathcad. The Mathcad function to do this is called READPRN. It is probably easiest to use something like:

A:=READPRN(infile)

and then use the "Associate Filename" menu option to associate the variable infile with an actual SPICE output file. This way, you can make a copy of a Mathcad file, and change the calculations performed. You can then read different data in to the new file by simply changing what infile is associated with it.

READPRN returns a matrix in Mathcad. For graphing or manipulation, you will need to convert the data to a vector. The sample Mathcad document in Figure 2 shows how this can be done. It also shows how the data is converted from units of voltage to units of dBm.

When you are using a vector from imported data, you need to set the range variable correctly for the number of lines of data in the file that was imported. If you don't do this, Mathcad will give you an "index out of bounds" error if the range variable is too large or truncate your data if it is too small. Many text editors will give you a line count, which is why I

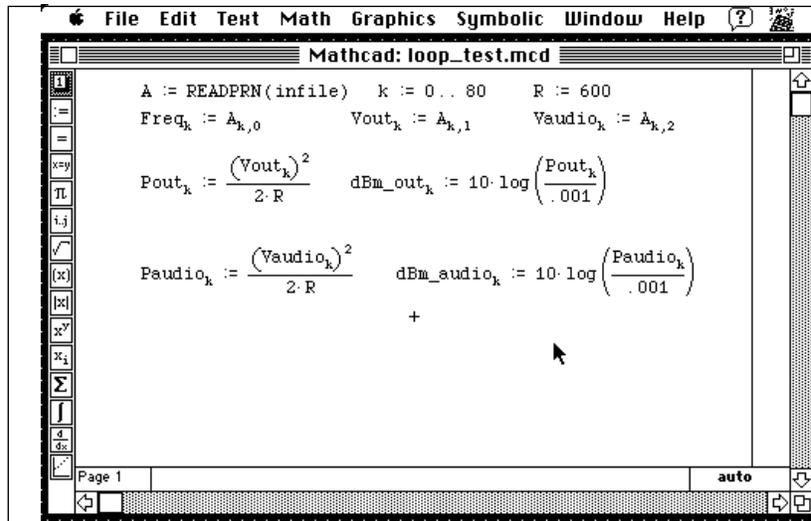


Figure 2, Mathcad document showing conversion of SPICE data into a vector. Calculation of output power is also shown.

recommend stripping off extra lines. In the example, IsSPICE produces a file with 81 lines of numerical data, so the range variable k is defined from 0 to 80, for 81 elements of data.

The following example shows the expected frequency response of a telephone loop at the point of connection to a phone line and across a loaded 1:1 isolation telephone transformer. The example is from a load test for the input to a DTMF (Dual Tone Multi Frequency or the more familiar name, Touch-Tone) detector. The schematic is shown in Figure 3. An AC analysis is run using the IsSPICE command .AC DEC 20 1 10K to produce the data. A .PRINT statement, .PRINT AC V(3,1) V(7,8) is used to produce the magnitude of the voltage difference in a tabular data format.

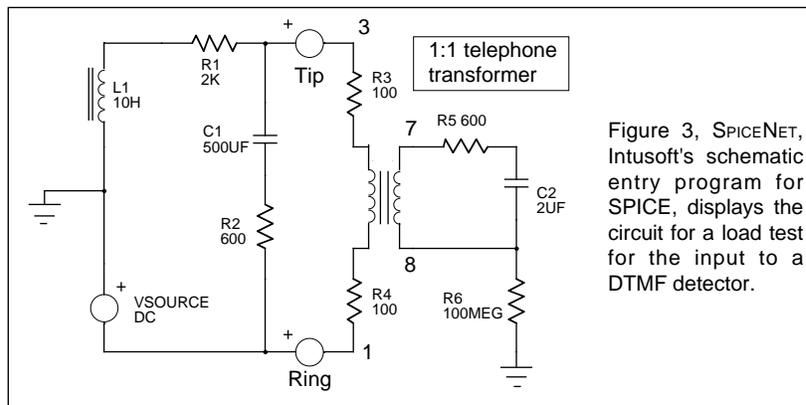


Figure 3, SPICENET, Intusoft's schematic entry program for SPICE, displays the circuit for a load test for the input to a DTMF detector.

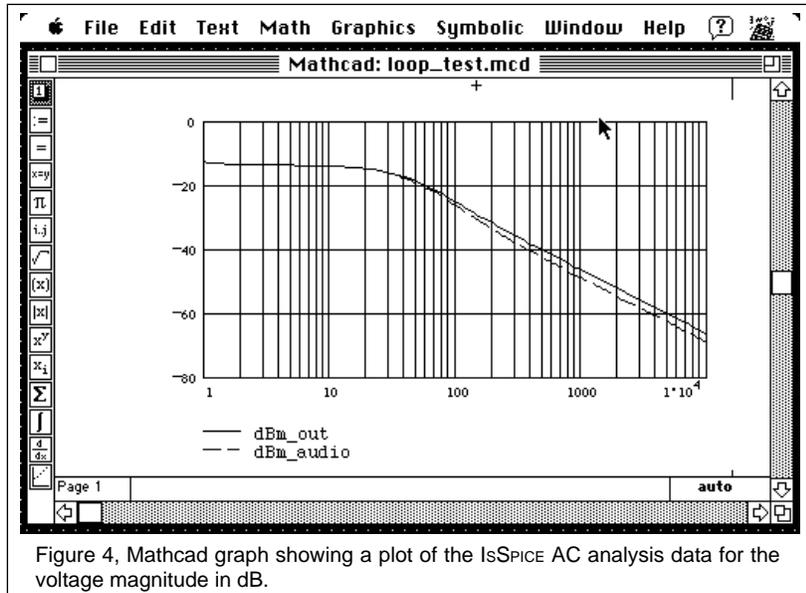


Figure 4, Mathcad graph showing a plot of the IsSPICE AC analysis data for the voltage magnitude in dB.

In telephone work, signal strengths are often measured in dBm (decibel milliwatts) across a load. The data is read into Mathcad and then converted from units of Voltage magnitude (output of the IsSPICE AC analysis) to units of dBm across a 600 ohm load. While IsSPICE can make the conversion to DB, this operation is performed with the Mathcad function shown in Figure 2 under dBm_out and dBm_audio. Figure 4 is a screen snapshot of the Mathcad graph of the IsSPICE simulation, in units of dBm.