Flyback Converter with Type3 Compansation

Enes Erçin
Faculty of Electrical Engineering
Istanbul Technical University

1. Aim

This study aims to provide practical steps to implement and design a type-3 error-compensated flyback dc-dc converter circuit with simulation tools. Calculation of parameters is done by Matlab software the parameters which are calculated are then used in circuit simulation in Matlab. There are some assertions to limit user inputs to choose different parameters but it is not very sophisticated yet due to the complexity of the system. It is possible to transfer parameters into different platforms like Ltspice or Orcad but the automation and circuit templates are not yet prepared.

2. Introduction

Flyback is a commonly used dc-dc converter topology. Similar to other converters dc-dc converters flyback circuit stores energy via magnetizing inductance at one stage, and consumes that energy at the other stage. In a simple design approach, we can make some assumptions to choose the parameters of the inductance, capacitance, load resistance, and switching period values. Most of the assumptions are derived from making circuit components like inductors and capacitors and ideal circuit components with no additional resistance to them. For a flyback converter, we can deduce that the output voltage is proportional to, source voltage, duty ratio, and turns ratio by the assumptions mentioned $V_o = V_s \cdot (\frac{D}{D-1}) \cdot (\frac{N1}{N2})$. In reality, this would not give an exact result simply due to components being not an ideal one in reality.

One widely used solution might be to investigate the circuit by transfer functions of the small signal disturbances around steady-state operating points such as $v_o = V_o + \widetilde{v_o}$ and $v_s = V_s + \widetilde{v_s}$.

Investigating the circuit by doing small signal ac analysis with the addition of loss elements to the switch, indicator and capacitor we can extract a useful information. It is than possible to derive a transfer function and investigate the circuit in frequency domain. Analysing frequency response will indicate crucial information about the circuits stability. By looking at critical gain and phase margin in critical frequancy called crossover frequancy (f_{co}). To make the circuit stable in wide frequancy range than before it is possible to add a feedback loop to compansate the error by adjusting the switching voltage. In later chapters possible modifications will be discussed.

The main concerns are two parameters which are the disturbances in input voltage and disturbances in the control voltage of the switching element. To find the effects of small changes in $G_s(s) = \frac{\widetilde{v_o}(s)}{\widetilde{v_s}(s)}$ or $G_c(s) = \frac{\widetilde{v_o}(s)}{\widetilde{v_c}(s)}$ either one the other ones effect is reduced to zero.

3. Flyback analysis

The basic approach used in many textbooks such as Hart's Power Electronics (2011) is initially separating the circuit into stages. Then make some general trivial assumptions like the average current passing through the magnetizing inductance is zero.

Assuming an inductor, transformer, switching element, and capacitors are ideal. The average inductor current is zero as well as average capacitor voltage is zero and the transformer and switching elements has no losses.

$$V_o = V_s \cdot \left(\frac{D}{D-1}\right) \cdot \left(\frac{N_1}{N_2}\right)$$

$$(L_m)_{min} = \frac{(1-D^2) \cdot R}{2f} \cdot \left(\frac{N_1}{N_2}\right)^2$$

$$(L_m) = \frac{V_s \cdot D}{f \cdot \Delta i_{Lm}}$$

$$(\frac{\Delta V_o}{V_o}) = \frac{D}{RCf}$$

Desired parameters:

Properties:	**Features:**
V_o	48 V
V_s	24V
P_o	50W
$\frac{\Delta V_o}{V_o}$	0.05

TABLE 1. DESIRED PARAMETERS

Properties:	**Features:**
D	0.5
f	1kHz

TABLE 2. ARBITARY PARAMETERS

Properties:	Features:
R_l	50 Ω
L_m	16.5uH
$N1/N2$	0.5
C	2uF

TABLE 3. CALCULATED PARAMETERS:

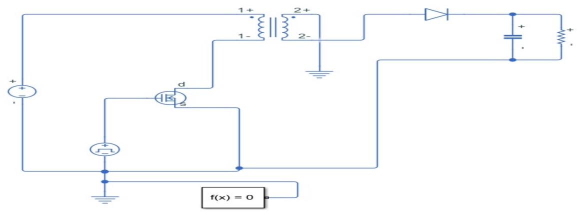

Figure 1. Circuit in simulink

The circuit layout is given in figure 1. The output of the simulation results given in Figure 2 is relatively satisfactory for the initatal attempt. The desired output voltage was 48 V and the average output voltage that we get from the simulation is 43V with the given parameters. Keep in mind the equation used to derive appropriate component values are an approximation. It is often not possible for simulation tools to evaluate ideal cases. For example, the transformer in Figure 1 in Simulink had a parameter called parallel resistance at each side. When it is made zero solution of the circuit cannot be calculated thus some value had to be added as a parallel resistance. To make their effect as minimal as possible large value of resistance is chosen. Another case of this difference from the expected output voltage is the switch. Although the ideal switch option was available the operation of that component was not as expected I had to use MOSFET as a switch, which adds additional complexity to the solution.

4. Averaged Model of Flyback

The previous model is evaluated by the simulation for each intermediate step. This increases the computational workload and may not be always necessary. For instance, if we would not care about the inductor current ripple we would not need the transient response of the circuit for every step. To fix this issue for faster circuit analysis through simulations is to use an average model. Through one period the action of a switch is the same as the action of a transformer which does not necessarily work with only AC. In other words, what the switch does on average reduces the output current and voltage to a certain ratio which we can equate to say turns ration of the equivalent transformer. That ratio is the same as duty ratio D. The adapted circuit for average simulation is given in Figure 4 and the simulation outputs are given in Figure 3. Although some transient values are not clear it gives important general information about the circuit. One side note which may be important to mention is that this average model cannot be simulated

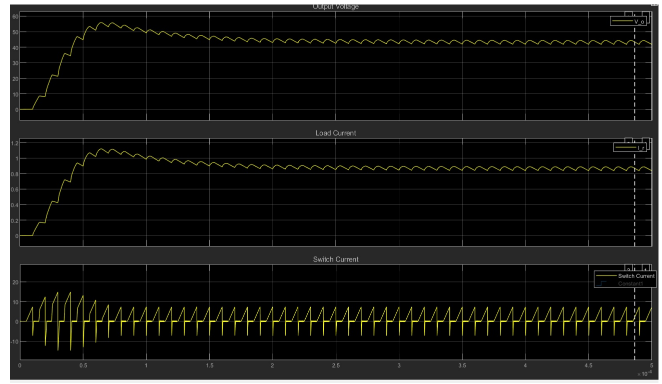

Figure 2. Output Voltage, Load Current, Switch Current

with tools such as spice or Orcad. The error message you might see is that there is an infinite loop of the dependent current source and the voltage. To solve this problem one suggested in the text book of Daniel Hart (2011) is adding a dummy voltage source serial to the magnetizing inductor but this did not work for my version of ltspice or orcad.

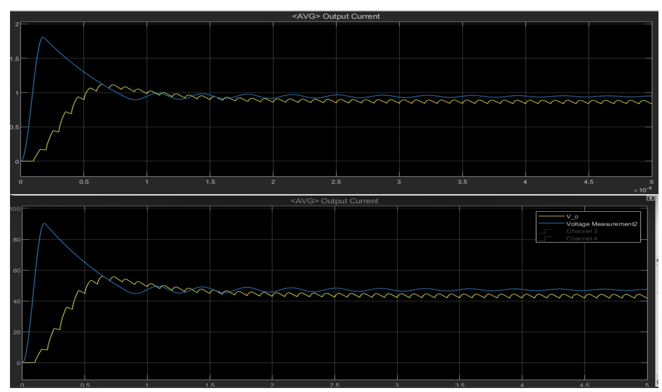

Figure 3. Average model vs Transient model Output Voltage and Current

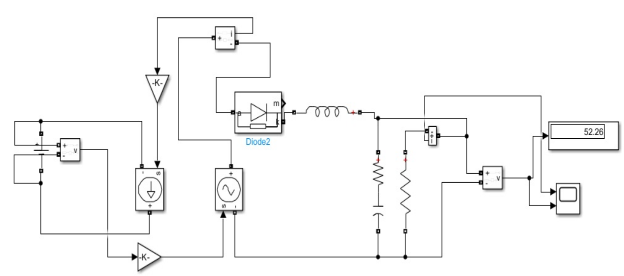

Figure 4. Average model circuit, Gain(K) = D

5. Small Signal Analysis

As mentioned in the introduction this analysis and simulation are valid when we assume the components are ideal or close to ideal. When there are disturbances there would be a difference between the output and the expected value. The small changes in the system may also cause instability which is the main concern. To improve the design we have to make a detailed analysis via a small signal approach to analyze circuits' behavior in a range of frequencies, then by focusing on crossover frequency values we can derive an error compansator.

5.1. Open Loop Transfer Function

Since there are a lot of calculation steps the necessary derivations are uploaded into a repository. Hand-written derivations are included in this : GitHub repository.

The final open loop that I derived produced an unreasonable bode plot thus I utilize the Ekrem Çengelci's (2018) derivation of open loop transfer function (eq.7) of the flyback converter in continuous current mode

$$G_c(s) = \frac{\tilde{v_o}(s)}{\tilde{v_c}(s)} =$$

$$K_{dc} * \frac{(1+\frac{s}{w_{z1}})\cdot(1+\frac{s}{Q_Z \cdot w_Z}+\frac{s^2}{w_z^2})}{(1+\frac{s}{w_{p1}})\cdot(1+\frac{s}{Q_p * w_p}+\frac{s^2}{w_p^2})}$$

To extract the bode plot I built a transfer function in Simulink with described parameters. Due to the long derivation of the terms w_p, Q_p, K_{dc} it won't be described in this section but it is provided in the Matlab code which uses the parameters provided by dc analysis as well as optional parameters for ac analysis such as capacitor resistance, inductor resistance, and switching element resistance. For this analysis, those parameters are as follows. The Bode plot given includes other testing with different parameters.

Properties:	**Features:**
r_l	0.2 Ω
r_{sw}	0.3Ω
r_c	0.4 Ω

TABLE 4. AC ANALYSIS PARAMETERS

6. Type 3 Error Compansator

The Bode plot of the ac equivalent circuits gives critical information about the reaction of the circuits to disturbances. Important knowledge that we need to design a good op-amp-based error compensator is the gain and the phase margin at the crossover frequency which is taken as 0.1 of the switching frequency as suggested in the homework. The plot indicates at crossover frequency the gain of 2.49 dB and the phase margin of -149^o. By using this parameter we can design a type 3 error compensator that will give at least 45^o of phase margin.

$$K = tan(\frac{\theta_{comp}+90^o}{4})^2$$

$$R_2 = \frac{G(f_{co})\cdot R_1}{\sqrt{K}}$$

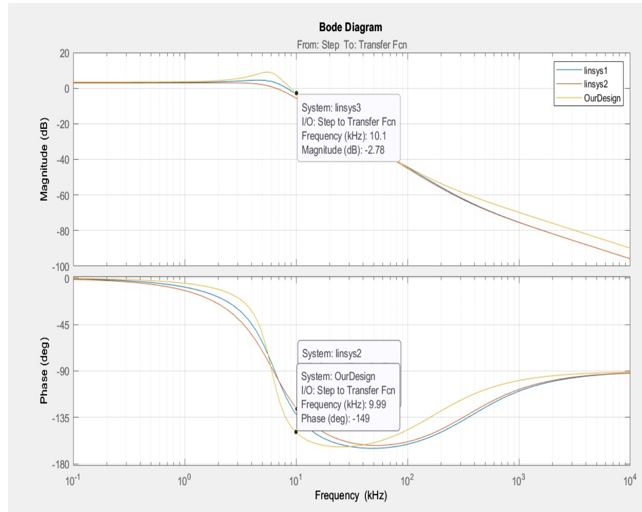

Figure 5. Bode Plot AC analysis of the open loop transfer function

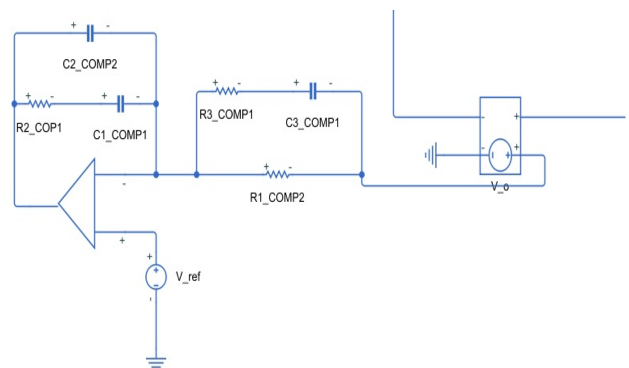

Figure 6. Bode Plot AC analysis of the open loop transfer function

$$C_1 = \frac{\sqrt{K}}{w_{co}\cdot R_2}$$

$$C_2 = (w_{co}\cdot R_2 \cdot \sqrt{K})^-1$$

$$C_3 = \frac{\sqrt{K}}{(w_{co}\cdot R_1)}$$

$$R_3 = (w_{co}\cdot C_3 \cdot \sqrt{K})^-1$$

Properties:	**Features:**
K	9.4
R_2	1.35kΩ
R_3	0.1Ω
C_1	36.4nF
C_2	3.9nF
C_3	48.9nF

TABLE 5. RESULTING PARAMETERS $V_{ref} = 3V$, $f_{co} = 10kHz$, $R1 = 1K\Omega$

The calculations needed to derive parameters of this type 3 error compositor are given in the Matlab code. The only thing the user has to do is to choose arbitrary parameters

(V_{ref}, R1) and enter the phase margin and the gain of the ac analysis of the circuit at the crossover frequency.

```
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%% Error Compansation Type 3 %%%
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%

%%% Enter parameters here

phase_margin = 45;
assert(phase_margin>=45);
V_p = 3;
R_1_comp = 1e3;
fco = 10e3;
%%% @ 10kHz
%%% Bode plot of the ac equvalent circuit
phase_converter = -149;
converter_gain = -2.78;
```

Figure 7. User insertion to the matlab code

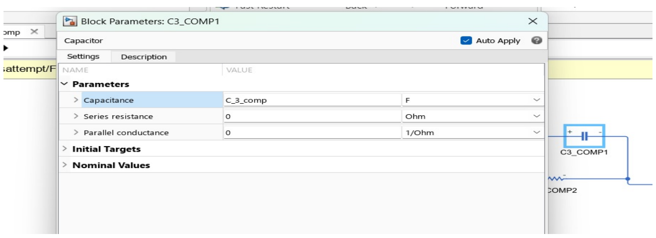

Figure 8. Simulink Uses Calculated Parameters

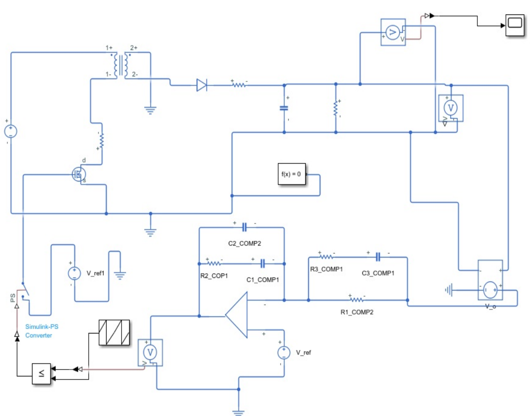

Figure 9. Type 3 Error Compansated Flyback Layout

7. Results

The simulation of the error-compensated circuit did not work at all. Currently, I have no idea why the design did not work maybe one of the parameters of the error compensator was not calculated accurately, or maybe the frequency response of the ac equivalent circuit of the flyback had some error in its structure. Since the circuit worked fine without the error compensator the issue must be on that section of the circuit. When I investigate the voltage output of the op-amp meaning the control output v_c the value was occurring very frequently and rather far from 0 volts. Due to a lack of evaluation time, it is not easy to deduce the error hopefully will find it in the future.

8. Sources

References

[1] D. Hart, *Power Electronics*. McGraw-Hill Education, 2010.

[2] Marian K. Kazimierczuk, *Small-Signal Analysis of Open- oop PWM Flyback DC-DC Converter for CCM*. IEEE, 2002.

[3] SW Lee, *Demystifying Type II and Type III Compensators Using OpAmp and OTA for DC/DC Converters*. Texas Instrumants, Power Management , 2014.

[4] E. Çengelci, *Small Signal Audio Susceptibility Analysis of Flyback Converter With Peak Current Mode Control*. SAKARYA UNIVERSITY JOURNAL OF SCIENCE, 2017.

[5] Ercin Enes, *Github Repo*. Flyback Converter
Simulink and matlab scripts are included as well as the graphs and plots in the form of .mat