

MEASURING AND PLOTTING THD VERSUS INPUT LEVEL IN LTSPICE

This application note shows how to measure and plot THD (Total Harmonic Distortion) versus input level in LTSpice. We will cover

- Installing the files
- An example measurement
- The meaning of the parameters
- About the algorithm and accuracy
- A few words about parameters and curly brackets

Installing the files

Download and unzip the THD.zip file into the same directory as the circuits you are working on. You should see these files after unzipping:

- test_thd.asc
- thd_pct.asc
- thd_pct.asy

An example measurement

Open test_thd.asc. It will look like this:

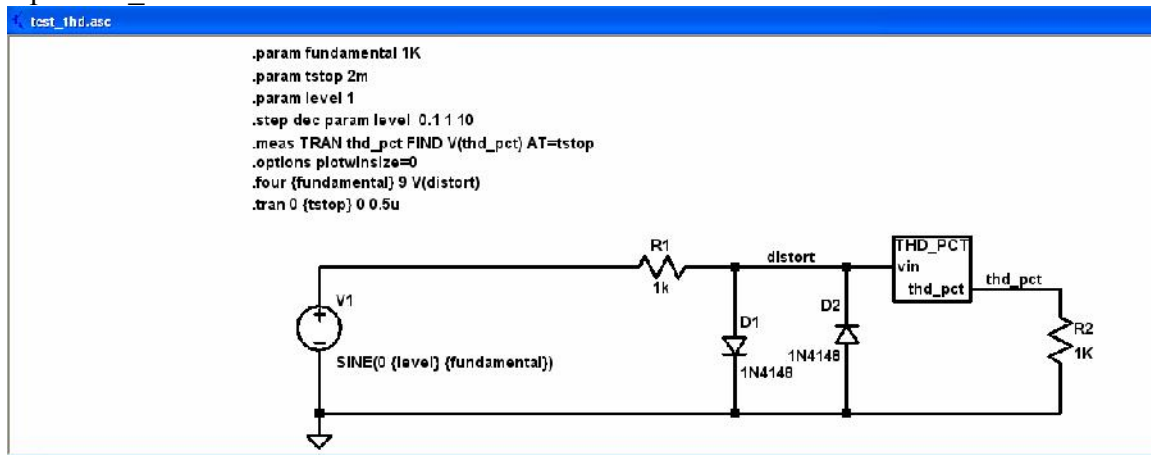


Figure 1 – example file

V1 is the sine wave source we'll use to test the circuit. In this example, R1, D1, and D2 generate some distortion that depends upon the level of V1. The THD_PCT sub-circuit generates an output voltage, thd_pct that is equal to the distortion when measured by the .meas statement.

Let's just run this test exactly like it is. Just click on the running man in the usual way to start the transient simulation. The .step statement causes 10 simulations to be run at 10 levels between 0.1 and 1 volt peak. When the simulation completes, you can probe any node in the usual way. For example, probe V(distort), where R1 meets the diodes. You'll see progressive flattening in the waveform as the level increases. Figure 2 shows that plot.

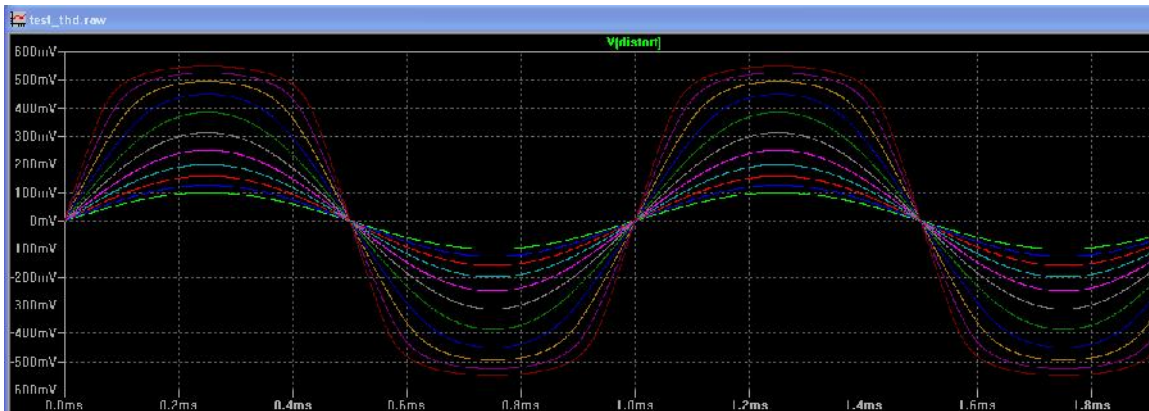


Figure 2 – showing the distortion at higher input levels where the diodes meet the resistor

To get a graph of the distortion versus the input level, do the following:

1. Open the Spice Error Log. There are two ways to do this:
 - a. Via menus – View=>Spice Error Log
 - b. Via short-cut keys=>hold down the Ctrl key while typing L
2. Move the cursor *over* the now opened Spice Error Log File. *Right click* the mouse, then *left click* the mouse to select “Plot .step’ed .meas data” as shown in Figure 3. A graph of THD versus level will be displayed, as shown in Figure 4.

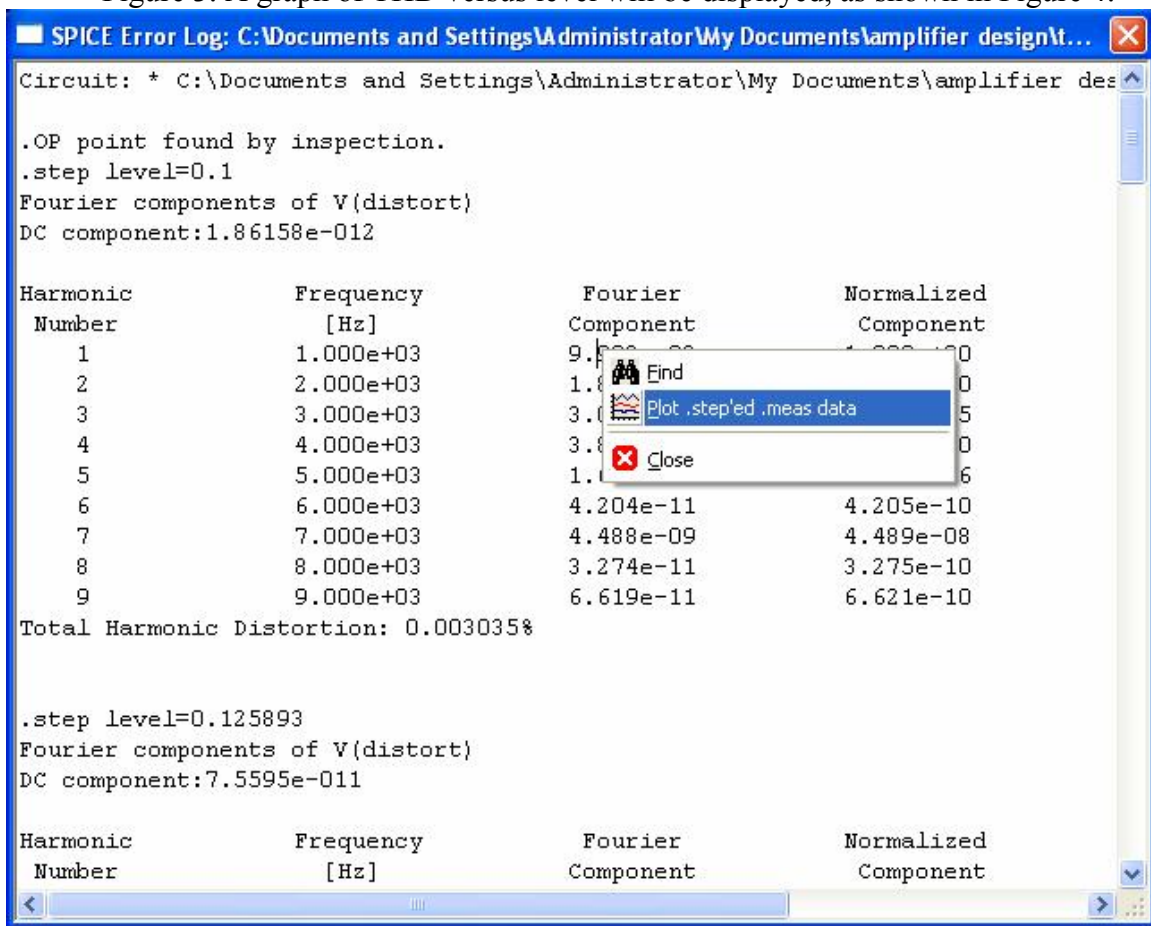


Figure 3- causing the plot to be made

Written by Daniel Joffe, Akitika LLC

© 2011, All Rights Reserved

Page 2 of 5

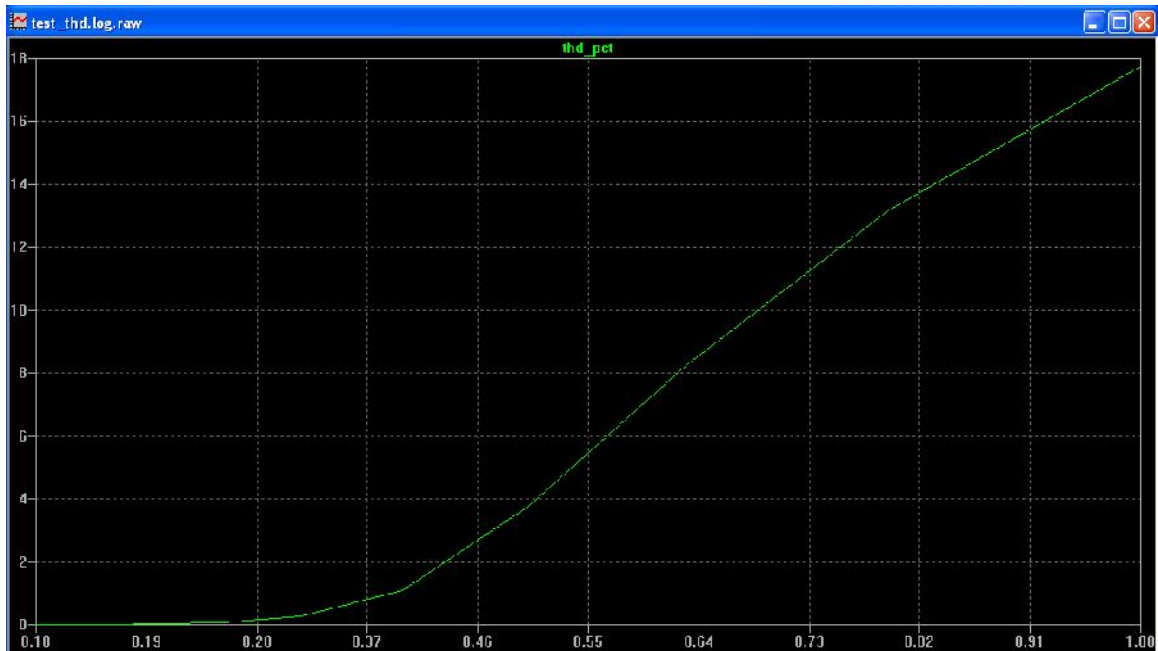


Figure 4 – THD in percent (vertical axis) versus input level (horizontal axis)

Here's the same plot, but I've changed both axes to log axes.

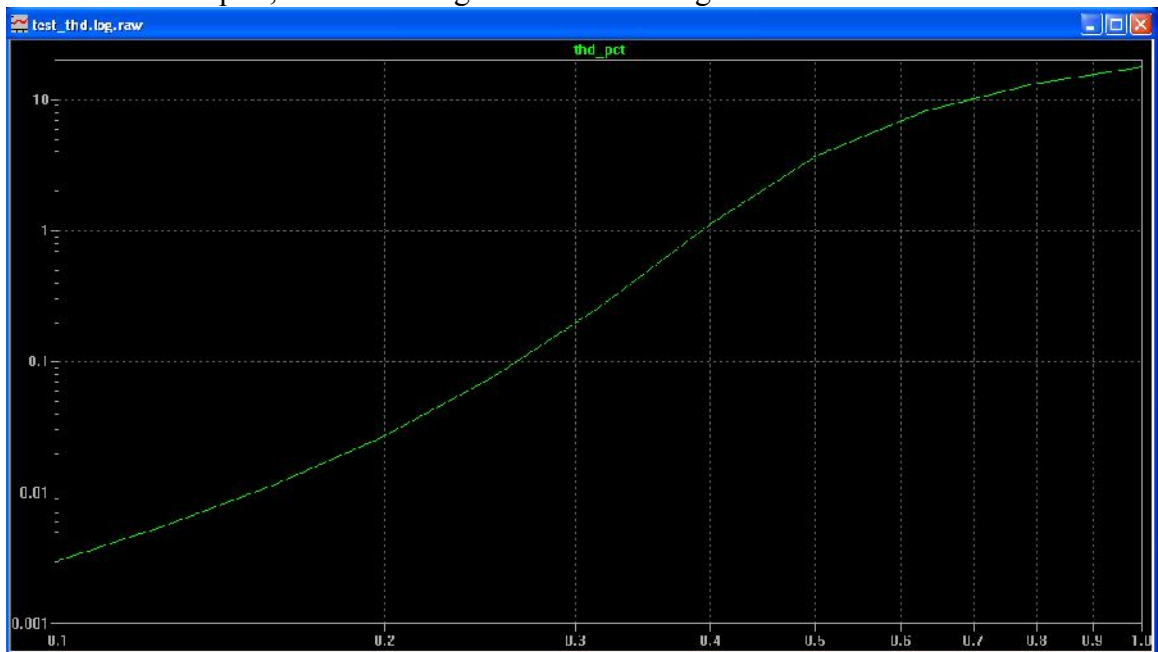


Figure 5 – same as Figure 4, but with log axes for both vertical and horizontal

The Meaning of the Parameters

Here's what the parameters mean, along with advice about how you can change them to suit your needs.

- `fundamental`- this is the frequency in Hz of the test source. In the example, I have made it 1K, but you can use whatever frequency you want.
- `tstop`- this is time where the simulation ends. It must be longer than 2 periods of the fundamental. For example, if fundamental is 1000, then `tstop` must be greater than 2/1000, or 0.002 seconds. It may need to be even longer than this if the amplifier under test should be allowed to settle before the distortion measurement is made.
- `level`- the peak level of the sine wave input. In the test circuit, we sweep this with a `.step` command.

About the other Commands

Here's a short description of the other commands:

- `.step dec param level 0.1 1 10` means that the simulator steps the level parameter in a decade wise fashion, using 10 steps in the decade between 0.1 Volts and 1.0 Volts.
- `.meas TRAN thd_pct FIND V(thd_pct) AT=tstop` measures the output voltage of the THD_PCT block at the correct time, the end of the simulation, and prints it into the Spice Error Log file.
- `.options plotwinsize=0` stops data compression in the plotting algorithm from compromising accuracy.¹
- `.four {fundamental} 9 V(distort)` displays the equivalent result of the classic Spice fourier calculation for this THD measurement, also in the Spice Error Log File. If the results of `thd_pct` and those reported by `.four` differ widely, then perhaps the maximum timestep is too large.
- `.tran 0 {tstop} 0 0.5u` specifies a transient analysis, stopping at `t stop`, with a maximum step size of 0.5 micro seconds. Having the step size about 1/2000th of the period of the fundamental seems to give pretty good accuracy.

About the Algorithm and Accuracy

The algorithm finds the strengths of the fundamental and harmonics 2 through 9 of the input signal by evaluating the coefficients of the Fourier series. To aid the accuracy of the calculation, we remove any DC component of the input. We do this as follows:

- We measure the DC component of the input in the time between two periods before the end of the simulation, and one period before the end of the simulation.

¹ See <http://audioperfection.deal-times.com/spice-ltspice/distortion-measurements-with-ltspice.html> for more details.

- The DC value, so measured, is subtracted from the input signal. That DC cancelled signal has its Fourier coefficients calculated in the last period (e.g. 1/fundamental seconds) before the end of the simulation.

This DC cancellation is what requires the simulation to run for at least two periods of the fundamental frequency. If the amplifier circuit is settling during the transient run, the DC component won't be steady between the cancellation and measurement periods, leading to an error in the reported THD. To be sure of the THD, run simulations using a nominal and a longer value of tstop. The sims are probably trustworthy if both sims report substantially the same number.

Similarly, you may want to diminish the maximum step size to assure that the results of the simulation don't change. These kinds of sanity checks aren't unique to the THD_PCT block, but rather are a usual feature of all Spice simulations.

We compared the results of THD_PCT to .four, and found that so long as the time steps were reasonable, the two were in close agreement.

About Curly Brackets

Study the example carefully to see where curly brackets {} are called for to insert a parameter. In this example, curly brackets are used to:

- Insert a parameterized value for tstop into the .tran statement.
- Insert a parameterized value of level into the amplitude of the sine wave source
- Insert a parameterized value of frequency into the sine wave source frequency

Acknowledgements

Mike Engelhardt was kind enough to answer a question about LTSpice and the plotting of .step data from the log file. That clue made this application note possible.