

AN1692

Spice model eases feedback loop designs

Christophe BASSO

MOTOROLA SPS, Toulouse Center

Avenue Eisenhower, BP1029, 31023 TOULOUSE Cedex France

33 (0)5 61 19 90 12; e-mail: R38010@email.sps.mot.com

Introduction

Within the wide family of Switch Mode Power Supplies (SMPS), the Flyback converters represent the preferred structure for use in small and medium power applications (50–100W) such as wall adapters, off-line battery chargers, fax machines etc. Thanks to the recent introduction of MOTOROLA Very High Voltage Integrated Circuits (VHVICs), the design phase of a Flyback converter has almost turned into a child's play: a bunch of passive components around the MC33370 and you are all set! However, the impact of the environment upon the system can be much longer to iterate in order to cover the numerous situations the converter will encounter in its future life: ESRs variations due to temperature cycles, aging of the capacitors, various types of load, load and line transients, effect of the filter stage etc.

In this article, we will show how a Spice simulator can help the designer to quickly implement his concept and show how it reacts to the outside world constraints. This brings several benefits such as an immediate understanding of how the circuit operates and naturally, an improved time-to-market cycle.

Defining the chain

Figure 1 depicts the complete chain implemented in a standard Flyback converter. In order to draw a Bode plot and thus place appropriate pole/zeros combinations, we need to individually assess every gain block.

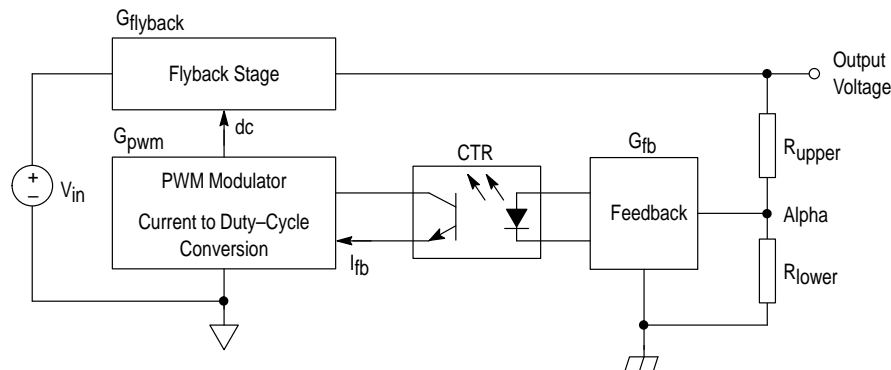


Figure 1. A Flyback chain implementing current feedback

In a shunt regulator, such as the new VHVIC series, the duty-cycle is adjusted by injecting a current into the feedback pin. The approach differs from the classical voltage feedback devices and requires a detailed explanation to help you comfortably deal with this concept.

The duty-cycle conversion

The members of the MC3337X series implements a duty-cycle control through a shunt regulator. That is to say, the duty-cycle (dc) is adjusted by injecting a current into a feedback pin (FB). When the current is low or zero, the duty-cycle is pushed to the max (74% typical for the MC33370). If more current is pushed into the pin, the duty-cycle goes toward a few percents (0.9% typical). The amount of current needed to go from full dc to null dc determines the PWM gain. **Figure 2a** details how the duty-cycle conversion takes place:

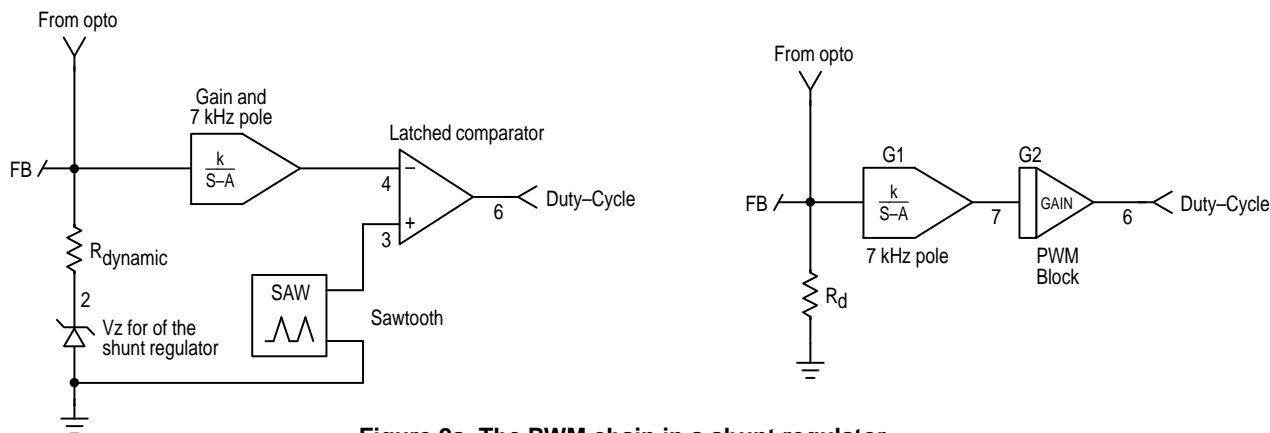


Figure 2a. The PWM chain in a shunt regulator

To remove any spurious noises due to the high peak currents flowing around the IC, the designers have installed an internal 7kHz low-pass filter. This pole shall be included in the final AC study.

As previously said, the FB corresponds to the input of a shunt regulator. To better understand the way it works, you can replace the shunt regulator by a power zener: when the voltage you apply on the FB pin is below the shunt breakdown level, no current flows into the pin and dc is maximum. When the FB level reaches the zener threshold, a current circulates in the pin and is converted into a lowering duty-cycle. First remark, in steady-state operation, the FB pin is at the shunt level as given in the data-sheet: 8.6V for the MC3337X. You shall then provide the feedback current through a source whose value is, at least, two or three volts above the shunt value. Otherwise you will not reach the appropriate level to regulate and you will force the optocoupler to operate in a low V_{CE} region where the conductance dI_C/dV_{CE} is rather poor.

For AC analysis, the FB pin can be replaced by the dynamic resistor of the power zener, dV_{zener}/dI_{diode} or 18Ω for the MC3337X series. This value gives you the AC impedance seen from the FB pin. It will dictate the locations of the poles and zeroes you create by adding capacitors around this pin. This is NOT the PWM gain, but rather an intermediate current/voltage conversion gain. The complete gain, as highlighted by figure 1, depends on the internal sawtooth amplitude (1.4Vpp for the MC3337X) and the maximum duty-cycle. In these ICs, the presence of the latched comparator makes the error signal reach the sawtooth peak value when the duty-cycle is at the maximum (74%). The total gain calculation is done following the steps:

ΔI_{FB} of 6mA $\rightarrow \Delta dc$ of 74% or a slope of 12.3%/mA typical.

$6mA \cdot 18\Omega = 108mV$

$$G1 = 20 \cdot \text{Log}\left(\frac{1.4}{108 \text{ m}}\right) = 22.25\text{dB}$$

$$G2 = 20 \cdot \text{Log}\left(\frac{0.74}{1.4}\right) = -5.53\text{dB}$$

Finally since the internal FB dynamic impedance also participates to the gain, the complete PWM chain exhibits the following values:

MC3337X: $22.25\text{dB} + 25.1\text{dB}_{18\Omega} - 5.53\text{dB} = 41.82\text{dB}$

Figure 2b gives the data-sheet information on the duty-cycle conversion ratio while **figure 2c** gives simulation results of the PWM stage only as it is offered in the MC3337X Spice model.

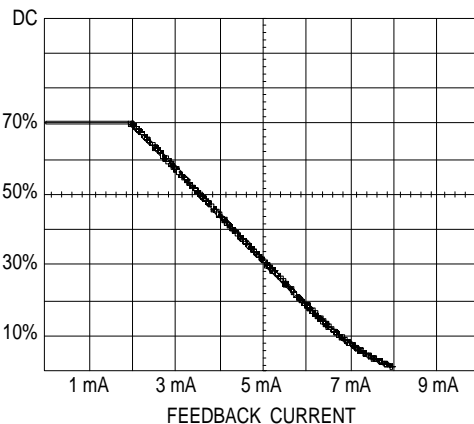
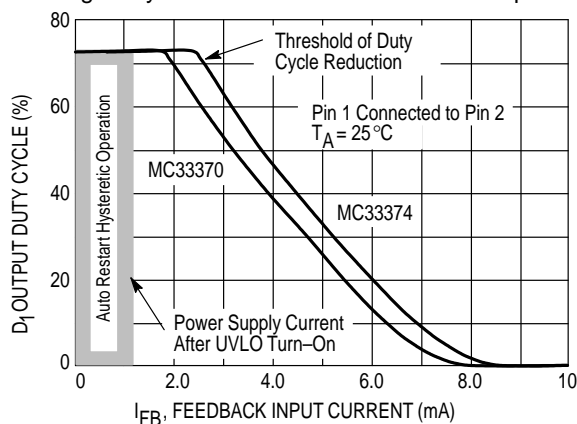


Figure 2b. MC3337X duty-cycle versus injected current **Figure 2c. Simulated PWM section of the MC3337X**

As we will see later on, the calculation of the whole chain (Flyback stage, feedback etc.) *static* gain does NOT involve the knowledge of the input impedance of the FB pin, but rather the current/ dc conversion ratio, SP_{PWM} .

First case: the optocoupler is alone

This is the most economic case where the output voltage does not require a tight regulation. The optocoupler Light Emitting Diode (LED) is simply inserted in series with a zener diode. The output level is then close to $V_z + V_f$, with V_z the zener voltage and V_f the forward level of the LED. **Figure 3a** depicts this situation when a compensation network made of R_s and C_1 is added (Z_1 impedance). In the calculation, we are evaluating the voltage present on the FB pin to isolate the feedback network contribution.

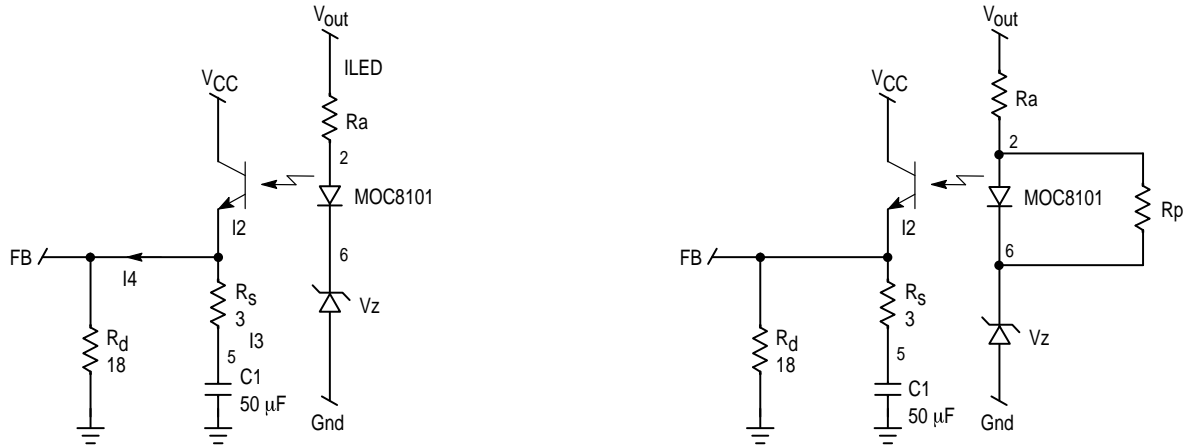


Figure 3a and 3b.

On the right side, a resistor R_p is added to refine the stabilization but it also *slightly* decreases the loop gain

The calculation steps are rather easy and will be reproduced all along this document. It consists in a) evaluating the LED current b) finding its relationship with I_{FB} or V_{PWM} c) derive the result to obtain the small-signal gain. However, in a so simple schematic, we can highlight the parasitic element the LED and the zener are made of (**figure 4a**). Taking into account that the perfect sources V_f and V_z do not move with V_{out} , the final equation for the LED current reduces to:

$$I_{LED} = \frac{V_{out}}{R_A + R_{d_{LED}} + R_{dz}}$$

$$I_2 = I_{LED} \cdot CTR$$

$$V_{FB} = I_4 \cdot R_d = I_2 \cdot \frac{Z_1 \cdot R_d}{Z_1 + R_d} = I_{LED} \cdot CTR \cdot \frac{Z_1 \cdot R_d}{Z_1 + R_d}$$

$$V_{FB} = \frac{V_{out}}{R_A + R_{d_{LED}} + R_{dz}} \cdot CTR \cdot R_d \cdot \frac{1 + \frac{1}{R_s \cdot C_1 \cdot p}}{1 + \frac{1}{(R_s + R_d) \cdot C_1 \cdot p}}$$

$$\frac{dV_{FB}}{dV_{out}} = [R_d // R_s] \cdot \frac{CTR}{R_A + R_{d_{LED}} + R_{dz}} \cdot \frac{1 + \frac{1}{R_s \cdot C_1 \cdot p}}{1 + \frac{1}{(R_s + R_d) \cdot C_1 \cdot p}}$$

we then define a zero f_z and a pole f_p : $f_z = \frac{1}{2 \cdot \pi \cdot R_s \cdot C_1}$ and, $f_p = \frac{1}{2 \cdot \pi \cdot (R_d + R_s) \cdot C_1}$

In DC, the gain simplifies to: $DC_{gain} = \frac{d_{dc}}{dV_{out}} = - \frac{R_d \cdot CTR}{R_A + R_{d_{LED}} + R_{dz}}$ while in high-frequency, when C_1 is a complete short:

$HF_{gain} = \frac{d_{dc}}{dV_{out}} = - \frac{[R_d // R_s] \cdot CTR}{R_A + R_{d_{LED}} + R_{dz}}$. To obtain the final chain V_{out} to the PWM's output, you need to add G_1 and G_2 as defined above.

However, if you only account for the I_{FB}/dc conversion slope (S_{PWM}) of $-12.3\%/mA$ (or $-123/A$), the *complete* DC gain from V_{out} to the PWM stage's output is simply:

$$DC_{gain} = \frac{d_{dc}}{dV_{out}} = - \frac{CTR \cdot S_{PWM}}{R_A + R_{d_{LED}} + R_{d_Z}}$$

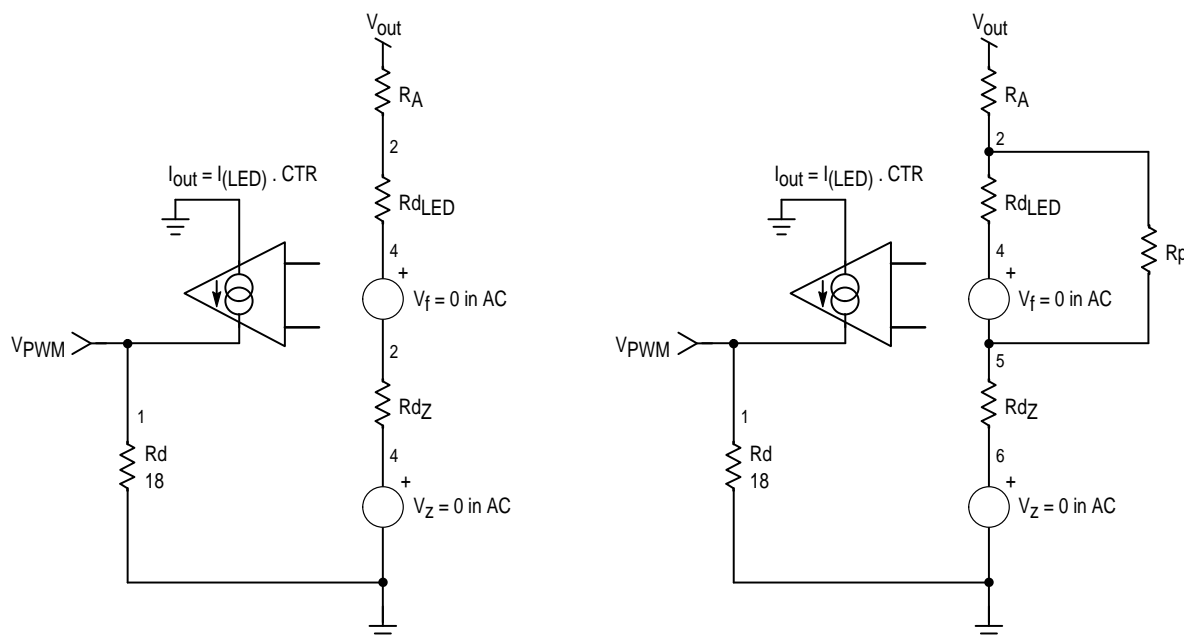


Figure 4a and 4b.

Since V_F and V_Z do not vary in AC, we can put them to zero for the analysis

In some applications, it is interesting to increase the current flowing into the zener to gain in precision: the zener operates away from its knee where dV_Z/dI_{Zener} is rather high. To implement this option, simply wire a resistor in parallel with the LED as **Figure 3b** and **Figure 4b** show. In AC, R_p now comes in parallel with R_d and affects the total DCgain (from V_{out} to the PWM's output) by:

$DC_{gain} = - \frac{1}{R_A + R_{d_{LED}} // R_p + R_{d_Z}} \cdot \frac{R_p}{R_p + R_{d_{LED}}} \cdot CTR \cdot S_{PWM}$. When R_p becomes infinite, this formula simplifies to the previous one. As we can imagine, with rather low values of $R_{d_{LED}}$, the gain is *slightly* degraded by the presence of R_p .

Numerical application for figure 3a example:

$R_d = 18\Omega$ $CTR = 1.8$ (180%) $S_{PWM} = 12.3\%/mA$
 $C_s = 50\mu F$ $R_A = 270\Omega$
 $R_s = 3\Omega$ $R_{d_{LED}}$ and R_{d_Z} are neglected.

$DC_{gain} = -18.41dB + G_1 + G_2 = -1.723dB$

$HF_{gain} = -35.32dB + G_1 + G_2 = -18.6dB$

1st pole = 151Hz

1st zero = 1.061kHz

The very low static-gain engendered by this configuration is not compatible with a good audiosusceptibility (input line rejection). The TL431 will help us to raise this poor value.

An integrator with the TL431 to boost the DC gain

By wiring a TL431 as depicted by **figure 5a**, we will offer better ripple rejection by raising the DC gain. For the sake of clarity we have removed the previous passive RC network, but its action is similar as the one already calculated. Please note that V_{out} is split in two values: V_{out} and $k \times V_{out}$. In Flyback converters operating in Discontinuous Conduction Mode (DCM), the high secondary peak current generates a thin output spike when combined with the output capacitor's ESR. To fight against this problem, you can add a small series inductor of a few μH . Unfortunately, it also adds a second order high-frequency pole that you do not want to include into the final feedback path. You then split the feedback in two ways: a fast one with low gain through the LED anode ($k \times V_{out}$) and a low-frequency with high gain on the resistive divider (V_{out}). As a first remark, we can see that when either the TL431's gain or the network across it roll its gain to zero, we come back to figure 3a configuration. Therefore we cannot really roll the whole loop gain to zero! That is a typical pain of the TL431 but we can also turn it into an advantage as we will see later on.

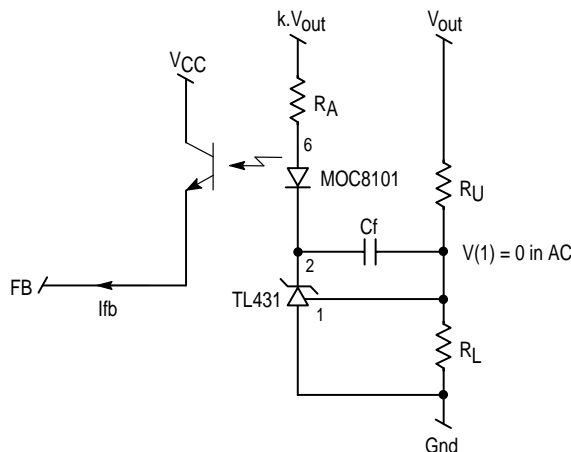


Figure 5a. A TL431 helps to rise the gain in DC

Let us start by the DC analysis where C_f is open and in lack of feedback on the TL431, the node 1 is NOT a virtual ground (or at internal V_{ref}):

$$I_{LED} = \frac{k \cdot V_{out} - (V_f + V_z)}{R_A} \text{ with } V_z = \text{TL431's Anode-Cathode voltage.}$$

$$V_z = -V_{out} \cdot \frac{R_L}{R_U + R_L} \cdot A_{v_{TL431}} \text{ and } I_{FB} = I_{LED} \cdot CTR. \text{ The final equation for } I_{FB} \text{ is then:}$$

$$I_{FB} = \frac{k \cdot V_{out} - \left(V_f - \frac{V_{out} \cdot R_L}{R_L + R_U} \right) \cdot A_{v_{TL431}}}{R_A} \cdot CTR$$

$$\frac{d_{dc}}{dV_{out}} = - \frac{k \cdot (R_L + R_U) + R_L \cdot A_{v_{TL431}}}{R_A \cdot (R_L + R_U)} \cdot CTR \cdot S_{PWM} \rightarrow DC_{gain}$$

In AC, the first I_{LED} equation still holds. But this time C_f closes the TL431 feedback path and maintains a true virtual ground on node 1: $V(1) = 0$ in AC. C_f creates an integrator with R_U (R_L does not play in AC because of the virtual ground) and the V_z parameter is expressed by:

$$V_z = -V_o \cdot \frac{1}{R_U \cdot C_f \cdot p}$$

$$I_{LED} = \frac{k \cdot V_{out} - \left(V_f - V_{out} \cdot \frac{1}{R_U \cdot C_f \cdot p} \right)}{R_A}$$

$$\frac{d_{dc}}{dV_{out}} = - \frac{(k \cdot R_U \cdot C_f + 1)}{R_U \cdot C_f \cdot p} \cdot \frac{CTR \cdot S_{PWM}}{R_A} \rightarrow AC_{gain} \text{ with a pole } f_p = \frac{1}{2 \cdot \pi \cdot R_U \cdot C_f} \text{ and a zero located at } f_z = \frac{1}{2 \cdot \pi \cdot k \cdot R_U \cdot C_f \cdot p}.$$

To verify our calculations, a Spice engine is a very good tool. **Figure 5b** depicts an INTUSOFT's IsSpice4 (San–Pedro, CA) simulation schematic of the TL431 architecture.

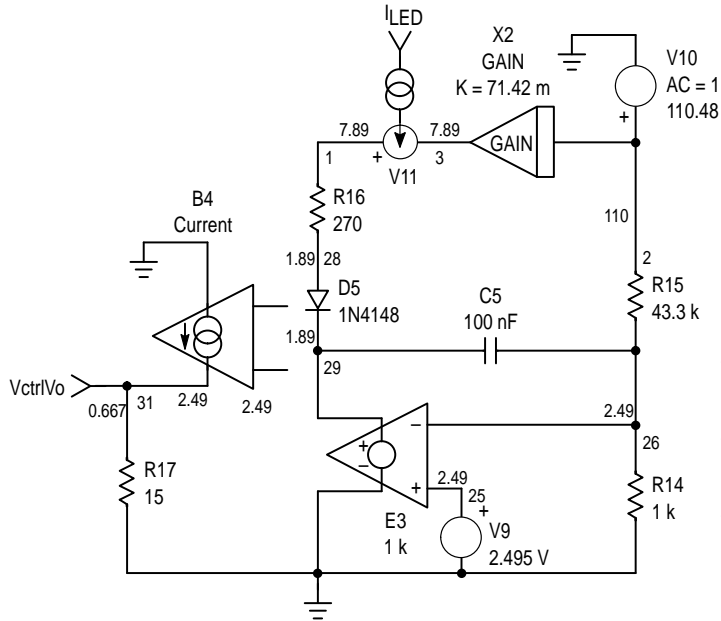


Figure 5b. A IsSpice4 simulation schematic of the TL431 structure

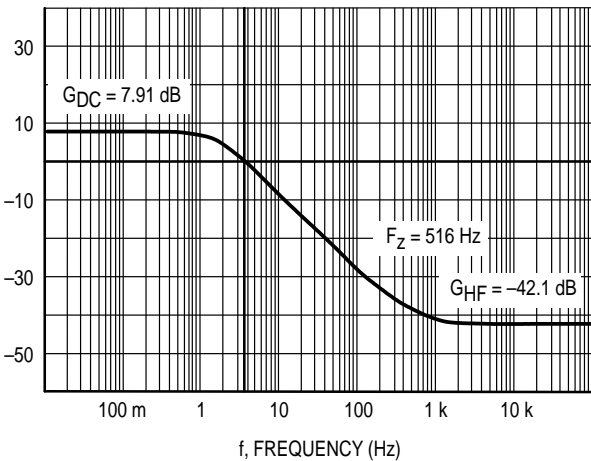


Figure 5c. The Bode plot revealed after simulation

This example simulates a Flyback converter delivering a 112V level (e.g. in a TV application) while the LED is biased through an auxiliary 8V winding. Therefore $k = 8/112 = 71.42\text{m}$. The optocoupler is replaced by a current–controlled current source with a gain of 2 (CTR = 200%) for simpler calculations. V10 is adjusted to keep a correct DC point (as the schematic values testify) and is AC modulated to draw the output Bode plot. Again, we purposely isolate this feedback system and the output is taken *upon* R17 and NOT, as before, at the PWM's output.

Numerical application for figure 4b example:

$R_d = 15\Omega$ CTR = 2 (200%)
 $R_A = 270\Omega$ TL431 gain = 1000
 $R_U = 43.3\text{k}\Omega$ $R_L = 1\text{k}\Omega$
 $C_f = 100\text{nF}$ $k = 71.42\text{m}$

DCgain = 8 dB

HFgain = -42 dB

1st pole $\times \frac{\text{CTR} \cdot R_d}{R_A} = f_c @ 0 \text{ dB} = 4.08 \text{ Hz}$ (the small mismatch is due to the natural low $A_{v\text{TL431}}$ of 55dB)

1st zero = 514.6 Hz

The TL431 with a DC feedback

We now add a simple feedback resistor between the TL431's cathode and its reference pin (**figure 5c**).

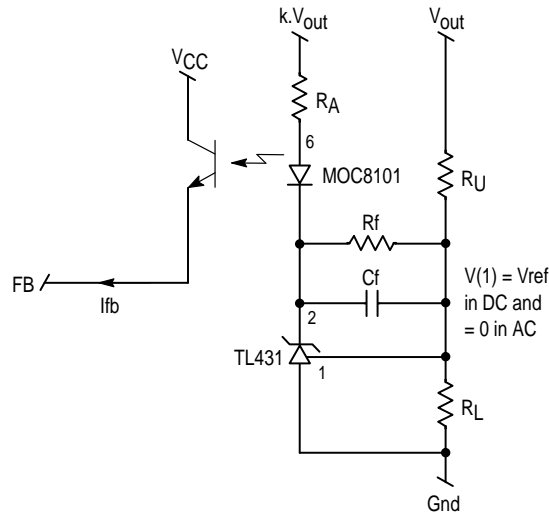


Figure 5c. The DC is no longer in open-loop for the TL431 thanks to Rf

This time, we have a virtual ground in DC ($V_{node1DC}=V_{ref}$) and AC. The expression of the LED current changes a bit as R_L no longer acts:

$$I_{LED} = \frac{k \cdot V_{out} - \left(V_f - V_{out} \cdot \frac{R_f}{R_U} \right)}{R_A} \text{ and } I_{FB} = I_{LED} \cdot CTR$$

$$\frac{d_{dc}}{V_{out}} = - \frac{k \cdot R_U + R_f}{R_U \cdot R_A} \cdot CTR \cdot S_{PWM} \rightarrow DC_{gain}$$

In AC, $V_z = -V_{out} \cdot \frac{R_f}{R_U \cdot (1 + R_f \cdot C_f \cdot p)}$ and the final gain becomes:

$$\frac{d_{dc}}{dV_{out}} = - \frac{(k \cdot R_U + R_f) \cdot \left(1 + \frac{k \cdot R_U \cdot R_f}{k \cdot R_U + R_f} \cdot C_f \cdot p \right)}{R_U \cdot R_A \cdot (1 + R_f \cdot C_f \cdot p)} \cdot CTR \cdot S_{PWM} \rightarrow AC_{gain} \text{ expression}$$

Spice eases the Bode plot generation

As we have seen, many parameters affect the feedback path and greatly complicate the manual drawing of the Bode plot. Therefore, an automated solution calculating automatically the operating DC point while plotting the AC response of the whole system would be welcome. A Spice simulator is the ideal tool for this purpose. **Figure 6a** shows a simple secondary regulated SMPS made with the new member of the MC3337X family Spice model [2]. Please note that a Spice model is also available for TRANSient simulations.

The usual method consists in opening the loop at a place where ALL the feedback paths are gathered. It is obviously the PWM's output fortunately made available in the AC model. The most difficult thing in a high-gain closed-loop system is to open the loop without disturbing the operating point (e.g. at given conditions). Various methods including injection transformers have been extensively described and relevant information can be found on Venable's Web site [1]. With Spice, L1 opens the loop in AC (huge inductor value) but nicely closes it in DC, thus always keeping the good operating point.

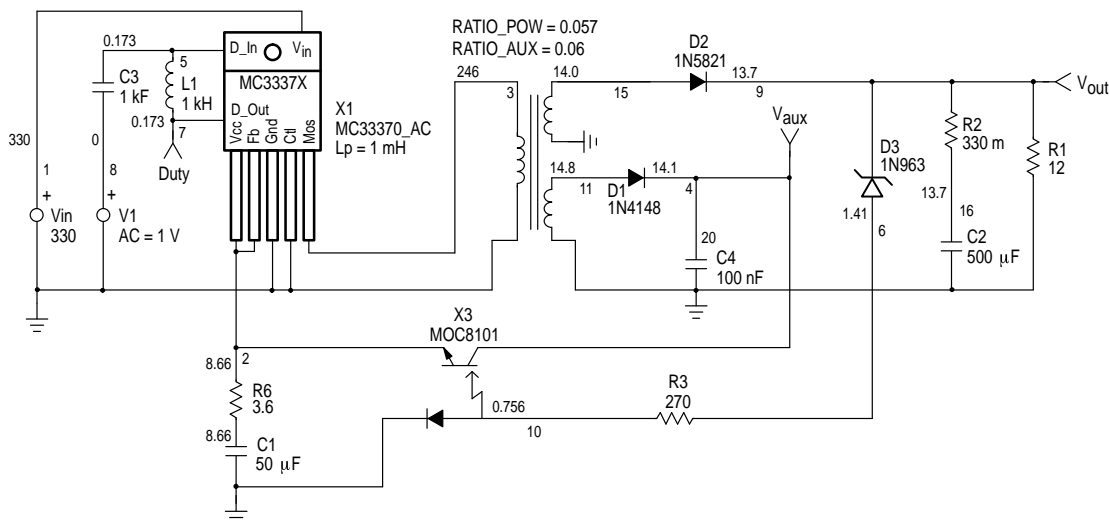


Figure 6a. A simple 12W SMPS made with an MC3337X

Node 5 will receive the AC stimulus via C3, the injection capacitor. To check the transient response, simply reduce *both* values to 1p. The Bode plot is easily generated and the DC point moves in function of the external conditions (line level, load variations etc.). Figure 6a numerical values are transferred to the schematic once the simulation has completed. Their values let you check the validity of the selected DC point. It also delivers the static duty-cycle corresponding to the operating conditions (with AC models, 1V gives 100% dc, so 173mV corresponds to 17.3%). As you can see, Spice offers a tremendous help since it naturally includes the optocoupler pole but also the internal PWM 7kHz pole. The bode plot is given below (**figure 6b**).

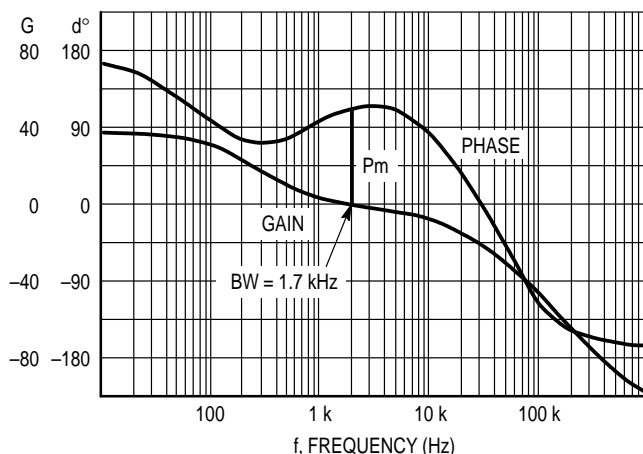
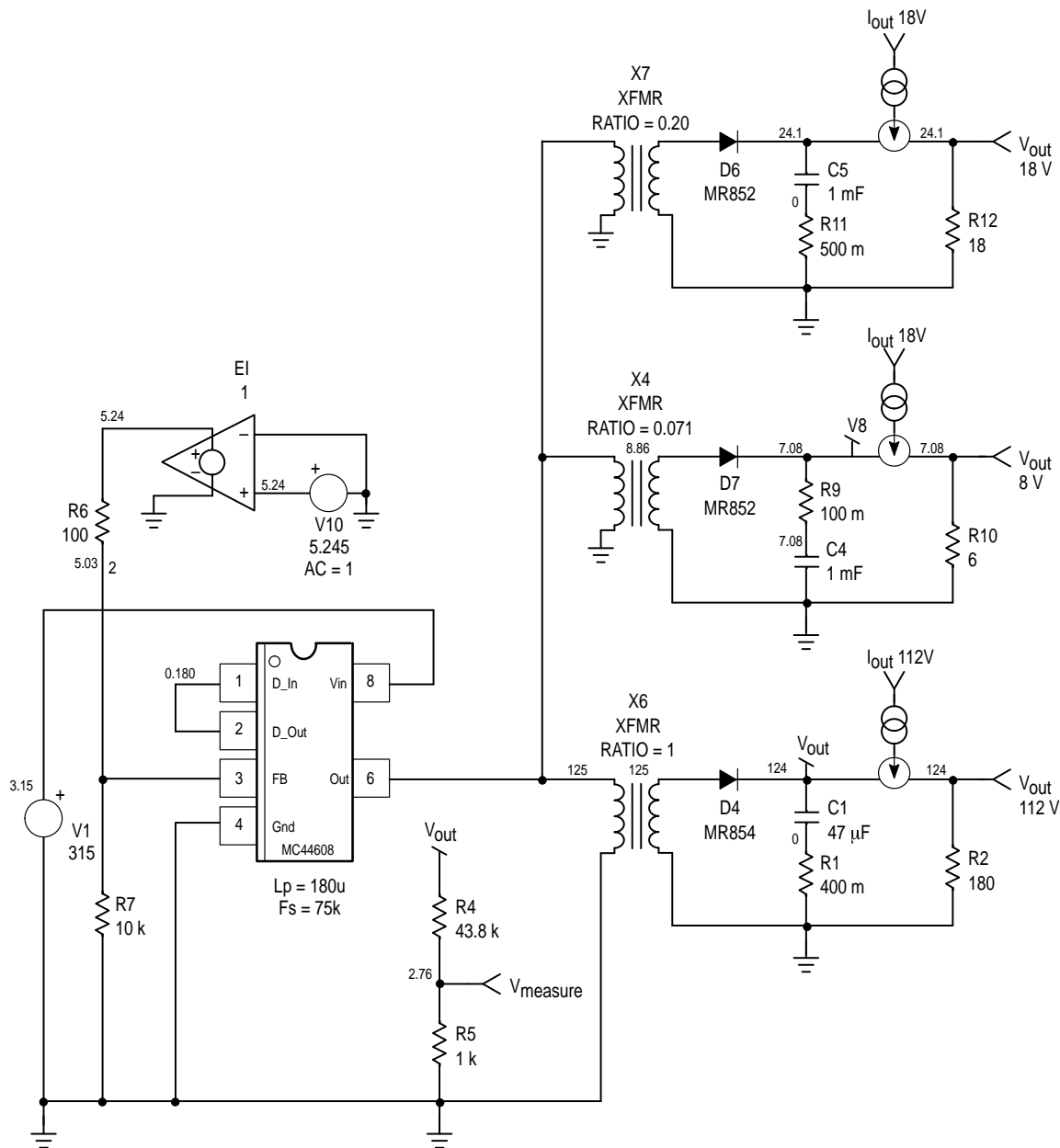


Figure 6b. The bode plot is drawn in a snap-shot

By adjusting the values of the load, mains input etc. you can immediately verify if the SMPS stays stable for instance on universal mains. If not, simply move the location of the pole/zeroes with R6–C1 until you obtain a sufficient phase margin (Pm). Very easy, isn't it? The Spice model of the MC3337X operates in both Discontinuous Conduction Mode (DCM) and Continuous Conduction Mode (CCM). Please note that this simulation example netlist is available to download on the MOTOROLA Web site [2] in both INTUSOFT's IsSpice4 and MICROSIM's PSpice syntax. The other goods news is that it works on the demonstration versions!

Figure 7a depicts a multi-output SMPS using another member of the VHVICs, the new MC44608. This IC is dedicated to monitor applications where standby power is premium. It also uses a PWM shunt regulator but its FB pin is kept at 5V rather than 8.6V. The PWM modulator and the Flyback stage are identical to that of the MC3337X. Figure 7a strengthens the help of the model because Spice naturally reduces the multi-output system to a single output (the regulated one) loaded by $R_{load_{eq}}$ and filtered by $C_{out_{eq}}$. These elements simply represent the actual components you should have reflected back to the regulated winding accordingly with their squared turn ratios. You easily imagine the pain when covering load or ESR variations! In the below example we have completely opened the loop and the injection point is the FB pin via a 100Ω resistor.



Motorola Applications Data

To verify the validity of our approach, a real Bode plot has been drawn from the regulated point Vout 112V using a network analyzer. As you can see on **figures 7b** and **7c**, the results are in very good agreements with an IsSpice4 plot.

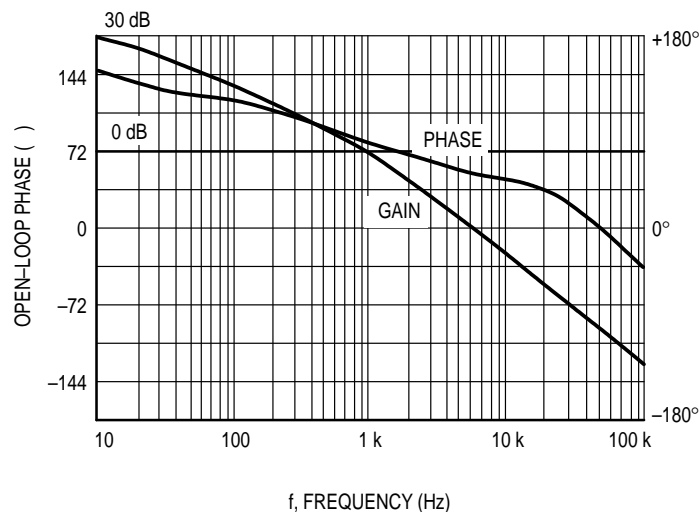


Figure 7b. IsSpice4 Simulation Results

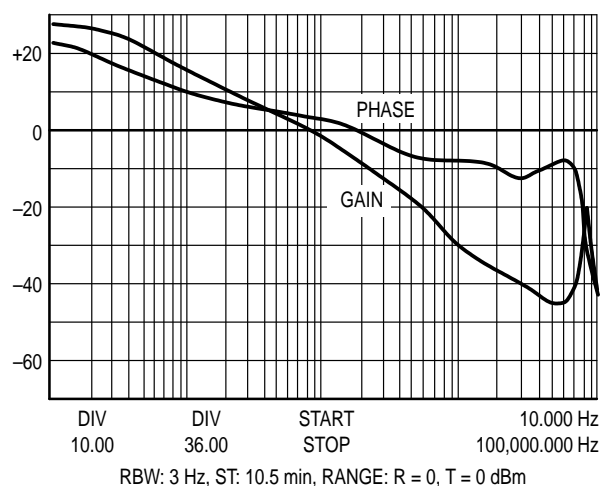



Figure 7c. A real Bode plot measurement

Conclusion

This paper shows the benefits of using a Spice model when designing Switch-Mode Power Supplies. It also details the way to introduce various poles/zeros combinations when implementing a shunt regulator. Finally, it confirms the power of a simulation engine to help adapting the system performance to the application needs in a minimum of time and without tears!

1. <http://www.venableind.com/>
2. MOTOROLA's Spice models and example netlists: <http://sps-mot.com/scg> then click on "Device models"
3. MOTOROLA Application Note AN1683, "A Spice Model for High-Voltage Integrated Switchers"

NOTES

Motorola reserves the right to make changes without further notice to any products herein. Motorola makes no warranty, representation or guarantee regarding the suitability of its products for any particular purpose, nor does Motorola assume any liability arising out of the application or use of any product or circuit, and specifically disclaims any and all liability, including without limitation consequential or incidental damages. "Typical" parameters which may be provided in Motorola data sheets and/or specifications can and do vary in different applications and actual performance may vary over time. All operating parameters, including "Typicals" must be validated for each customer application by customer's technical experts. Motorola does not convey any license under its patent rights nor the rights of others. Motorola products are not designed, intended, or authorized for use as components in systems intended for surgical implant into the body, or other applications intended to support or sustain life, or for any other application in which the failure of the Motorola product could create a situation where personal injury or death may occur. Should Buyer purchase or use Motorola products for any such unintended or unauthorized application, Buyer shall indemnify and hold Motorola and its officers, employees, subsidiaries, affiliates, and distributors harmless against all claims, costs, damages, and expenses, and reasonable attorney fees arising out of, directly or indirectly, any claim of personal injury or death associated with such unintended or unauthorized use, even if such claim alleges that Motorola was negligent regarding the design or manufacture of the part. Motorola and  are registered trademarks of Motorola, Inc. Motorola, Inc. is an Equal Opportunity/Affirmative Action Employer.

Mfax is a trademark of Motorola, Inc.

How to reach us:

USA/EUROPE/Locations Not Listed: Motorola Literature Distribution;
P.O. Box 5405, Denver, Colorado 80217. 1-303-675-2140 or 1-800-441-2447

JAPAN: Motorola Japan Ltd.; SPD, Strategic Planning Office, 141,
4-32-1 Nishi-Gotanda, Shinagawa-ku, Tokyo, Japan. 81-3-5487-8488

Customer Focus Center: 1-800-521-6274

Mfax™: RMFAX0@email.sps.mot.com – TOUCHTONE 1-602-244-6609
Motorola Fax Back System – US & Canada ONLY 1-800-774-1848
– <http://sps.motorola.com/mfax/>

ASIA/PACIFIC: Motorola Semiconductors H.K. Ltd.; Silicon Harbour Centre,
2, Dai King Street, Tai Po Industrial Estate, Tai Po, N.T., Hong Kong.
852-26668334

HOME PAGE: <http://motorola.com/sps/>



MOTOROLA



AN1692/D