

Combining Experimental Measurements and Simulation within a Software Environment for DC/DC Converters

Peng Li and Brad Lehman*
Dept. Elect. & Comp. Eng.
Northeastern University
Boston, MA 02115

Ed Chan and Fei Ma
International Power Devices/Power-One
20 Linden Street
Boston, MA 02134

Abstract: This paper demonstrates that it is possible to combine experimental measurements with theoretical predictions within a unified software environment. Nominal experimental measurements are taken, and data is imported into MATLAB. Theoretical formulas are then used to predict changes in nominal performance under different load and line conditions. A specific case study is presented where performance is predicted when a capacitor bank is placed across a DC/DC converter.

1. INTRODUCTION

Performance of power electronic systems is often described through numeric simulation. Packages such as SPICE, Saber, and other circuit emulator tools attempt to characterize steady state and dynamic performance of circuit topologies. These packages allow designers to predict behavior of complicated circuits. Loads and voltages can be arbitrarily changed though simple changes in the circuit program and performance under different conditions can be judged. Painstaking effort has been placed in modeling the individual components, such as diodes, switches, capacitors, transistors, etc., and SPICE/Saber codes can be downloaded from a company's web site for thousands of components. The ease and flexibility of software design has made simulation a vital step before a final power electronic product is developed and put to market.

As helpful as these circuit emulation packages are to designers and engineers, there are also major shortfalls. Specifically, accuracy of the simulation package is often suspected, and sometimes it is not known until the product is built whether the predicted performance is accurate. To guarantee performance, experimental measurements are usually taken, and it is not uncommon for these measurements to differ from the simulated performance. In many cases, in fact, the simulation package is used to only guarantee the qualitative behavior of a system. The exact behavior is only known through experimentation. On the other hand, although experiments will give data that is more accurate, it is often time-consuming, difficult, and costly to measure performance under all possible circuit conditions

that are of interest. Table 1 presents a comparison of some issues between experimentation and simulation.

Usually, designers and engineers select either simulation or experimentation as a method to understand power electronic system behavior. That is, they normally select one or the other, depending on what information is desired. For example, to understand how a system performs with hundreds of different loads, an engineer will almost always be forced to simulate behavior. To characterize the exact nonlinear behavior, perhaps of switches, experimental measurements will probably be selected. There are clear capabilities of both methods, as well as clear limitations.

The purpose of this paper, however, is to show that it is not always necessary to make a choice between simulation and experimentation. The two methods can be combined within a fully integrated software package. Section 2 will discuss the possibility of combining experiments and simulations. A case study on DC/DC converter's small signal stability will be given in Section 3. Conclusions are given in Section 4.

2. COMBINE EXPERIMENT AND SIMULATION

One of the main advantages of simulation is that it is easy to manipulate the parameters and then to study the changes in the outputs. However, a commercial DC/DC converter has hundreds of components on each board, and each component has some influence in system performance. Often a designer must decide which components must be modeled and which should be neglected. For example, in small signal stability

TABLE 1. Comparison of Experiment Analysis and Simulation Analysis

Experiment Analysis	Simulation Analysis
Accurate results	Approximate results
Nonlinear effects included	Some nonlinear effects ignored
Unavailable in some cases	Manipulate parameters easily
Time-consuming and costly	Efficient and convenient

* This work is supported by the International Power Devices-Power One/ Northeastern University DC-DC Converter Research Center and by the National Science Foundation through Grant CMS 9596268.

simulation, the averaging method must be utilized. Averaging usually assumes ideal switching, and hence, there is some error between the simulation result and the real result. Furthermore, the influence of auxiliary circuits such as protection circuits, auxiliary power supplies, and opto-couplers are sometimes not modeled in small signal stability. An even more difficult problem is determining how to accurately model parasitics in transformers, such as leakage inductance or inter-winding capacitance.

However, in many cases, these components that are difficult to model/simulate do not vary much from module to module or as the operating point of the DC/DC converter changes. Additionally, we are not directly concerned with their exact behavior, but only on their influence on important system outputs. As a result, if we characterize their nominal behavior, we can use this information in multiple modules under different operating conditions. Alternatively, we can also characterize the nominal behavior of the entire module. This characterization will include the nominal behavior of all internal components, which is what we desire.

Specifically, it is possible to collect experimental data on the nominal performance of the power electronic system, and then use a software environment to predict performance changes in this nominal behavior for different load, line, temperature, etc. conditions. In essence, the experimentation is being used to capture the nonlinear circuit performance. Simulation is only being used to predict the changes in nominal performance and is not used to predict the nominal circuit behavior. In this way, it is possible to utilize simulation capabilities of varying circuit parameters while at the same time directly measure many of the circuit nonlinearities that can accurately be captured through experimentation.

Moreover, with such simulation, we even can ignore some model details, if such details have no impact on our results. This further simplifies the approach. For example, in the next section we introduce the problem of studying the influence of output capacitor bank on the small signal stability of DC/DC converters [1,2]. The frequency range of interest is near the crossover frequency, which is often around 5kHz – 50kHz, depending on the module. In this case, it is often reasonable to omit ESL influences in the capacitor bank, since these appear at high frequencies.

From a philosophical point of view, the “problem” being studied is different from the standard problem one thinks of when discussing the simulation vs. experimentation debate. The beginning assumption in this work is that at least part of a power electronic circuit has already been designed and built. Having the circuit already available allows the designer more flexibility in understanding behavior. As logical as this is, however, it is unusual for a designer to mix experimental data within a simulation environment, even though it should

be apparent of the potential increased accuracy that could be achieved.

3. CASE STUDY

In this section, we will show how it is possible to predict small signal changes from nominal behavior in DC/DC converters for different load and line conditions. Specifically, we propose the following steps:

Step 1: Take sample measurements of nominal small signal behavior.

Step 2: Import data measurements into software program.

Step 3: Manipulate experimental data in software by using theoretical formulas to predict changes in behavior for different load and line conditions.

Step 4: Display results.

In our work, we select MATLAB to use for data manipulation. Other programs such as MathCad, Maple, or Mathematica could as easily be selected. Alternatively, it is possible to write programs in C or Fortran. The important aspect of the software is that the user should be able to manipulate the data without too much effort. That is, experimental data files should be available to combine and manipulate, as the user desires. MATLAB provides a powerful simulation environment for data manipulation and graphical display of results, which is why it is utilized in this research.

CASE STUDY- LOOP GAIN FOR CAPACITIVE LOADS[1,2]

Here, we explain the software environment created to obtain the results in [2]. As [2] explains, telecommunication and datacommunication users often place an external capacitor bank across the output of commercial DC/DC converters to reduce the output peak deviation to load disturbances. This is particularly necessary for computer systems that involve large load step output current disturbances. However, in some cases the interaction between the capacitor bank and the DC/DC converter may cause performance degradation or even instability [1,2,3,4]. Specially, there can be a reduction in bandwidth, phase margin and gain margin.

Traditional methods [1,3] attempt to predict performance changes either by experimentation or by simulation. However, [2] proposes to combine small signal experimentation with simulation. First the output impedance, Z_o , and the outer loop gain, T , of the DC/DC converter is measured. Various load and line conditions are measured.

The goal is to determine the new loop gain when a capacitor bank, with impedance Z_c , is placed in parallel with the load. Let T' denote the new outer loop gain with the

added capacitor bank. The results of [2] derive the relationship

$$T' = \frac{T}{(1+T)Z_o/Z_c + 1} \quad (1)$$

Hence, a software program has been designed that calculates the loop gain for all possible values of Z_c . The program varies R_{ESR} and C and then uses the above formula and experimental measurements to predict the new phase margin, crossover frequency, and gain margin. For example, Fig. 1 shows a three-dimensional plot of phase margin versus R_{ESR} and C . Accuracy of this plot, and all others generated with similar methods, has been within 5% error. This is substantially better than those that we generated by purely numerical simulation, which have, at times, around 50% error.

How to Take Measurements

For our experiments, measurements should be frequency based, since (1) is based on a frequency response. In this case, the experimental data of the outer loop gain and output impedance is to be measured. All frequency measurements should be taken at the *same* and *known* frequency points. That is, the frequency measurements for output impedance should be at the same frequency points for the loop gain. This is because (1) requires point-by-point algebraic manipulation (multiplication, addition, etc.) of different measurements at the same frequency.

Data is stored in files and then imported into the simulation environment. Frequency response analyzer, Venable 260, is used in our experiments. For each frequency point measurement, Venable records the frequency point, magnitude, and phase.

There will be noise in each measurement. For example, the Bode plot of each loop gain is not a smooth curve if taken directly from experimental measurements. However, we assume white noise. Then the loop gain is simply measured several times and the average value is taken as the loop gain of such module. This averaging/filtering is implemented in the simulation environment. (At frequencies around 100 Hz, a huge input bank may be needed to get a stable loop gain value. However, we are interested in the values around 10 kHz and it will not influence our results.)

Impedance/Gain-Phase Analyzers, such as HP 4198A may also be used to obtain the data. Its output data has already been filtered, and then the filter in the simulation is not necessary. However, an interface program is needed to save the data in files.

It is important to note that outer loop gains vary among module-to-module for commercial power supplies, even for the same model number. There will always be parameter variance, e.g., the mismatch of the opto-couplers. Therefore, the above measurements need to be repeated on multiple modules so that entire sets of data are obtained. Different approaches to use the data sets include:

1. *Average Value* - Simply average the data of all the modules. Use this average value as the data used in simulation (approach used in this paper). This gives the averaged prediction of performance.
2. *Worst Case Module* - Perhaps the module with the least phase margin should be used. If the worst module is selected, then the results from simulation output will guarantee the stability of all the commercial modules.
3. *Typical Module* - It is possible to disregard the worst case modules and then average the remaining modules. In this way, it is possible to separate typical performance from worst case performance.

It is obvious that the above three values will approximately equal if the manufacturing process is controlled well.

Import Data into MATLAB

With the data format defined above, it is easy to use the file command in MATLAB to import the data:

```
%The name of the data file is loopgain.
%The codes below import the data into the
%simulation environment.
DataFile=fopen('loopgain','r');
TempBuffer=fscanf(DataFile,'%f',[5,172]);
fclose(DataFile);
```

Manipulation of Data

The data measured by Venable is the magnitude and the phase of a transfer function. Therefore, each frequency data point in TempBuffer is first converted to a complex number. Next, we compute the impedance of the capacitor bank for the experimental frequency range. If the capacitance C and the ESR of the capacitor bank is assumed known, its impedance is $Z_c(f) = ESR + 1/(i2\pi fC)$, where i is the imaginary part of complex value. Therefore, the derived formula will be utilized to calculate the output loop gain with capacitor bank.

```
%ImpedanceCap is Zc(f); other variables are just as
%their names mean.
LoopGainWithCap(f) = LoopGainWithoutCap(f) / ((1 + LoopGainWithoutCap(f)) * OutputImpedance(f) / ImpedanceCap(f) + 1);
```

Here LoopGainWithCap is the same as T' in (1). Computing the new phase margin is easily obtained by searching for when T' equals 1 in magnitude. The corresponding phase, θ , at this frequency is used to obtain the phase margin = $180 + \theta$. The frequency when $|T'| = 1$ is the crossover frequency.

Finally, we repeat the process for new values of C and ESR in the capacitor bank. This is achieved through using simple loops in MATLAB.

Display Results

MATLAB has numerous scientific visualization commands that permit three-dimensional color plots. Figure 1 shows the power of the method (the original figure is in color). As a matter of reference, we utilized the mesh command.

4. FINAL DISCUSSION

When experimental data is available for a power electronic system, it makes sense to use this data to improve modeling accuracy. This paper begins a discussion on how to combine experimentation and simulation. Although a case study is presented for "outer loop gains" similar case studies can be written for transfer functions such as audiosusceptibility, input impedance, output impedance, etc. The methodology is almost exactly the same.

There are many open challenges in the proposed technique. For example, often a wide operating range must be studied in

a nonlinear power electronic circuit. Performance can vary greatly when nonlinearities are included. In the above discussion, we have assumed that the converter remains close to linear operation (small signal). Future work will discuss how it is possible to define several operating conditions and then use a polynomial curve-fit prediction to estimate performance.

REFERENCE

- [1] A. F. Rozman, K. J. Fellboelter, "Circuit considerations for fast, sensitive, low-voltage loads in a distributed power system," *Proceedings of IEEE APEC*, 1995.
- [2] P. Li, B. Lehman, E. Chan, F. Ma, "Influence of capacitive load on DC/DC converters," to appear *IEEE INTELEC*, Phoenix, AZ, Sept. 2000.
- [3] C.M. Wildrick, F.C. Lee, "A method of defining the load impedance specification for a stable distributed power system," *IEEE Transaction on Power Electronics*, Vol. 10, No. 3, May 1995, pp. 280-285.
- [4] X. Feng, Z. Ye, K. Xing, F. C. Lee, D. Borojovic, "Individual load impedance specification for a stable DC distributed power system," *Proceedings of IEEE APEC*, 1999.

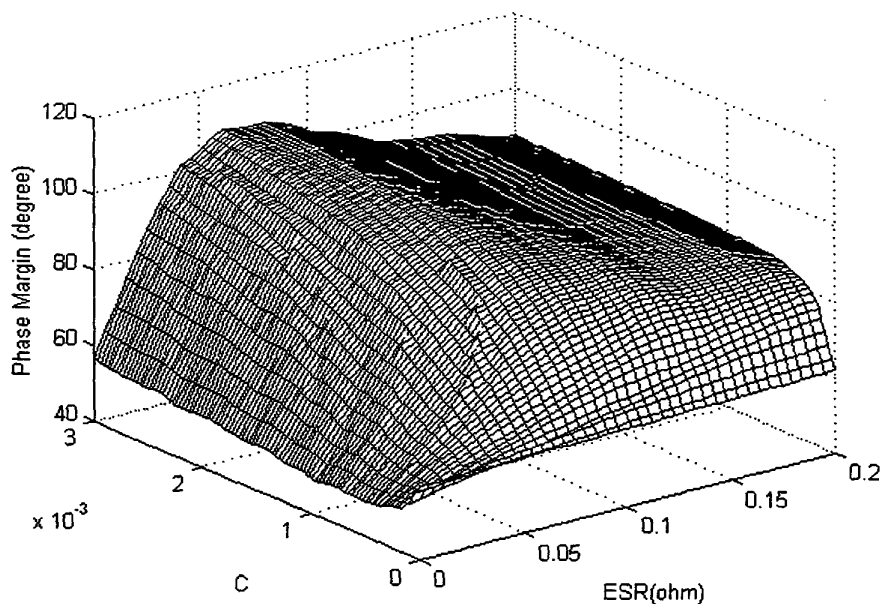


Fig. 1 Phase Margin with Capacitor across Load for a Commercial DC/DC Converter (International Power Devices/Power-One: HAS030ZGAN Forward Converter) [2]