Modelling Board Level DC-DC Converters in SPICE

Martin O'Hara
Newport Components Ltd, Tanners Drive,
Milton Keynes, England, MK14 5NA.

Abstract

Presented in this paper is a macro model which is applicable to any fixed frequency DC-DC converters and provides a fast and accurate simulation of the circuit behaviour at the device terminals.

The macro model avoids using an oscillator circuit and hence provides exceptionally fast simulation speeds (over 100 times faster than an equivalent oscillator in transient mode simulation) and works under DC analysis as well as providing a suitable noise characteristic under transient analysis. The simulated results are compared to measured results from a 1W board mounted DC-DC converter and provide exceptionally close correlation for the DC transfer curves of load regulation, line regulation and efficiency, as well as accurate noise spectra under transient simulation at the correct DC bias offset.

Introduction

Electrical simulation of circuit function is a common practice as part of the design cycle of individual circuits and complete systems. The more of the circuit a designer can simulate, the faster the circuit can get into production and hence to market. For analogue simulations SPICE (Simulation Program with Integrated Circuit Emphasis) has become a common tool and is widely used, even for some mixed mode circuit simulations and discrete circuit designs. However, as with most simulation tools, the results are only as good as the models contained within the simulator.

In SPICE models are used to describe the behaviour of individual components. In simple circuits these are accurate and relatively easy to implement. With more complex circuits, especially integrated circuits (IC's), the simulation can become very slow as each and every component is modelled and simulated. For example a bipolar transistor has 38 parameters to fully describe its behaviour, consider an IC with several hundred transistors such as a simple linear regulator. Any circuit using such a regulator immediately has a slow simulation time as the regulator function alone requires possibly as much computational time as the rest of the circuit. Consequently for complex integrated circuits a sub-set of the elementary component models have been derived which simulate the function without all the components explicitly being included, these are termed macro-models.

DC-DC Converter Macro-Model

DC-DC converters are notoriously difficult to simulate as component models in SPICE, due to the inherent instability of oscillator circuits in computer simulation. SPICE can all too often settle into a static mode where the circuit no longer oscillates, or takes so long to converge between time iterations as to make simulation too time consuming to be useful. For the designer of DC-DC converters these problems are difficult enough, for a user who is taking an off-the-shelf device and requires its simulated function as a part of a larger circuit these problems are intolerable. The problems of an oscillator circuit are further exacerbated in DC-DC converters when attempting to obtain a load circuit DC bias point, since under a DC simulation oscillators give no output.

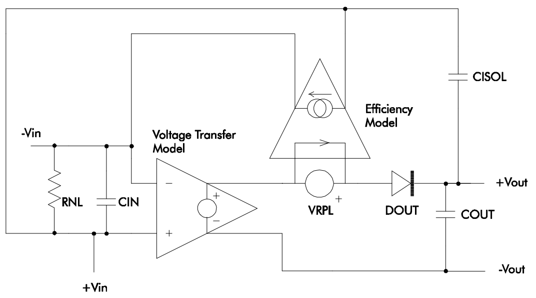
The decision to produce a macro-model for the DC-DC converter does not need further justification compared to the problems that would be encountered with a full component model. The characteristics of a DC-DC converter which describe the transfer characteristic of the device to be modelled need to be determined. In this paper an unregulated 1W single output board level DC-DC converter was chosen as an example of a relatively straight forward device (NME0505S). Operating from a 5V DC input and producing an isolated 5V DC output, the device features a fixed switching frequency of 100kHz, a load regulation better than 15% and a line regulation better than 1.1% for a 1% change in input voltage. Other characteristics to be modelled are the efficiency load curve (efficiency is a maximum with 1W load applied) and the ripple noise at the input and output terminals.

The required data is available from the DC-DC converter data sheet, hence, the model could be constructed without an intimate knowledge of the internal circuitry. This has the advantage of allowing the circuit designer to get an estimation of a model where none is available from the supplier.

Constructing the DC-DC Converter Macro Model

The aim of a macro model is to use stable components and elements to model the function of the target circuit. Hence the DC transfer characteristic for a DC-DC converter can be described using a voltage-controlled-voltage-source in SPICE, this avoids oscillator and transformer requirements and maintains isolation from input to output. Some modelling of the no load power consumption can be achieved at the input by simply using a resistor across the terminals to model the dissipation due to the oscillator operation.

Input and output terminals should appear electrically as they would in the actual component, this meant a low impedance input (provided by the no load resistor) and a DC blocking output. The output DC block is achieved in the real DC-DC converter by using a diode rectifier arrangement, since this is a stable element this was also used in the model. Another input and output feature are ripple capacitors, since the model may be used with additional external filtering, these needed to be included to ensure any potential poles and zeros are represented in this filtering.



The efficiency curve required feedback of the DC output current to sink at the input, otherwise the converter would be more than 100% efficient. This is the direct equivalent to the voltage-controlled-voltage-source used to provide the voltage transfer characteristic, hence a current-controlled-current-source was used. A model of the ripple would also be required, it was decided to add this simply as an AC voltage source at the output. The AC voltage source could also be used to measure the DC output current flow for the efficiency feedback function, hence provided two functions and the AC modulation would also feed back to the input as input ripple.

A final element for the isolation capacitance was included. This is to allow common mode effects to be modelled if required.

Values for the Model

Deciding the elements is only half way to completing the model, values needed to be determined for all of the elements and modelling terms. The input and output capacitors were chosen from the known input and output capacitor values (C_{OUT}=C_{IN}=1uF in the example being used here). This data is not always explicitly stated by the manufacturer, but can be determined from application notes on appropriate filter values if these are given. Isolation capacitance is in reality a measure of the capacitive coupling across the transformer, this is a very low value compared to most discrete capacitors (C_{ISOL}=24pF).

The no-load resistor (R_{NL}) value can be calculated knowing the no-load power consumption (P_{inNL}) or input current (I_{inNL}) at the nominal input

Install Equation Editor and double-click here to view equation.

voltage (V_{nom});
For the NME0505S, P_{inNL}=100mW and V_{nom}=5V hence R_{NL}=250Ω.

The voltage and current transfer characteristics are the transfer ratio of the internal transformer (the ratio is used directly for the voltage transfer and its reciprocal for the current function). If the transformer ratio is not known, an approximation can be obtained from the transfer ratio of the

DC-DC converter (e.g. for a 5V to 5V converter the ratio is 1:1).

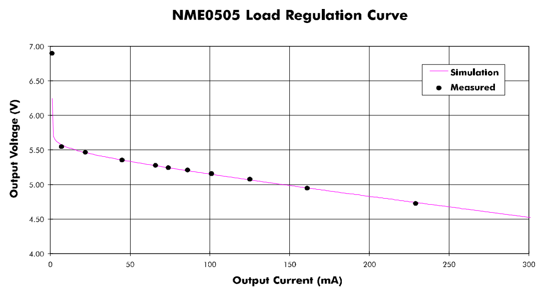
The AC ripple voltage was chosen to give a zero DC bias effect so as not to offset the output voltage bias levels during AC simulations, hence a small positive and negative excursion is modelled. The ripple frequency was chosen at twice the nominal switching frequency since full wave rectification is used in the actual device being modelled. The switching waveform is square from the oscillator and to model this a fast rise/fall pulse waveform is used.

```
*******************************************************

*** NEWPORT COMPONENTS LIMITED
*** NME0505S DC-DC CONVERTER MODEL
*** 1W ISOLATED SINGLE OUTPUT DEVICE
*******************************************************
*** NODE DESCRIPTION: VIN GND +VOUT 0V
.SUBCKT NME0505 1 2 3 4
DOUT 5 3 NCLD105
.MODEL NCLD105 D (IS=1E-7 N=2.3 RS=2.8
+ EG=1.11 XTI=3 BV=50 IBV=25.9E-3 TT=2E-9 )
COUT 3 4 0.5UF
E1 6 4 1 2 1.25
F1 1 2 VRPL -1.25
RNL 2 1 250
CIN 2 1 0.5UF
VRPL 5 6 DC 0 PULSE -0.68 0.02 0 0.25US 0.25US 4.5US 5US
CISOL 1 3 24PF
.ENDS NME0505
*******************************************************
```

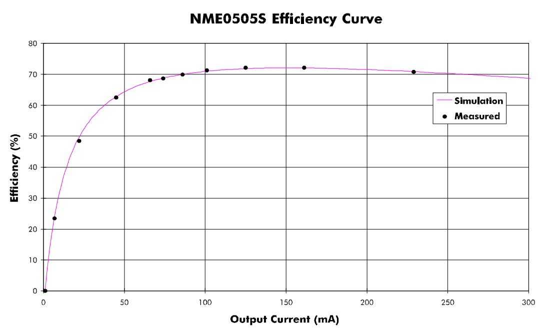
The most difficult part to determine values for is the diode. This is a model in itself and included in this model is an estimation of all the DC resistance effects due to tracking, wire and the active components. The diode model was derived from the models of the diode used in the circuit with an estimation of the resistive contribution. The values of the parameters governing resistance (RS) and saturation current (IS) were adjusted to produce a load curve which matched that of the data sheet.

If the diode model is not known, using an ideal diode and adjusting the above two diode parameters manually is adequate.

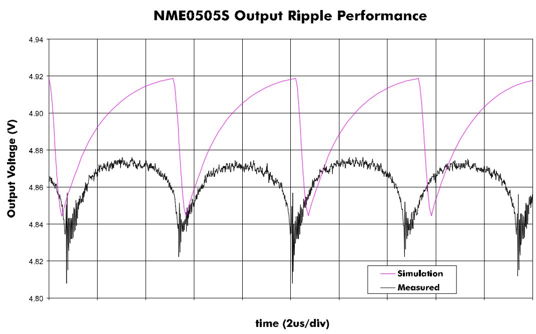


Simulation Compared to Measured Performance

Measurements of the output voltage and efficiency were taken from a single NME0505S device and graphically compared to the simulated load and efficiency curves. The results illustrate that the model is an extremely close representation of the real circuit's DC operating characteristic. Additional tests of the line regulation at a fixed output load exhibited similar levels of close agreement.



Ripple was measured on a DSO and compared to the simulated response. The measured result has a lot more noise than the simulation and the switching frequency is not an exact match. Despite these small differences the general shape of the ripple is similar, the measured result being slightly more 'rounded' than the simulation, and the magnitude is similar, approximately 60mV measured compared to 75mV simulated. The DC offset voltage is also very close, being only 40mV apart, hence the AC simulation is a reasonably close representation to the measured result.



Limitations of the DC-DC Converter Macro Model

The macro model is always a compromise, usually sacrificing accuracy under all operating conditions for convenience and ease of use. The DC-DC converter is no exception and although the ripple and DC load and line regulation curves are good matches, this situation is only true over the normal operating conditions of the device.

Outside of the maximum recommended load condition, the oscillator circuit would produce a relatively rapid fall in output voltage as the ripple capacitor failed to hold up the output, whereas the model continues to hold a linear regulation curve. At zero load the actual output voltage rises to quite a high value as the ripple voltage charges up the capacitor until the output capacitor leakage and ripple charge are in equilibrium, in the model a much lower zero load output voltage is simulated.

The output ripple voltage itself is constant in the model regardless of load conditions, in reality ripple is much lower at light loads. The frequency of oscillation also changes with input voltage, again this is not modelled here and the ripple magnitude and frequency are simulated as being constant.

Summary

Despite the limitations the model is a close representation of the behaviour of a fixed input DC-DC converter. The model provides a fast and accurate simulation in both AC and DC analyses modes.

The model can be used by manufacturers of DC-DC converters to provide accurate modelling data to their customers without disclosing the entire circuit design of their products. Likewise customers using DC-DC converters can produce their own models where none are available from the supplier. The user of the DC-DC converter can then model the characteristic of their target circuits, including the behaviour of the DC-DC converter, without the previously high simulation overhead or non-convergence problems of an oscillator circuit.

References

[1] Switching Power Supply Design, A.I. Pressman, McGraw-Hill, 1991.

[2] IsSPICE User's Guide, Intusoft, 1994.

[3] Power Supply Design Seminar, SEM-1000, Unitrode, 1994.