



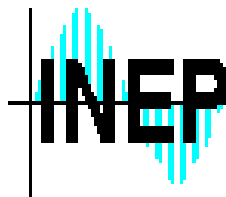
Ecole Navale
Et
Groupe des Ecoles du Poulmic

**PROJET DE FIN D'ETUDES
GENIE ENERGETIQUE**

**ANALYSIS AND DESIGN OF HIGH POWER FACTOR
SINGLE-PHASE RECTIFIER BASED ON THE BOOST
CONVERTER**



**EV2 BELLIER
EV2 SALHI**



Instituto de Eletrônica de Potência (Brasil)

Acknowledgement

We are heartily thankful to Professor Ivo Barbi, whose encouragement, guidance and support from the initial to the final point enabled us to lead this project.

We would like to thank Silvia Helena Pini: this project would not have been possible without her help and explanations.

Moreover, we would like to show our gratitude to Gabriel Tibola for his help during the practical work.

It is a pleasure to thank Antônio Luiz S. Pacheco who built our prototype and made our practical work possible.

We want to thank Eloi Agostini Junior for all his help in all every aspect of practical life in Brazil.

We also thank all the INEP members for their warm welcome.

Ultimately, we thank Professor Jaouen for his help in the English aspect of the report.

Analysis and design of high power factor single phase rectifier based on the Boost converter.

Students : EV2. J. Bellier and A. Salhi, EN 2008
Officier : Capitaine de Frégate Guyot
Type of report : Rapport de Projet de Fin d'Etude (PFE)
University : Federal University of Santa Catarina (U.F.S.C)
Power Electronic Engineering Department (I.N.E.P)
Project chief : Ivo Barbi, Professor in Power Electronic Engineering at the UFSC

ABSTRACT

The goal of this project was to lead an entire study, from the conception until the realisation of a high power factor single-phase rectifier based on the boost converter. This work carried out according to an imposed scope statement and followed the methodology of the department where we did our internship. It is relied on the knowledge of both powers electronic and engineering, since it was expected that the circuit could correct itself to respect the scope statement. After defining the boost operation and its expected results (boost an input voltage from 220V to 400V), all the circuit was designed: it included all the power stage components, voltage and current controllers and power supply. At each step, the entire sizing of the boost circuit and the auxiliary circuits (power supply and controllers) has been validated thanks to simulations done by the software PSIM. We were thus able to understand how the INEP engineers and their know-how met the needs of the industrial.

Once the prototype made and calibrated, it was tested: our system reached an output voltage equal to 400V. A harmonic analysis made it possible to see that our active filter mitigated the harmonics of ranks higher than 1. The dynamic behaviour led the conclusion that our system was well designed: by switching the load from 33% of the nominal load to the full nominal load, the response time is about 100 milliseconds and the output voltage reaches the expected value of 400V. Ultimately, the power factor value is 0.978 (for the nominal load).

RESUME

L'objectif de ce projet est de mener une étude, de la conception à la réalisation d'un redresseur monophasé à haut facteur de puissance basé sur un hacheur parallèle. En faisant appel à des connaissances sur l'électronique de puissance et sur l'asservissement des systèmes, ce projet vise à répondre à un cahier des charges imposé par notre organisme d'accueil. Une fois les objectifs posés, tout le circuit a été dimensionné tant au niveau du hacheur, que des circuits de contrôle et d'alimentation, le but étant de rehausser une tension sinusoïdale initiale de 220V à une valeur finale constante de 400V. Tous nos résultats ont été entrés dans le logiciel de simulation PSIM afin de vérifier la cohérence de ces derniers. Cette méthodologie de travail permet de voir la façon dont les ingénieurs de l'INEP mettent en adéquation leur savoir-faire et les besoins des entreprises avec lesquelles ils travaillent.

Une fois le prototype construit et étalonné, ce dernier a été testé : il a fonctionné du premier coup et les résultats sont ceux attendus. Le survoltage est effectué, et la valeur atteinte est bien de 400V à partir d'un signal 220V/60Hz. Le circuit auxiliaire asservit le système en tension : la tension de sortie reste bien constante et égale à 400V. Une analyse spectrale permet aussi de conclure quant à l'efficacité de notre filtre désigné. Par ailleurs les qualités dynamiques de notre système ont été testées : en faisant varier la charge auquel le système est branché, le temps de réponse est de l'ordre de 100 millisecondes et la tension de sortie atteint sa valeur souhaitée. Le facteur de puissance finale est quasi-unitaire : sa valeur exacte est 0.978.

Key words: Boost, converter, controller, power supply, conception, sizing, power factor, output voltage.

Contents

INTRODUCTION.....	3
I- THEORETICAL ANALYSIS	4
1) POWER STAGE	5
a) <i>Operation stages</i>	5
b) <i>Waveforms</i>	7
c) <i>Static gain</i>	7
d) <i>Boost inductor determination</i>	8
e) <i>Output capacitor determination</i>	9
2) CONTROL STRATEGY	11
a) <i>Current control</i>	11
b) <i>Voltage control</i>	16
II- PROJECT DEVELOPMENT	20
1) SPECIFICATION DATAS	20
2) POWER STAGE COMPONENTS CALCULATION	20
3) SEMICONDUCTOR STRESS	21
a) <i>Semiconductor losses</i>	21
b) <i>Thermal analysis</i>	22
4) INDUCTOR DESIGN.....	24
a) <i>Determination of the core</i>	24
b) <i>Air gap calculus</i>	26
c) <i>Conductors diameter calculus</i>	26
d) <i>Temperature Rise calculus</i>	27
5) TECHNOLOGY OF THE TWO CONTROLLERS	28
a) <i>Technology of the current controller</i>	28
b) <i>Technology of the voltage controller</i>	30
c) <i>Technology of the current sensor</i>	30
d) <i>Technology of the voltage sensor</i>	32
III- INTEGRATED CIRCUIT	33
1) VOLTAGE CONTROL.....	33
2) MULTIPLIER.....	35
3) CURRENT CONTROL	36
4) PWM MODULATOR CONFIGURATION	37
5) OVER CURRENT PROTECTION.....	38
6) SOFT-START CAPACITOR	38
7) MANUFACTURER RECOMMENDATIONS	39
8) SIMULATION ON PSIM.....	39
IV- PRACTICAL WORK.....	41
1. CIRCUIT CONSTRUCTION	41
2. INDUCTOR CONSTRUCTION.....	41
3. OBTAINED PROTOTYPE	42
4. TESTS AND RESULTS.....	42
a) <i>Calibration</i>	43
b) <i>Input signal</i>	43
c) <i>Harmonic aspect</i>	43
d) <i>Output voltage</i>	44
e) <i>Dynamic behaviour of the system and yield</i>	45
CONCLUSION.....	47
APPENDIXES	48
BIBLIOGRAPHY	50

Symbol table

Variable	Specification	Unity	Variable	Specification	Unity
t	Time	Second(s)	L	Inductance	H
f	Frequency	Hz	C ₀	Output capacitor	F
T	Period	s	θ	Normalized time variable	/
ω	Pulsation	rad/s	f _{ci}	Current controller cut off frequency	Hz
s	Laplace variable	s ^{-a}	f _{cv}	Voltage controller cut off frequency	Hz
Z _k	Impedance of the component k	Ω	ω_{cik}	Current controller pulsation (number k)	rad/s
V _k	Voltage across the component k	V	ω_{cvk}	Voltage controller pulsation (number k)	rad/s
I _k	Current in the component k	A	f _{cik}	Current controller frequency (number k)	Hz
R _k	Resistance of the component k	Ω	f _{cvk}	Voltage controller frequency (number k)	Hz
C _k	Capacity of the component k	F	k _i	Current controller gain	SI
μ_0	Vacuum permeability	SI	k _v	Voltage controller gain	SI
f _e	Input source frequency	Hz	C _{cik}	Current controller capacitor (number k)	F
f _s	Switching frequency	Hz	R _{cik}	Current controller resistor (number k)	Ω
T _s	Switching period	s	C _{cvk}	Voltage controller capacitor (number k)	V
P ₀	Output power	W	R _{cvk}	Voltage controller resistor (number k)	V
P ₂	Input power	W	T _k	Temperature in the component k	°C
V ₂	Input voltage	V	T _{xy}	Temperature between components x and y	°C
V ₁	Rectified voltage	V	T _{amb}	Ambient temperature	°C
V ₀	Output voltage	V	X _{pick}	Pick value of the variable X	/
i _L	Inductor current	A	X _{max}	Maximum value of the variable X	/
v _L	Voltage across the inductor	V	X _{sense}	Sensed value of the variable X	/
i _{c0}	Output capacitor current	A	$\langle X \rangle$	Average value of the variable X	/
v _{c0}	Output capacitor voltage	V	ΔX	Variation of the variable X	/
i _{R0}	Resistor current	A			
V _{Dio}	Boost diode voltage	V			
D	Duty cycle	/			

Introduction

Every year, millions of computers are manufactured. It includes many components which use more and more power. Even the engines that control the windscreen wipers of our cars need a very specific electrical power. Consequently it is normal to see that power electronic engineering has increased a lot over the last few decades. And now Direct Current (DC) power supplies are extensively used inside most electrical and electronic appliances in the world, such as in the monitoring of electrical motors and others. One of the mayor applications is the DC/DC converter in order to have a boosted output voltage source, knowing that alternative current is the primary input. It is in this context that the INEP engineers' students work with industrial companies in order to design and build converters for complex electronic systems. Power electronic is present in many domains including on board warships, which is our main concern.

Our project aims at designing a high power factor single-phase rectifier based on the boost converter. By beginning with a boost converter, the goal is to design all the power stage components and a control circuit which enables to maintain a constant and equal output voltage of 400V from a sinusoidal input voltage of 220V/60Hz. Moreover, the sizing of our system has to include the power supply. All the theoretical part is done alongside with two software: PSIM for simulations and MathCAD for computing algebra .When all this design is finished, it will be possible to build the sized system and to control the expected results.

This work is articulated in four phases. First, a theoretical analysis is carried out with the power stages and the control strategy. In a second time, the project development is exposed: calculation of the power stage components, the analysis of the semiconductors, the design of the built inductor and the technology of the controllers. Then, a chapter will be dedicated to the design of a pre-regulator, which is an auxiliary circuit that allows the combination of many functions: our control circuit but also over current systems for example or the current and voltage sensors. Ultimately, experiences and results aim to test our circuit.

I- Theoretical Analysis

The **power factor** of an electric power system is the ratio of the real power flowing to the apparent power, and it is a dimensionless number between 0 and 1 [2]:

$$\text{power factor} = \frac{\text{True power}}{\text{Apparent power}} \in [0; 1]$$

The apparent power is the product of the current and the voltage. The true power is the real delivered power due to the phase between the current and the voltage [2][8].

Consequently, in order to have an economic system, the power factor has to be close to 1.

Moreover, it allows the reduction of the value of the current and then the mitigation of the losses due to the Joule effect [2] [9]. These magnitudes are illustrated in Figure 1.

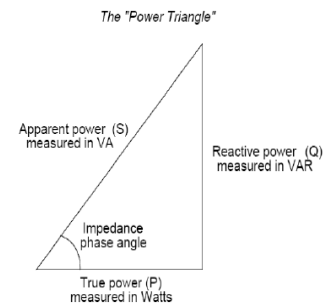


Figure 1 : The power triangle

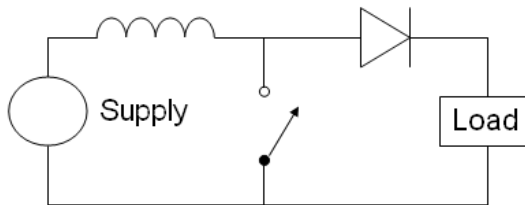


Figure 2: Switching system model

The Boost converter is converting a DC voltage into another DC voltage (usually from an input voltage $220V/f_e=60\text{Hz}$ to an output voltage of 400V). The Boost converter is used when an output voltage higher than the input voltage is required. (= Boost operation).

This switching mode power supply, shown in Figure 2, is frequently used by batteries powered systems. This kind of converters is very efficient. Moreover, it turns on and off very quickly and have low losses. The switching operation is controlled via a variable: the duty cycle D. This duty cycle is defined as a percentage: it is the ratio between the time the switch is closed (so when it conducts) and the period of the system [8].

Without considering capacitor, resistor and inductor, some components are needed: a boost diode, a mosfet and a diode bridge (D1 D2 D4 D4) allows rectifying the input voltage. The figure of the considered system is the Figure 3:

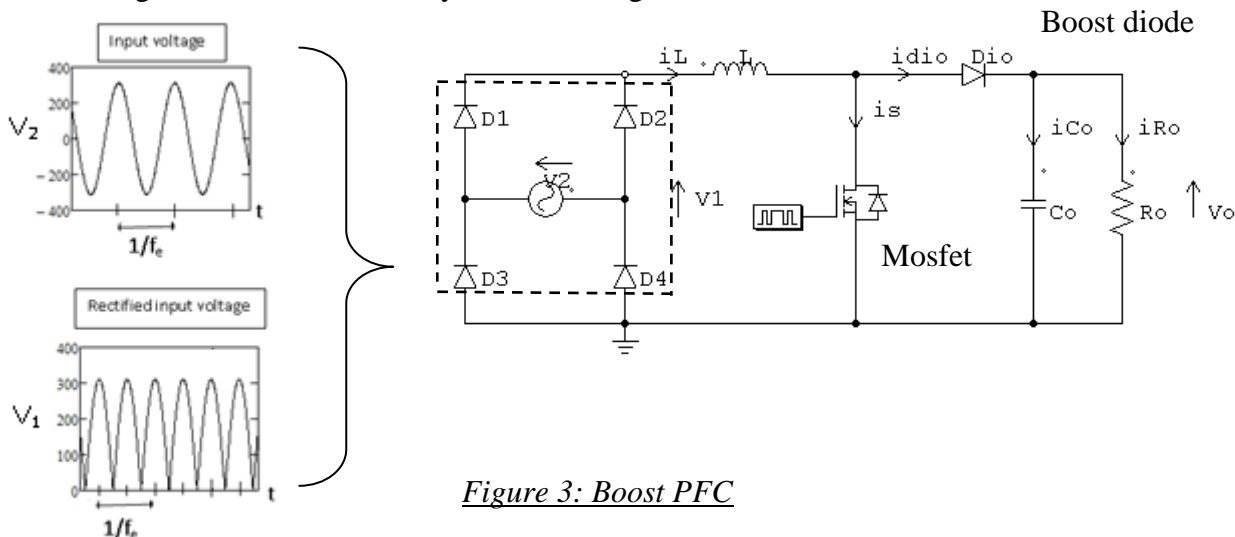


Figure 3: Boost PFC

1) Power stage

a) Operation stages

The input voltage positive half cycle is taken for the analysis, in which diodes 1 and 4 are conducting. For the negative half cycle the diodes 2 and 3 conduct, but the boost operation is the same [3].

T_s is our switching period and D the duty cycle.

First stage of operations: for t between 0 and $D \cdot T_s$

$V_{Di0} = -V_0$ with $V_0 \geq 0 \Rightarrow$ Diode locked. Here we have the mosfet on and the boost diode off. The equivalent electrical circuit is shown in Figure 4.

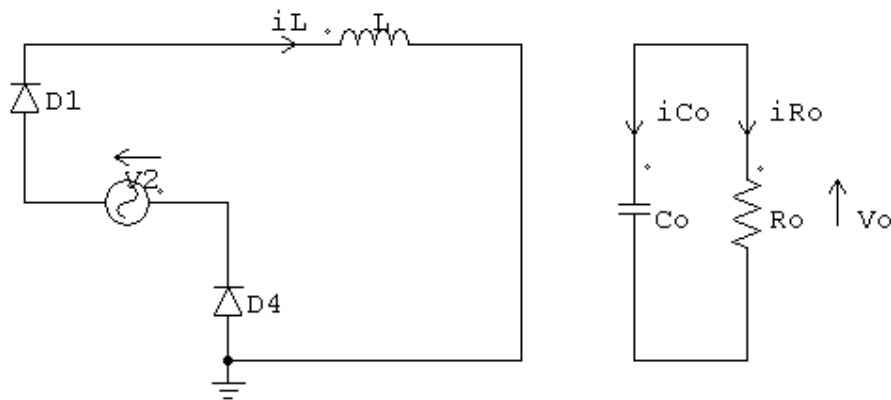


Figure 4: First stage of operations

During this first stage, the sinusoidal source is delivering energy to our solenoid which accumulates it. On the other side, considering that the capacitor is initially charged, this latter delivers its energy to the resistor.

It is possible to write the following equations:

$$\begin{cases} v_L(t) = v_2(t) \\ i_{C0}(t) + i_{R0}(t) = 0 \end{cases} \Rightarrow \begin{cases} L \frac{di_L(t)}{dt} = v_2(t) \\ R_0 C_0 \frac{dv_0(t)}{dt} + v_0(t) = 0 \end{cases}$$

This step is finished when the mosfet is turned off. [9]

Second stage of operations: for t between $D \cdot T_s$ and T_s

In the Figure 5, we have the mosfet turned off and the boost diode on.

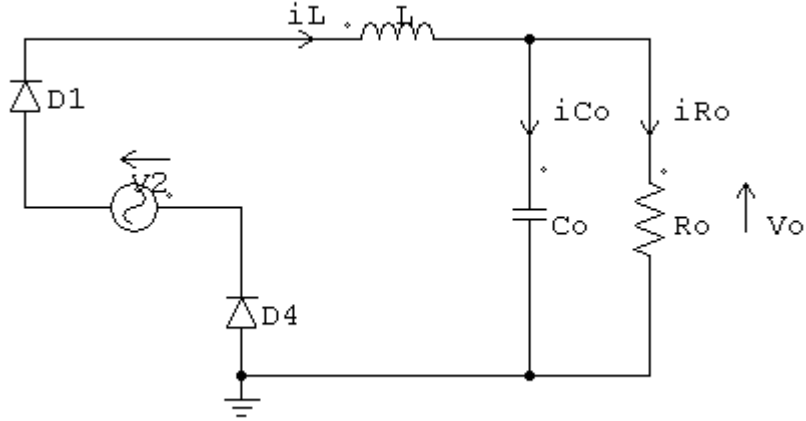


Figure 5: Second stage of operations

The energy stored by the inductor during the first step is returned to the circuit during this second stage. The energy coming from the inductor must be added to the energy coming from the sinusoidal source. The boost operation comes from this superposition of energies. The first step has allowed storing energy and increasing the power received by the output elements, more than if they were directly connected to the sinusoidal source [2].

This time S is open: consequently, this component is submitted to the voltage V_0 .

The following equations can be written:

$$\begin{cases} v_2(t) = v_L(t) + v_0(t) \\ i_L(t) = i_C(t) + i_R(t) \end{cases} \Rightarrow \begin{cases} L \frac{di_L(t)}{dt} = v_2(t) - v_0(t) \\ R_0 i_L(t) = v_0(t) + R_0 C_0 \frac{dv_0(t)}{dt} \end{cases}$$

These two operation stages describe the behaviour of the electrical circuit.

Moreover, these stages allow us to obtain waveforms, and to design our components (inductor and capacitor).

b) Waveforms

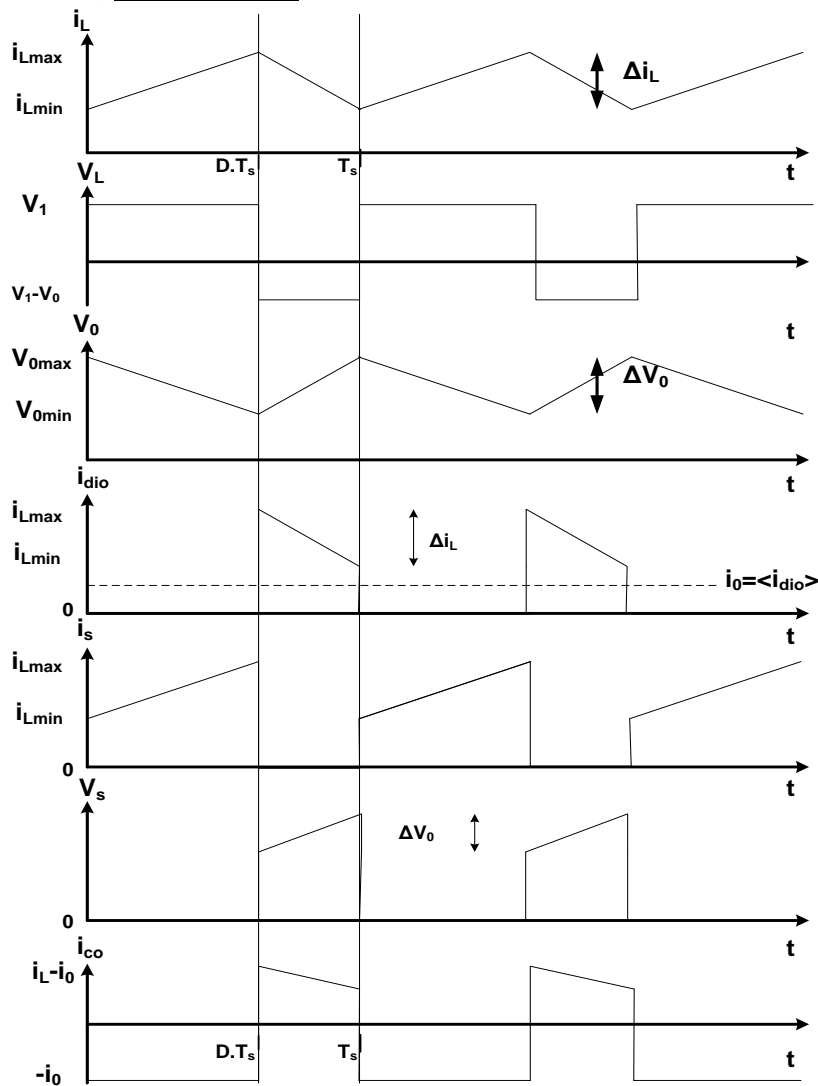


Figure 6: Main waveforms

c) Static gain

For a boost converter working on continuous conduction, we can write [5]:

$$v_0 = v_2 - D \cdot v_2 \quad (\mathbf{I.1})$$

It is possible to define the static gain G_s by the following expression, which is the ratio of the output voltage by the input voltage.

Since the aim of the boost converter is to “boost” the output voltage from the input voltage, this gain has to be higher than one. This fact can be checked on expression **(I.2)**.

That implies that:

$$G_s = \frac{v_0}{v_2(\theta)} = \frac{1}{1-D(\theta)} \quad D \in [0; 1] \quad (\text{I.2})$$

In this case, the input voltage in our converter is the following:

$$v_2(\theta) = V_{2pick} \cdot \sin(\theta) \quad \text{with } \theta = 2\pi f_e \cdot t \in]0; 180[$$

This voltage is rectified by 120 Hertz from the source and has a maximum value equal to the pick value of the system [5]. Moreover, for all the following study, D is a function of θ .

So:

$$D(\theta) = 1 - A \sin(\theta) \quad (\text{I.3})$$

With:

$$A = \frac{V_{2pick}}{V_0}$$

For θ from 0 to 180°, it is possible to draw D in function of this variable. Moreover, we modify the values of A. Consequently, it is obtained the figure 7:

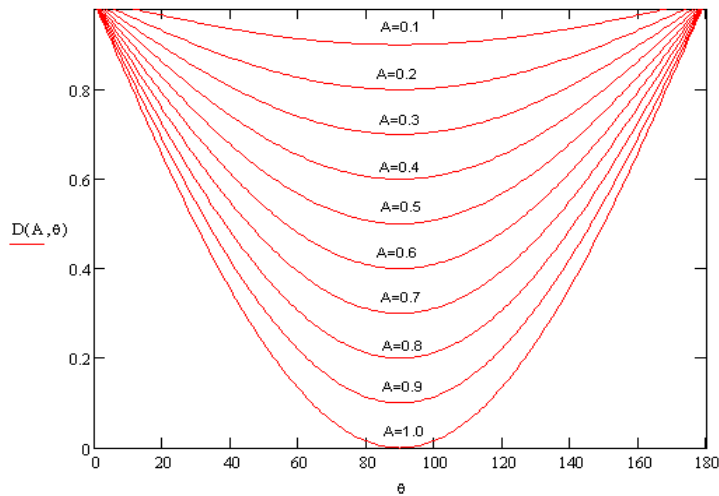


Figure 7: D(theta) illustration

d) Boost inductor determination

To do that, we consider the first stage of our operations.

We have written that: $L \frac{di_L(t)}{dt} = v_2(t)$.

During this $D.T_s$ time, it is possible to write that:

$$v_2(t) = L \frac{\Delta I_L}{D.T_s}$$

With **(I.3)** and $v_2(t) = V_{pick} \cdot \sin(\theta)$, we obtain:

$$\frac{L \cdot \Delta I}{V_{pick} \cdot T_s} = \sin(\theta) - A \sin^2(\theta)$$

Since the member on the right has no unity, it is therefore the same thing for the member on the left. Thus, we can normalize the current by this expression: its illustration is Figure 8:

$$\overline{\Delta I} = \frac{L \cdot \Delta I}{V_{pick} \cdot T_s} = \sin(\theta) - A \sin^2(\theta) \quad [4]$$

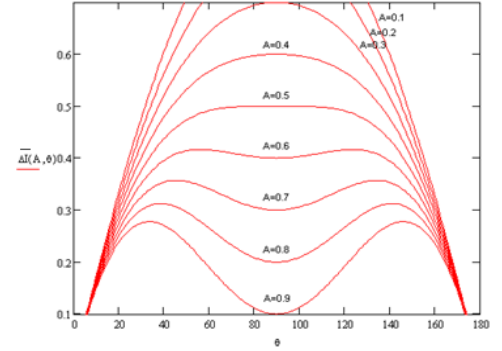


Figure 8: Illustration of the normalized current $\overline{\Delta I}(A)$

From this expression we can deduce our boost inductance.

However, $\overline{\Delta I}$ depends of the variable θ . That is why we have to maximize $\overline{\Delta I}$ to be sure: we consider that this value is a pessimistic one. So:

$$L = \frac{V_{pick} \cdot \overline{\Delta I}^{max}}{\Delta I \cdot f_s} \quad \text{(I.4)}$$

e) Output capacitor determination

The capacitor design has to respect three criteria [6]:

- the voltage ripple
- what we call the hold-up time
- the effective current

i) Voltage ripple

The output capacitor is designed for the 120 Hz ripple limitation. As $V_2(\theta)$ is the input source of the boost converter, the output voltage will vary with this input voltage.

So if: $V_c = X_c \cdot i_c \Rightarrow \Delta V_c = X_c \cdot i_c$

Where:

- X_c is the impedance of the capacitor $X_c = \frac{1}{2\pi f C_0}$, here $f=2 \cdot f_e$

- ΔV_{cp} is the variation of the voltage at the capacitor. We are using to take this value as a percent of the output voltage.

$-i_{cp}$ is the value of the current pick in the capacitor.

$$\text{Then } C_0 = \frac{i_{cp}}{4\pi f_e \Delta V_{cp}}$$

To determine the final value of C_0 we need it as independent function of the capacitor pick current. To reach this goal we have to consider the power in the circuit.

The input power:

$$\left. \begin{aligned} V_2(\theta) &= V_{pick} \cdot \sin(\theta) \\ I_2(\theta) &= I_{pick} \cdot \sin(\theta) \\ P_2(\theta) &= V_2(\theta) \cdot I_2(\theta) \end{aligned} \right\} \Rightarrow P_2(\theta) = V_{pick} \cdot I_{pick} \cdot \sin^2(\theta)$$

For the output voltage we consider the output source as a perfect source of voltage (we don't take into account ΔV_{cp}):

$$\text{So } P_0(\theta) = V_0 \cdot i_c(\theta)$$

Supposing we do not have losses in the circuit we can write:

$$P_2(\theta) = P_0(\theta) \Leftrightarrow V_0 \cdot i_c(\theta) = V_{pick} \cdot I_{pick} \cdot \sin^2(\theta)$$

$$\Rightarrow i_c(\theta) = \frac{V_{pick}}{V_0} \cdot I_{pick} \cdot \sin^2(\theta)$$

$$\text{Now regarding the average value: } \langle P_0 \rangle = \langle P_2 \rangle = \langle V_{pick} I_{pick} \sin^2(\theta) \rangle = \frac{V_{pick} I_{pick}}{2}$$

$$\text{So } i_c(\theta) = 2 \cdot \frac{\langle P_0 \rangle}{V_0} \cdot \sin^2(\theta) = 2 \cdot \frac{\langle P_0 \rangle}{V_0} \left[\frac{1}{2} - \frac{1}{2} \cos(2\theta) \right]$$

$$\text{Then } i_{cp} = \frac{P_0}{V_0}$$

Ultimately we find the value of the output capacitor:

$$\boxed{C_0 = \frac{P_0}{4 \cdot \pi \cdot f_e \cdot V_0 \cdot \Delta V_{cp}}} \quad (\text{I.5})$$

ii) Hold-up time

This hold-up time in Figure 9 is defined as following: even if the input source is off, the capacitor will have to maintain the output voltage V_0 in a certain value and this during a time t_{HT} which is called hold-up time.

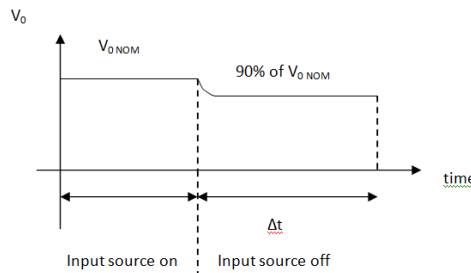


Figure 9: Hold-up time role

Usually, it is: $t_{HT} = \frac{T_{input}}{2}$. Moreover we are supposed to have 90% of the output voltage when the input source goes off. We write $V_0' = 0.9 \cdot V_{0NOM}$.

The energy that the capacitor gives to compensate the difference between V_{0NOM} and V_0' during the hold-up time is equal to the input power. So we can write:

$$\frac{1}{2} C_{HT} \cdot (V_{0NOM}^2 - V_0'^2) = t_{HT} \cdot P_0 \Rightarrow \boxed{C_{HT} = \frac{2P_0 \cdot t_{HT}}{(V_{0NOM}^2 - V_0'^2)}} \quad (\mathbf{I.6})$$

If the value of this capacitor is higher than the one previously obtained for the voltage ripple criteria, it must be therefore chosen.

iii) Effective current

By the simulation on PSIM, it is possible to value the capacitor effective current. This value must be checked to make sure that these two values are compatibles.

2) Control strategy

We expect to have a constant output voltage from the Boost converter. Consequently it is possible to include a current and a voltage control to obtain what we want.

In our control and regulation, there are many variables to study. The output, the reference signal and the error variable are very important, but the noise variable and the disturbance signal have to be considered. Without noises or disturbances, the output has to be equal to the reference signal. The idea is to use a feedback to create a circuit that automatically fits the duty cycle (a voltage is sent to the mosfet to adjust the duty cycle) as necessary to control input current and to obtain the desired output voltage without noises or disturbances. This will be done by the principle of the PWM (Power width modulation cf figure 10).

a) Current control

i) Modeling

A feedback can control this current since we have a relation between this variable and the duty cycle as illustrated in the figure 10.

Physically speaking, we sense the current I_{sense} just after the inductor component; then we compare this value to a reference current I_{ref} . Once this operation done, if the error is not equal to zero, a controller (that is what we have to design) launches a command V_{ci} through a comparator to correct the difference. This operation is done on the MOFSET by the gate source signal, what we call V_{gs} (voltage information commanding the opening and the closing operations of the Mosfet).

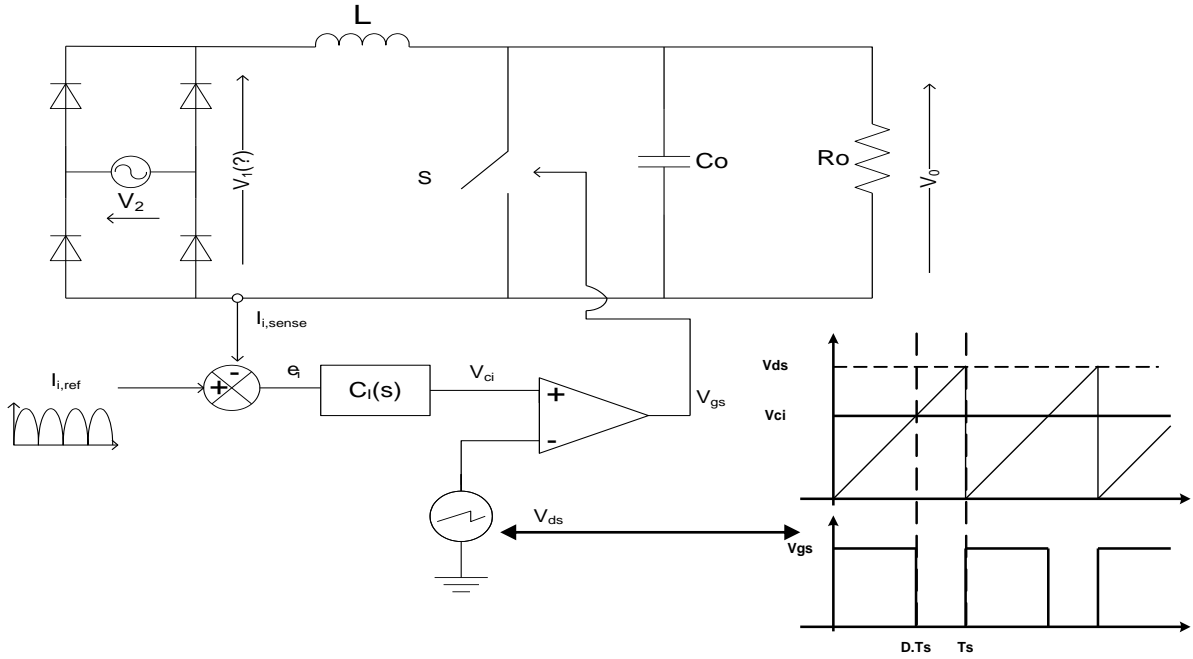


Figure 10: The current control and PWM principle

This feedback control can be scheduled by the following control engineering in Figure 11:

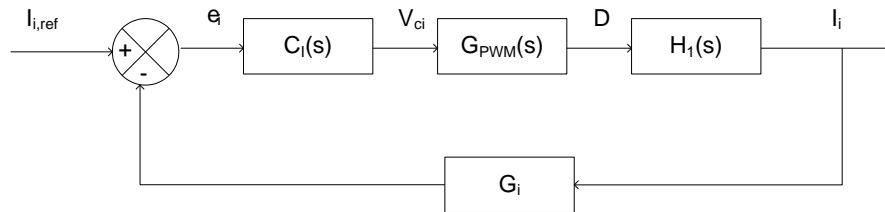


Figure 11: Current control block diagram

- ✓ G_i is the sensor gain between the reference signal and the real value of the current. Consequently, its expression is the following:

$$G_i = \frac{I_{L,ref}}{I_L}$$

- ✓ G_{PWM} is the PWM modulator gain between the duty cycle and the control signal(voltage):

$$G_{PWM} = \frac{D}{V_{CV}} = \frac{1}{V_{ds}}$$

- ✓ $H_i(s)$ is the transfer function describing the current model.

- ✓ $C_i(s)$ is the transfer function of the current controller (for correction) that has to be designed.

Moreover, we need a model for the current.

For this study with transfer functions, the variables that we can measure thanks to practical tools are the average variables; therefore we have to work with average variables. For example, when we write V_L , it must be read as his average value (with big letters).

For a switching period, $v_1(\theta)$ is considered constant since: $f=120\text{Hz} \ll f_s=40\text{kHz}$.

With the average considerations, we can write:

$$V_L = V_1 - V_0(1 - D)$$

(Average values with our new notations)

Small disruptions must be included for the variables in analysis. We consider small behind the values of our variables; consequently, it is possible to write that:

- $D=D+\Delta D$
- $I_L=I_L+\Delta I_L$

And injecting in our average equation, with $V_L = L \frac{dI_L}{dt}$

$$L \frac{d(I_L + \Delta I_L)}{dt} = V_1 - V_0(1 - (D + \Delta D))$$

This can be written with the following simplifications:

$$L \frac{d(I_L)}{dt} + L \frac{d(\Delta I_L)}{dt} = V_1 - V_0(1 - D) + V_0 \cdot \Delta D$$

This implies that:

$$L \frac{d(\Delta I_L)}{dt} = V_0(\Delta D)$$

Consequently, in the Laplace domain, the transfer function modeling the current is given by:

$$\boxed{H_i(s) = \frac{\Delta I_L(s)}{\Delta D(s)} = \frac{V_0}{sL}} \quad \text{(I.7)}$$

Figure 12 shows its Bode diagram:

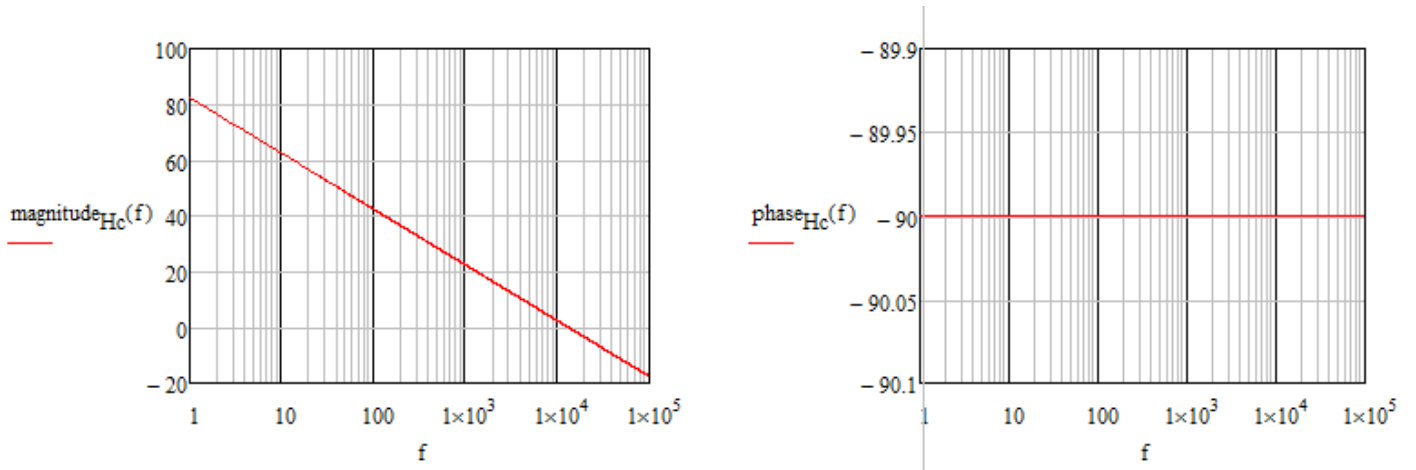


Figure 12: Bode diagram of the current model

ii) Design

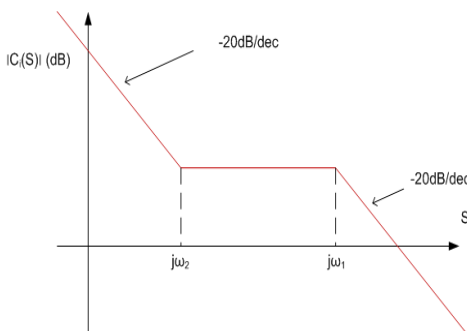
Since our reference signal is in low frequencies, and since we want to obtain the better possible stability, we have to integrate a PI control with a filter in our design. This tool makes it possible to delete noises that can be prompted by the switching effects. These noises are mitigated for frequencies higher than the cut off one [1].

That is why this frequency is chosen by the following expression:

$$f_{ci} = \frac{f_s}{10}$$

It enables to maintain an adequate phase margin. Moreover, PI control is used to increase the low frequency loop gain. At frequencies less than f_{ci} , the PI compensator improves the rejection of disturbances [1].

Then, we have to consider the higher frequencies. These frequencies imply disturbances which are not desirable for our system. These disturbances have to be deleted. This is made by a low pass filter in high frequencies. Consequently, our controller has to have the behaviour shown in Figure 13.



The current controller has the previous behaviour. By a mathematical analysis, it is possible to write the transfer function of this component:

$$C_i(s) = k_i \cdot \frac{s + \omega_1}{s \cdot (s + \omega_2)} \quad (\text{I.8})$$

Let's analyse this expression.

We have a zero which allows the system stability. The origin pole is for low frequency high gain. The second pole is for attenuation in high frequency. We define the zero and the second pole from the cut off frequency:

Figure 13: Basic representation of the searched controller

$$\checkmark \omega_{ci1} = 2\pi f_{ci1} = \frac{2\pi f_{ci}}{4}$$

$$\checkmark \omega_{ci2} = 2\pi f_{ci2} = 2 \times 2\pi f_{ci}$$

Figure 14 shows its Bode diagram:

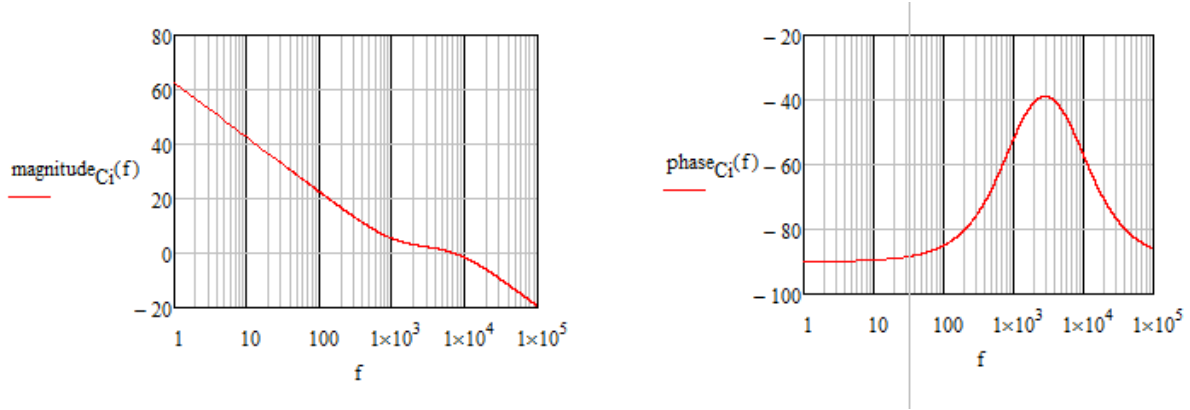


Figure 14: Bode diagram of the current corrector

iii) Circuit analysis

With a non compensated system, the open loop transfer function called T_{ui} is:

$$T_{ui}(s) = G_i \cdot G_{PWM} \cdot H_i(s) = G_i \cdot G_{PWM} \cdot \frac{V_0}{L \cdot s}$$

Which is $H_1(s)$ but just multiplied by two more gains such as shown in Figure 15.

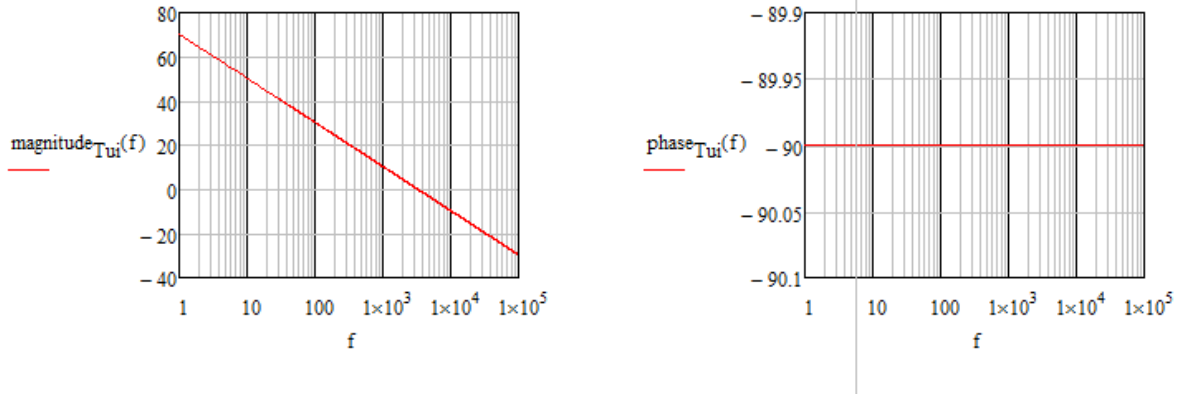


Figure 15: Bode diagram of the non compensated system

Now, we have to integrate the current controller designed previously in our loop, in order to obtain the compensated system transfer function:

$$T_i = T_{ui} \times C_i(s) = G_i \cdot G_{PWM} \cdot H_i(s) \cdot C_i(s)$$

So:

$$T_i(s) = G_i \cdot G_{PWM} \cdot \frac{V_0}{L \cdot s} \cdot k_i \cdot \frac{s + \omega_1}{s \cdot (s + \omega_2)} \quad \text{(I.9)}$$

The Figure 16 illustrates its Bode diagram:

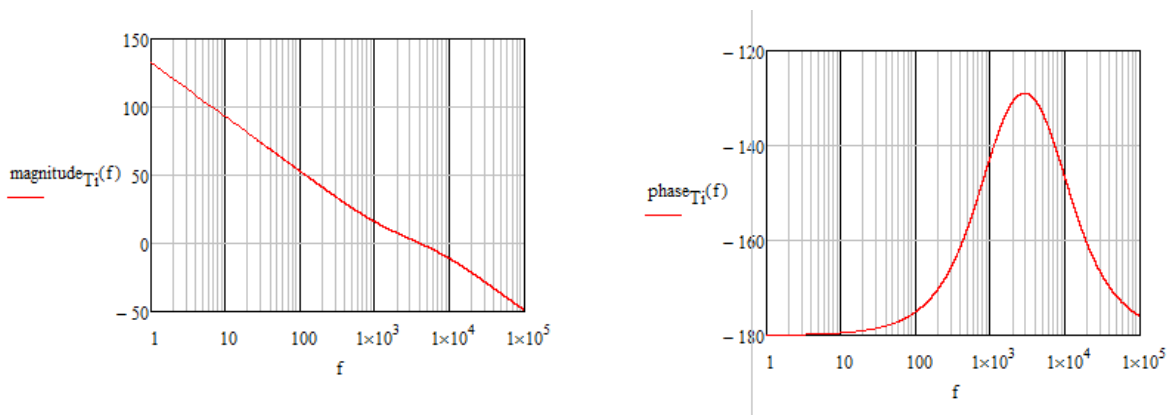


Figure 16: Bode diagram of the compensated system

Controller gain calculation

At the cut off frequency, it must be attended to have: $|k_i \times C_l(f_{ci}) \times T_{ui}(f_{ci})| = 1$

So:

$$k_{idB} = -20 \log (C_l(f_{ci}) \times T_{ui}(f_{ci}))$$

And :

$$k_i = 10^{\frac{k_{idB}}{20}}$$

b) Voltage control

i) Modeling

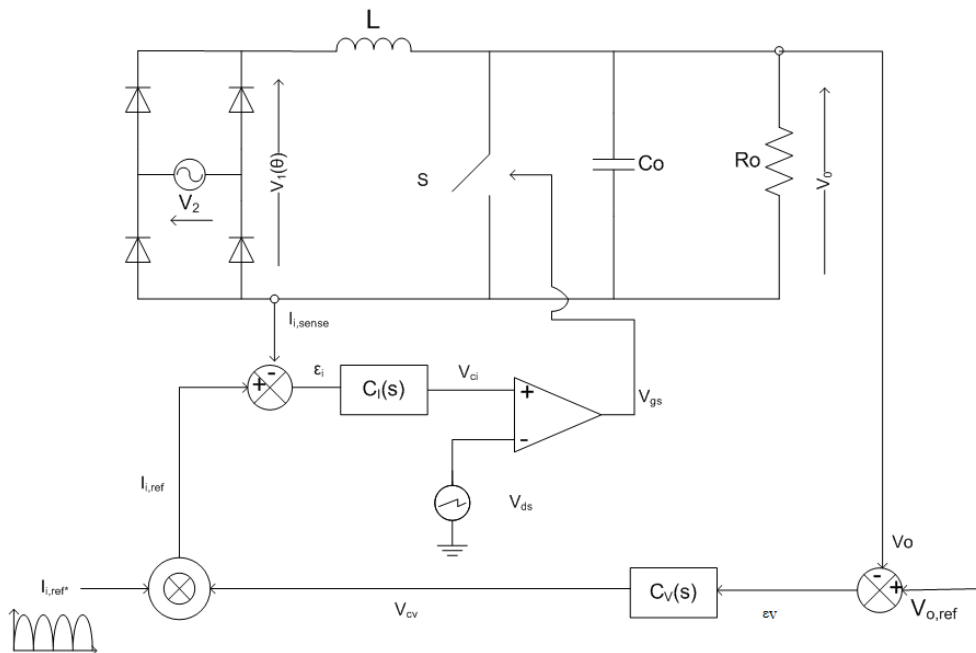


Figure 17: Voltage control strategy

When the current is controlled by the feedback we have to make voltage control engineering, by the same way (Figure 17). Then with an input command on the voltage ($V_{0,ref}$) we can obtain the input command on the current and then the duty cycle. Consequently the first feedback control of the current is going to be put in the second feedback control of the voltage.

Physically speaking, here, the output voltage is sensed and compared to the reference voltage. There is an error ε_v treated to have V_{cv} . This previous is multiplied by I_{Ref} to obtain the current reference. Then, it is included in the current control in order to rectify the error and finally have the constant wanted output voltage. The switch instruction to accumulate energy in the order is given to the mosfet by V_{GS} .

This feedback control can be scheduled by the following control engineering in Figure 18:

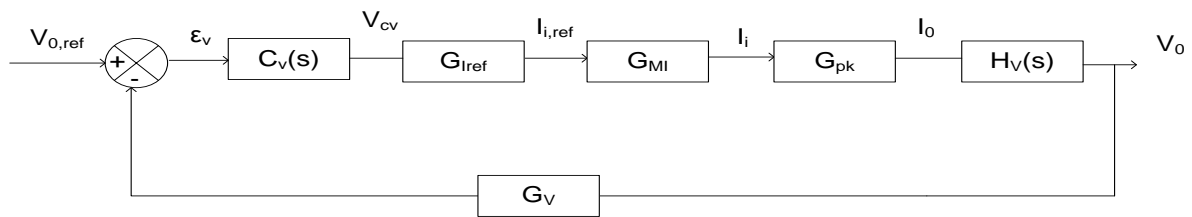


Figure 18: Voltage control block diagram

- ✓ $H_v(s)$ is the transfer function for the voltage model.
- ✓ $C_v(s)$ is the transfer function of the voltage controller.
- ✓ Firstly, G_{Iref} is the gain between the voltage information outputted of the voltage controller and the reference current by the loop of current control(=multiplication):

$$G_{Iref} = \frac{I_{i,ref}}{V_{cv}}$$

- ✓ Secondly, by considering that the control loop of the current control performs its work quickly, we can consider it with the following gain:

$$G_{mi} = \frac{I_i}{I_{i,ref}} = \frac{1}{Gi}$$

- ✓ Then, G_{CI} is the gain that allows supplying the current of reference on the inductance current.
- ✓ Finally, the gain G_{pk} is the gain between the pick current into the inductor and the average value of the output current. By the power conservation:

$$G_{pk} = \frac{A_0}{2}$$

✓ G_v is the sensor gain between the reference signal and the real value of the voltage.

A model for what we call H_v is necessary.

By considering an analysis during an input source period, we can write that:

$$i_0(t) = \frac{v_0(t)}{R_0} + C_0 \frac{dv_0}{dt}$$

Or:

$$\frac{i_0(t)}{C_0} = \frac{v_0(t)}{R_0 \cdot C_0} + \frac{dv_0}{dt}$$

In the Laplace domain, by considering average values as we have said before:

$$H_v(s) = \frac{V_0(s)}{I_0(s)} = \frac{R_0}{1 + R_0 C_0 s} \quad (\text{I.10})$$

We are still working with average values. Its Bode diagram in Figure 19:

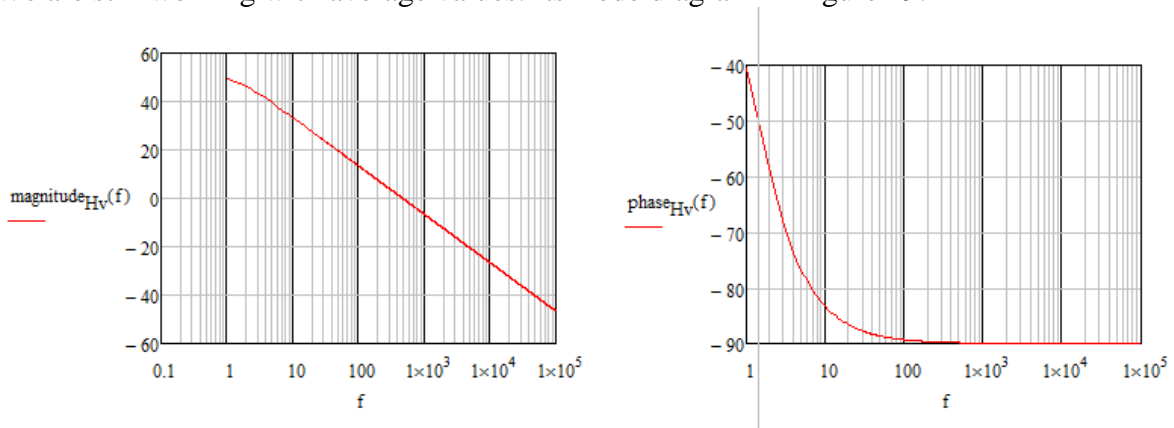


Figure 19: Voltage model Bode diagram

ii) Design

We proceed as we done previously with the current controller. To design our voltage controller we have to define the new cut-off frequency. But the voltage system cut-off frequency must be much slower than that of the current system one [1].

The voltage controller must not compensate the 120 Hz ripple of the output voltage. This ripple limitation is done by the output capacitor. That is why the cut off frequency must be smaller than 120hz.

That is why we can take:

$$f_{cv} = \frac{2 * f_e}{10}$$

The second pole and the zero of our system are defined as following:

$$f_{cv1} = \frac{1}{4} \cdot f_{cv} = \frac{\omega_{cv1}}{2\pi}$$

$$f_{cv2} = 4 * f_{cv} = \frac{\omega_{cv2}}{2\pi}$$

We search the same form than previously in the equation (I.8):

$$C_v(s) = k_v \cdot \frac{s + \omega_{cv1}}{s \cdot (s + \omega_{cv2})}. \quad (\text{I.11})$$

iii) Circuit analysis

With a non compensated system, the open loop transfer is the:

$$T_{uv}(s) = G_v \cdot G_{lref} \cdot G_{mi} \cdot G_{pk} \cdot H_v(s) = G_v \cdot G_{lref} \cdot G_{mi} \cdot G_{pk} \cdot \frac{R_0}{1 + R_0 C_0 s}$$

It is our function H_v multiplied by 4 more gains, as illustrates Figure 20.

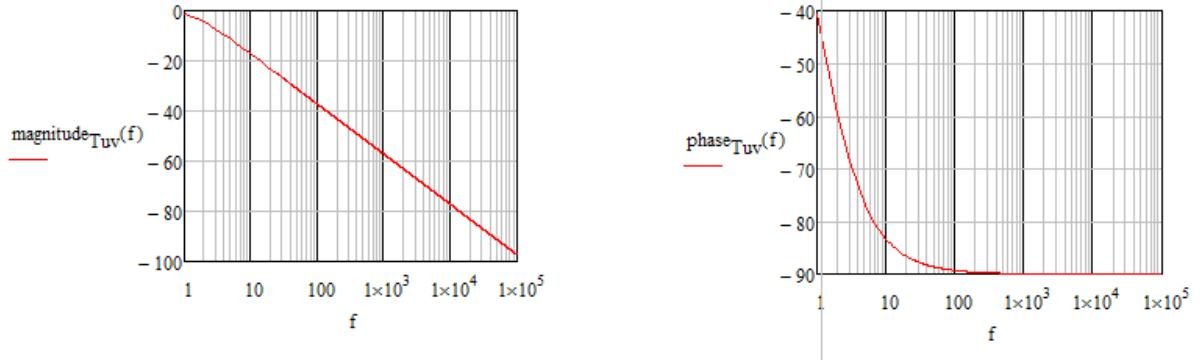


Figure 20: Non compensated system block diagram

When we include the voltage controller in the loop we obtain the following compensated open loop transfer function (Figure 21):

$$T_v(s) = C_v(s) \times T_{uv}(s)$$

So:

$$T_v(s) = k_{cv} \cdot \frac{(s + w_{v1})}{s \cdot (s + w_{v2})} \times G_v \cdot G_{lref} \cdot G_{mi} \cdot G_{pk} \cdot \frac{R_0}{1 + R_0 C_0 s} \quad (\text{I.12})$$

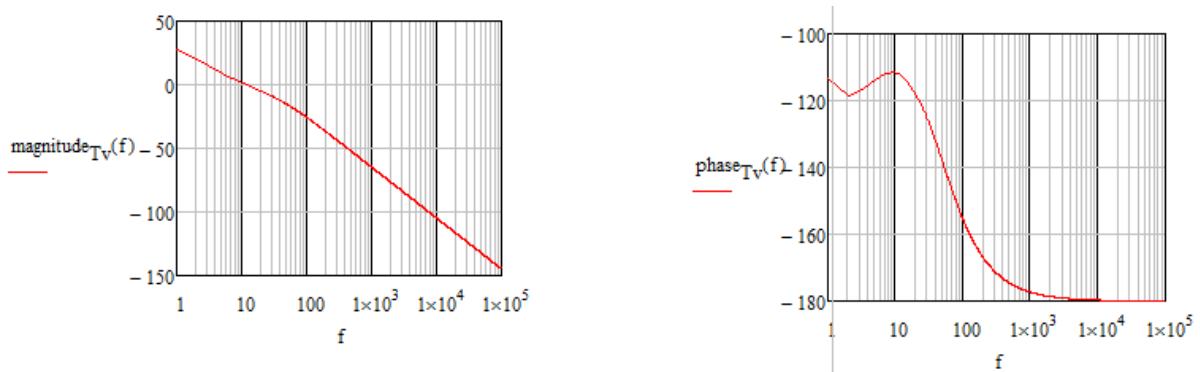


Figure 21: Compensated system block diagram

Every Power stage components and the two controllers are designed.

II- Project development

After a first chapter where the boost components were theoretically expressed and the control strategy exposed, this second focuses on the project development.

From a scope statement, the values of these components are evaluated. Moreover, this chapter takes into account the thermal aspect in the semiconductors, presents the inductor from its value (with a wire length and a number of turns), and the two controllers with physical components and their associated sensors.

1) Specification datas

We take the following values for the input variables, it defines our scope statement which is required by the INEP:

- $f_s = 40\text{kHz}$.
- $f_e = 60\text{Hz}$. We note $\theta = 2\pi f_e \cdot t = \omega_e \cdot t$.
- $v_2(\theta) = 220\sqrt{2} \sin(2\pi f_e \cdot t)$ so $V_{2pick} = 311\text{V}$.
- $P_0 = 400\text{W}$.
- $V_0 = 400\text{V} \Rightarrow R_0 = 400\Omega$.
- $\Delta V_0 = 0.04$
- $\Delta I_0 = 0.2$

2) Power stage components calculation

Inductor: According to **(I.4)**:

$$L = \frac{V_{2pick} \cdot \overline{\Delta I}^{max}}{\Delta I \cdot f_s}$$

Numerically: $L = 4.84 \text{ mH}$

Capacitor: According to **(I.5)**:

$$C_0 = \frac{P_0}{4\pi \cdot f_e \cdot V_0 \Delta V_{cp}}$$

Numerically: $C_0 = 165 \mu\text{F}$

By the expression **(I.6)**, we have to check if the hold-up capacity is higher or not than the value calculated just:

$$C_{HT} = \frac{2P_0 \cdot \Delta t}{(V_{0NOM}^2 - V_0'^2)}$$

Numerically: $C_{HT} = 219.3 \mu\text{F}$.

So we choose this value as we explained previously in the first part.

In the INEP store, the most closed value is 340 μ F (by associating two capacitors of 680 μ F).

Ultimately, we have to design our system with: $C_0 = 340\mu F$.

3) Semiconductor stress

a) Semiconductor losses

i) Mosfet

Power losses (P_{mosfet}) in any component operating in the switch mode can be divided in three groups [6]:

- conduction losses P_{cond} .
- switching losses P_{sw} .
- blocking(leakage) losses (P_b), usually neglected.

That is why it is possible to write:

$$P_{\text{mosfet}} = P_{\text{cond}} + P_{\text{cond}} + P_{\text{sw}} \approx P_{\text{cond}} + P_{\text{cond}} : \text{total mosfet losses}$$

Each term has to be evaluated.

Conduction losses

These losses are given by the following expression:

$$P_{\text{cond}} = R_{\text{Don}} \cdot I_{\text{seff}}^2$$

With:

- ✓ R_{Don} the resistance of the mosfet when is one is On given in the data sheet
- ✓ I_{seff} the root mean square value of the mosfet on-state current evaluated on PSIM.

Numerical result:

$$P_{\text{cond}} = (0.15\Omega) \times (1.064)^2 = 0.17 W$$

Switching losses

The following expression gives the switching losses:

$$P_{\text{sw}} = \frac{f_s}{2} \cdot (t_r + t_f) \cdot I_{\text{smax}} \cdot V_{\text{smax}}$$

With:

- ✓ f_s the switching frequency.
- ✓ t_r the rise time (time for the mosfet to conduct) given by the data sheet.
- ✓ t_f the fall time (time for the mosfet to block) given by the data sheet.
- ✓ I_{smax} the maximum value of the current through the mosfet which is given by PSIM.
- ✓ V_{smax} the maximum value of the voltage through the mosfet which is given by PSIM.

Numerical result:

$$P_{sw} = \frac{1}{2} \cdot (40 \cdot 10^3) \cdot (115 \cdot 10^{-9} + 53 \cdot 10^{-9}) \cdot (2.9) \cdot (404) = 3.94 \text{ W}$$

To conclude, mosfet losses are the following:

$$P_{mosfet} = P_{cond} + P_{sw} = 4.11 \text{ W}$$

ii) Boost diode losses

The switching losses for the diode can be neglected. By the same token than previously:

$$P_{Dio} = P_{Dcond} = V_t I_{med} + r_t I_{eff}^2$$

Usually : $r_t I_{eff}^2 \ll V_t I_{med}$. Consequently:

$$P_{Dio} = V_t I_{med}$$

Where:

- ✓ V_t is the forward voltage, given by the data sheet.
- ✓ I_{med} is the average value of the current through the diode, given by PSIM.

Numerical results:

$$P_{Dio} = 1.5 \text{ A} \cdot 1.25 \text{ V} = 1.875 \text{ W}$$

iii) Full bridge diodes losses

It is exactly the same token and the same theoretical expression.

$$P_{bridge} = 1.3 \text{ A} \cdot 0.75 \text{ V} = 0.975 \text{ W}$$

⇒ **TOTAL SEMICONDUCTOR LOSSES = 6.9 W.**

b) Thermal analysis

The mosfet and the diodes losses were previously calculated. Moreover, all our components are chosen according to different criteria (effective current, voltage ripple, function, economic considerations...). In the datasheets, we can read the junction temperature of these components.

The losses previously calculated will become heat losses and in a smaller proportion electrical losses. Consequently, it is necessary to analyze the junction temperature of these components to be sure that they will not overheat and burn. Practically speaking, it is very important for us to design a safe system and to make sure that it will not burn.

And in the case that the junction temperature is higher than the nominal value given in the datasheets of the components, we have to consider a heat sink and add it in the system to evacuate this power and preserve our components from burning.

We can use the following notations:

- T_j : junction temperature
- T_c : case temperature
- T_d : heat sink temperature
- T_{amb} : ambient temperature
- R_{jc} : thermal resistance between junction and case
- R_{cd} : thermal resistance between heat sink and case
- R_{damb} : thermal resistance between heat sink and ambient atmosphere
- P : power losses

The situation can be illustrated by the figure 22:

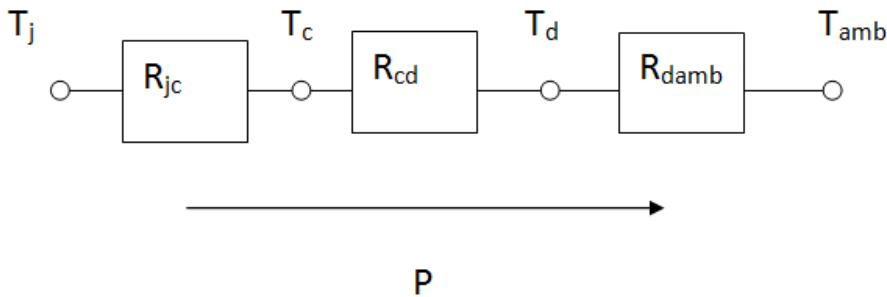


Figure 22: Thermal junction model

Consequently, the equivalent resistance between the junction point and the ambient atmosphere is given by:

$$R_{jamb} = R_{jc} + R_{cd} + R_{damb} \quad (\text{II.1})$$

With an analogy with an electrical circuit, considering that the power is the current and the gradient of temperature as the potential difference, we can write:

$$T_j - T_{amb} = R_{jamb} \cdot P$$

$$\Rightarrow T_j = R_{jamb} \cdot P + T_{amb}$$

First of all junction temperature has to be checked if it is lower or not to the datasheet value to conclude if a heat sink is necessary. So, with no heat sink:

$$T_j = (R_{jc} + R_{ca}) \cdot P + T_{amb}$$

If this value is higher than the datasheet value (T_{jmax}), a heat sink must be used. The maximum temperature must be specified T_{jmax} . The necessary junction ambient must be the following:

$$R_{jamb} = \frac{T_{jmax} - T_{amb}}{P}$$

Injecting that in the equation (II.1), we can deduce the heat sink resistance to use:

$$R_{damb} = \frac{T_{jmax} - T_{amb}}{P} - R_{ja} - R_{cd} \quad (\text{II.2})$$

The heat sink can be chosen; moreover, to guarantee the maximum specified junction temperature we have to choose an equal or a smaller value from the obtained heat sink resistance. The values of the two resistances in the final expression are given in the catalogue.

Consequently, it is necessary to do that for each component (mosfet, boost diode, diodes bridge). We take the following values for the data:

$$T_{amb}=40^{\circ}C, T_{jmax} = 120^{\circ}C.$$

By having the thermal resistances of the components in the datasheets, we can evaluate the T_j resulting temperatures for each component:

- $T_{j, mosfet} = 42.053^{\circ}C$
- $T_{j, boost diode} = 48.25^{\circ}C$
- $T_{j, full bridge} = 59.5^{\circ}C$

All these values are lower than the T_{jmax} temperature.

We can conclude that for each component, heat sink is not necessary. However, we will add a heat sink to make sure that if anything happen, our system will be able to face thermal difficulties.

4) Inductor design

a) Determination of the core

As the capacitor was chosen from the characteristics of the circuit, we are going to design the inductor and build it. First we have to find its characteristics from the equations of magnetic and circuits. We usually use a E-E kind of core drawn in Figure 23:

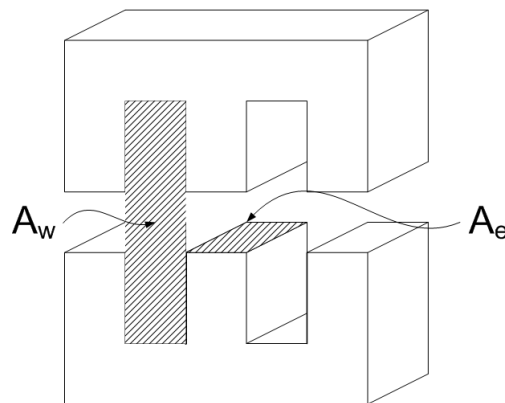


Figure 23: Geometric core

The material used for the core is Ferrite because we are working on high frequency [7]. In fact it has low saturation flux density (0.25T) and low strength to mechanical shocks.

$$\text{So } B_{max} = 0.25T.$$

Then we have to define:

A_e : Middle leg area

A_w : reel window area

A_p : cooper winding cross- sectional area

v_e : core volume

N : Number of turn

$l_{winding}$: The length of an average conductor circle

The flux gives $\Delta\Phi=\Delta B.A_e$

And the core equations give:

$$\begin{cases} v(t) = N \cdot \frac{d\Phi(t)}{dt} = N \cdot \frac{\Delta\Phi}{\Delta t} \\ v(t) = L \cdot \frac{di(t)}{dt} = L \frac{\Delta i}{\Delta t} \end{cases}$$

For the determination of our inductor we take the maximal value of current and magnetic flux.

$$N = \frac{L.I_{\max}}{B_{\max} A_e} \quad (\text{II.3})$$

Let us consider K_w the winding factor in transformer due to isolation between different

windings. Due to the geometry of the core it is define by $K_w = \frac{A_p}{A_w}$ (II.4). We usually take the value of 0.7 (70%) [6].

Still, the current density is defined by: $J_{\max} = \frac{N.J_{eff}}{A_p}$ (II.5).

It depends on the windings material characteristic. It is usually used $J_{\max}=300 \text{ A/cm}^2$

So from (II.4) and (II.5): $N = \frac{J_{\max} K_w A_w}{I_{eff}}$ (II.6)

And from (II.3) and (II.6): $\frac{L.I_{\max}}{B_{\max} A_e} = \frac{J_{\max} K_w A_w}{I_{eff}}$

For the inductor design we need the geometrical product $A_e * A_w$:

$$\boxed{A_w A_e = \frac{L.I_{\max} I_{eff}}{B_{\max} J_{\max} K_w} (\text{cm}^4)} \quad (\text{II.7})$$

For our project: $L=4.84\text{mH}$, $I_{\max}=3\text{A}$, $I_{eff}=1.87\text{A}$, then:

$$AwAe = 5.168 \text{ cm}^4$$

We chose a product on the manufacturer table (annexe*\$%) with an $AwAe$ higher than the one calculated. So we chose the core E-55/28/21: $AwAe=8.88\text{cm}^4$.

So the number of turns can be determined: $N=164.8 \approx 165$ turns.

The size of the total conductor is given close to: $L_{tot}=l_{winding}.N=19.14m \approx 20m$.

b) Air gap calculus

The inductance value depends on the winding turns number and the total reluctance of the

magnetic circuit: $L = \frac{N^2}{\mathfrak{R}}$ (II.8)

From the Figure 24, this reluctance is the addition of the reluctance of the core and the air gap (if we have one):

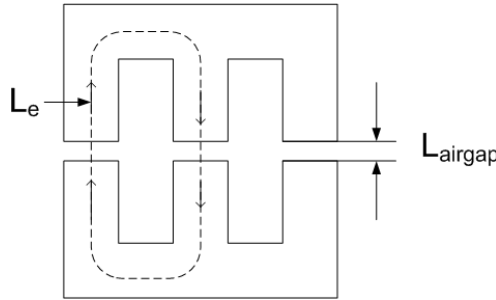


Figure 24: Reluctance situation

$$\mathfrak{R}_{core} = \frac{l_t}{\mu_{core}A_e} \text{ (Core reluctance) and the air gap reluctance } \mathfrak{R}_{airgap} = \frac{l_{airgap}}{\mu_0 A_e}$$

L_{core} is the magnetic path length and μ_{core} is the core permeability.

But considering the value of the permeability [6]: $\mathfrak{R}_{core} \ll \mathfrak{R}_{airgap}$

$$\text{So from (II.8): } L = \frac{N^2}{\mathfrak{R}_{airgap}} \Rightarrow l_{airgap} = \frac{N^2 \mu_0 A_e}{L}$$

In the E-E core usage, the air gap is divided through both lateral legs.

In our project: $l_{airgap} = 2.5mm$.

c) Conductors diameter calculus

Due to high frequency applications, the skin effect must be considered. In high frequency, the current in the conductor tends to distribute itself on the periphery. This effect results on the effective conductor area reduction. The penetration depth current value is given by:

$$\Delta = \frac{7.5}{\sqrt{f_s}} = 0.375mm$$

Therefore, the applied conductor must not have a diameter value larger than 2Δ .

The conductor diameter used to conduct the current depends on the maximum current density.

$$S_{aire} = \frac{I_{eff}}{J_{max}} \text{ (II.9)}$$

Generally, the conductor diameter given by (II.9) is larger than the one obtained by the skin limitation. Then it is usually necessary to associate conductors in parallel in order to conduct the current without overheating the conductors. The number of parallel conductors is given by

$$n_{cond} = \frac{S_{aire}}{S_{skin}} \quad (S_{skin} \text{ conduction area limited by the maximal diameter : } 2\Delta).$$

In our example $D_{max}=2\Delta=0.075\text{cm}$ so according to the manufacturer data we chose the conductor AWG23 because the diameter has to be smaller. So we have the following

parameters:

$$D_{awg23} = 0.056\text{cm}$$

$$S_{awg23} = 0.002582\text{cm}^2$$

d) Temperature Rise calculus

Due to non-idealises, there are losses on the inductor. The total losses are composed by copper losses (joule effect) and magnetic losses (core losses). Those losses result in heat and, therefore, temperature rise, getting higher than the ambient temperature.

i) Copper losses

These losses depend on the winding resistance: $R_{copper} = \frac{\rho l_{winding} N}{n_{cond}}$

So the joule losses are given by:

$$P_{copper} = R_{copper} I_{eff}^2$$

In our project: $R_{copper}=0.569\Omega$

$P_{copper}=1.987\text{W}$.

ii) Magnetic losses

To calculate the core losses we based on manufacturer

value. In fact it is given by: $P_{mag}=P_p \cdot 2m_{core}$

Where m_{core} is a manufacturer data, and P_p can be found on Figure 25 of our core characteristic [7].

As $f_s=40\text{kHz}$ $\Delta B = \frac{L\Delta i}{NA_e} = 0.043\text{T} \Rightarrow \frac{\Delta B}{2} = 0.0215\text{T}$

By graphical result, $P_p = 0.5 \text{ mW /gramme}$ then $P_{mag} = 0.109 \text{ W}$.

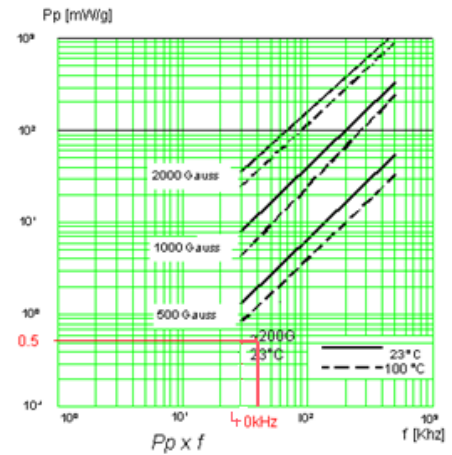


Figure 25: Manufacturer datasheet for magnetic losses

iii) Core thermal resistance

It is defined by:

$$R_{core} = 23 \left(\frac{A_e A_w}{\text{cm}^4} \right)^{-0.3f} \quad (II.10)$$

In our project: $R_{core}=10.3\Omega$

iv) Conclusion

$$\Delta T = (P_{copper} + P_{mag})R_{core} = 21.517^{\circ}C.$$

This value is smaller than $50^{\circ}C$ so it is suitable: the temperature in the inductor is lower than the security temperature that we take.

But to finish it is necessary to check the execution possibility, to verify if it is possible to fit the windings in the core window.

$$A_{w,core} = \frac{N.n_{cond}S_{avg23}}{K_w} = 2.278cm^2$$

$$Exec = \frac{A_w}{A_{w,core}} = 0.911 < 1$$

Exec is between 0 and 1 so it is good and the inductor design is finished. If it were higher than 1, we would have changed the geometrical parameters and then the core. The inductor is designed and will be built by our hands.

5) Technology of the two controllers

a) Technology of the current controller

We want to find the component of the controllers with this bloc structure in Figure 26:

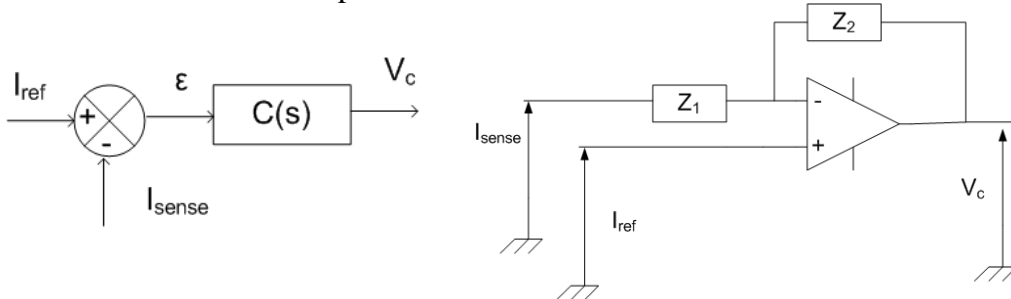


Figure 26: Technology of the current controller

The analysis of the amplificatory input tension gives:

$$\begin{cases} V_- = \frac{V_c Y_2 + I_{sense} Y_1}{Y_1 + Y_2} = \frac{V_c Z_1 + I_{sense} Z_2}{Z_1 + Z_2} \\ V_+ = I_{ref} \end{cases}$$

By considering that the operational amplifier is perfect: $\epsilon = V_+ - V_- \rightarrow 0 \Rightarrow V_+ = V_-$

Then $V_c Z_1 + I_{sense} Z_2 = I_{ref} (Z_2 + Z_1)$

For the impedance Z_2 the appropriate circuit is [6]:

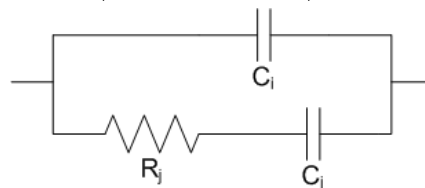


Figure 27: Impedance choice

$$Z_2 = \frac{R_j C_j S + 1}{C_i S \cdot (R_j C_j S + 1) + C_j S}$$

Then

$$Z_2 = \frac{1}{C_i} \frac{S + \frac{1}{R_j C_j}}{S \cdot (S + \frac{C_i + C_j}{R_j C_i C_j})}$$

For the impedance Z_1 the appropriate circuit is a resistance R_i , so $Z_1 = R_i$
 The static gain of Z_2 is $1/C_i$ and the magnitude of C_i is ηF .
 The static gain of Z_1 is R_i and its magnitude is $k\Omega$.

Consequently $|Z_2| \gg |Z_1|$ and $\frac{|Z_2|}{|Z_1|} \gg 1$

$$\begin{aligned} V_C Z_1 + I_{sense} Z_2 &= I_{ref} (Z_2 + Z_1) \\ \Rightarrow V_C Z_1 + I_{sense} Z_2 &= I_{ref} Z_2 \\ \Rightarrow \frac{V_C}{I_{ref} - I_{sense}} &= \frac{Z_2}{Z_1} + 1 \end{aligned}$$

And then
$$\frac{V_C}{I_{ref} - I_{sense}} = \frac{Z_2}{Z_1} = \frac{1}{R_i C_i} \frac{S + \frac{1}{R_j C_j}}{S \cdot (S + \frac{C_i + C_j}{R_j C_i C_j})}$$

We have the good form of controller, which is: $C(s) = G \frac{s + \omega_1}{s(s + \omega_2)}$

So:
$$\begin{cases} k_i = \frac{1}{R_i C_i} \\ \omega_{ci1} = \frac{1}{R_j C_j} \\ \omega_{ci2} = \frac{C_i + C_j}{R_j C_i C_j} \end{cases}$$

We have three equations and four parameters. We can choose the value of one of the component to find the three other.

For the example of the current controller:

$$\begin{aligned} \omega_{ci1} &= 6283 \text{ rad/sec} & C_i &= 1.84 \text{ nF} \\ \omega_{ci2} &= 50270 \text{ rad/sec} & \text{We choose } R_i &= \mathbf{8.2 k\Omega} \text{ and we deduct: } C_j &= 12.87 \text{ nF} \\ k_i &= 6.632 \cdot 10^4 & R_j &= 12.4 \text{ k}\Omega \end{aligned}$$

These three variables will be called R_{cii} C_{cii} C_{cij} and R_{cij} .

b) Technology of the voltage controller

As the transfer function of the voltage controller is basically the same we can use exactly the same technologic structure than the current controller. The values of the components have to be changed only. We just add a little v to the variables to specify that this is Voltage controller components.

$$\text{So: } \begin{cases} k_v = \frac{1}{R_{cvi} C_{cvi}} \\ \omega_{cv1} = \frac{1}{R_{cvj} C_{cvj}} \\ \omega_{cv2} = \frac{C_{cvi} + C_{cvj}}{R_{cvj} C_{cvi} C_{cvj}} \end{cases}$$

In the example of the voltage controller:

$$\omega_{cv1} = 18.85 \text{ rad/s}$$

$$C_{cvi} = 11.4 \text{ nF}$$

$$\omega_{cv2} = 301.6 \text{ rad/s} \quad \text{We choose } R_{cvi} = 33 \text{ k}\Omega \text{ and we deduct: } C_{cvj} = 0.17 \mu\text{F}$$

$$k_v = 2.663 \cdot 10^3$$

$$R_{cvj} = 311 \text{ k}\Omega$$

c) Technology of the current sensor

The current sensor has to give a tension, which is an image of the current i_L . Since the ground is fixed on the output voltage source, with a resistor (R_{shunt}) we can have this image of i_L , V_{shunt} (Figure 28):

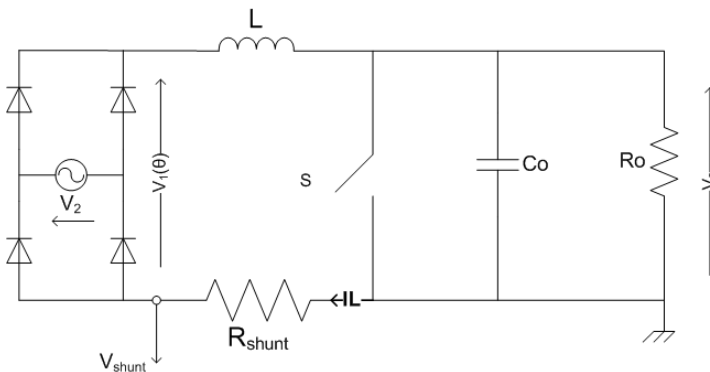


Figure 28: Current sensor strategy

But before this signal can be available for the comparator it is better if it passes through a low-pass filter in order to filter the noise of the R_{shunt} resistor [6]. And if necessary, it has to be multiplied by a gain G_i (cf current control).

The low pass filter is shown in Figure 29:

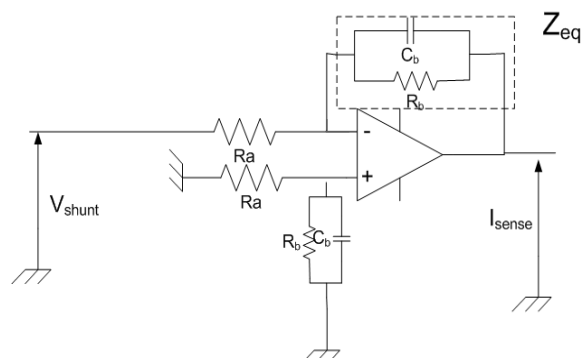


Figure 29 : Circuit rejection

The analysis of the amplificatory input tension gives:

$$\begin{cases} V_+ = 0 \\ V_- = \frac{I_{sense} Y_{eq} + V_{shunt} Y_a}{Y_a + Y_{eq}} = \frac{I_{sense} R_a + V_{shunt} Z_{eq}}{R_a + Z_{eq}} \end{cases}$$

Where $Z_{eq} = \frac{R_b}{1 + R_b C_b s}$

Considering that the operational amplifier is perfect: $\varepsilon = V_+ - V_- \rightarrow 0 \Rightarrow V_+ = V_-$

Then $I_{sense} R_a = -V_{shunt} Z_{eq} \Rightarrow \frac{I_{sense}}{V_{shunt}} = -\frac{Z_{eq}}{R_a} = -\frac{R_b}{R_a} \frac{1}{1 + R_b C_b s}$

Ultimately we have this connection in Figure 30:

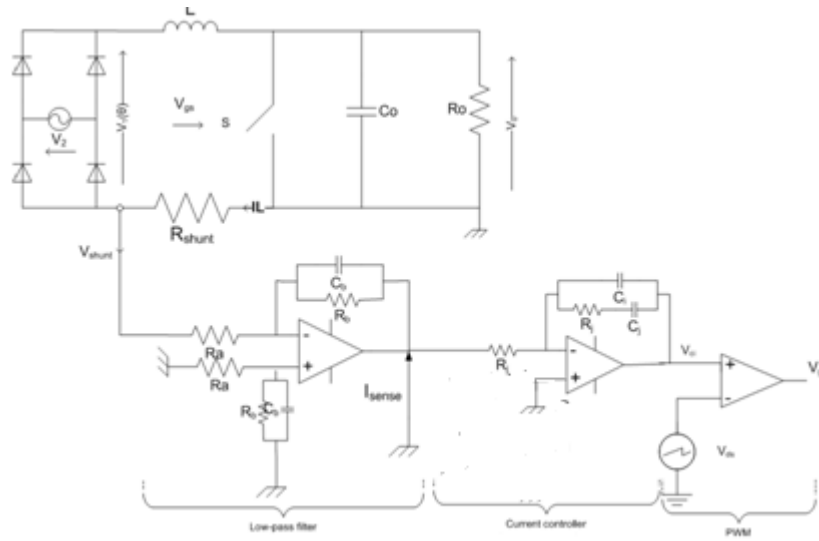


Figure 30: Current loop technology

To determine the value of the components the sensor gain and the frequency of the montage has to be considered. First we choose a value of R_b and then we find the others:

$-C_b = \frac{1}{2\pi \cdot R_b \cdot f_{cut}}$ Where f_{cut} is the low pass filter cut-off frequency, it has to be higher than

120Hz but slower than the switching period (40 kHz).

In the circuit we can take $f_{cut}=10\text{KHz}$ then $C_b=1.6\text{nF}$.

$$\begin{cases} G_I = \frac{I_{sense}}{I_L} \\ \left| \frac{I_{sense}}{V_{shunt}} \right| = \frac{R_b}{R_a} \Rightarrow G_I = \frac{R_b}{R_a} R_{shunt} \Rightarrow R_a = \frac{R_b \cdot R_{shunt}}{G_I} \\ \frac{V_{shunt}}{I_L} = R_{shunt} \end{cases}$$

In the circuit we take $R_b=10\text{k}\Omega$, the R_{shunt} value will be the one we have available. If, $R_{shunt}=20\text{m}\Omega$ and $G_I=1\Omega$, then $R_a=200\Omega$.

We sum up these values:

$$R_{shunt} = 20m\Omega$$

$$R_a = 200\Omega$$

$$R_b = 10k\Omega$$

$$C_b = 1.6n$$

d) Technology of the voltage sensor

To conclude with the controller loop we have to focus on the voltage sensor. We have to measure V_0 . In the model, the voltage gain is about 0.0075.

For this sensor we are going to take the designed structure in Figure 31 :

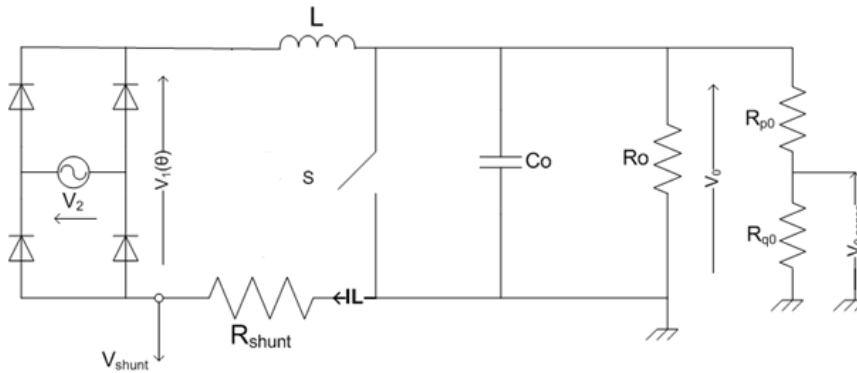


Figure 31: Voltage sensor technology

For V_0 :

$$G_v = \frac{V_{0,sense}}{V_0} = \frac{R_{q0}}{R_{p0} + R_{q0}} \text{ If we chose, for example } R_{q0}=10k\Omega \text{ then } R_{p0}=1.3M\Omega.$$

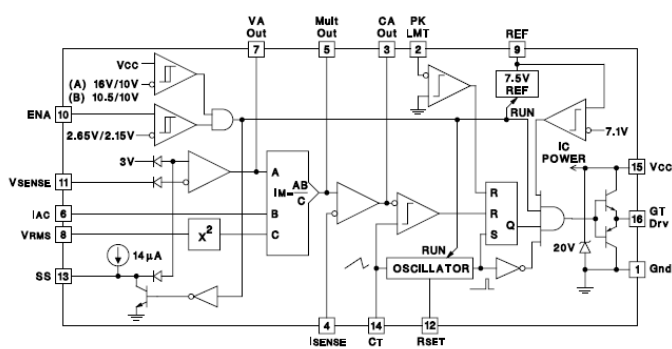
These two previous parts have allowed sizing all a system and a control strategy: a first circuit including the power stage components (inductor, capacitor, resistor, boost diode, rectifier, mosfet) and a second circuit that allows controlling the first one. Controllers are technologically designed.

There are many auxiliary functions that can be done by only one element: sensors, multiplication, some controller components. This element is called pre-regulator or integrated circuit. It allows summing up many functions in only one component.

The next chapter is about sizing this component.

III- Integrated circuit

Previously, the entire Boost rectified circuit and the auxiliary circuitry with its elements were define. Concerning the second one, it allows the control strategy and uses many elements: drivers, dedicated PWM power supply, current and voltage controllers, gates etc. From a practical point of view, the entire auxiliary circuitry can be synthesized by only one element: a High Power Factor Preregulator. Its functions are the followings: current and voltage controller AMPOP, internal multiplier, over current protection, internal driver. Integrated circuit provides the duty cycle. In fact the duty cycle is generated by the comparison between a triangular signal and a continued signal. Thus Integrated circuit is a very complicated component since it uses miniaturized transistors and comparators. We will choose one preregulator from the *INEP* workshop
The preregulator used at the INEP is the UC3854/B in Figure 32:



PIN1	Gnd (Ground)
PIN2	PKLMT (limit current)
PIN3	CA Out (current controller output)
PIN4	I _{sense} (current sensor)
PIN5	Mult out (multiplier output)
PIN6	I _{cc} (over current sensor)
PIN7	VA Out (voltage controller output)
PIN8	V _{REF}
PIN9	V _{REF} (information of over current reference)
PIN10	ENA (power supply)
PIN11	V _{SENSE} (sensor voltage)
PIN12	R _{SET} (oscilloscope calibration resistor)
PIN13	SS (soft start capacitor)
PIN14	C _T (switch frequency setting capacitor)
PIN15	Vcc (integrated circuit power supply)
PIN16	GT Dr (mosfet command voltage)

Figure 32: Block diagram of the pre-regulator UC3854/B

It is necessary to connect this element to our Boost rectified circuit as well as the elements to add to respect the control strategy (resistors and capacitors to synthesize elements as gains or controllers elements etc).

The way to connect each component is listed in the following study thanks to the datasheet. Moreover, we have to include more components if we want our integrated circuit work properly. The strategy control previously shown is taken to describe the argument. Moreover, we consider the manufacture recommendations for connection (add a capacitor to control the response time or resistor to have a voltage which is the image of a current, or vice versa).

1) Voltage control

First, the output voltage is sensed as we can see in Figure 33:

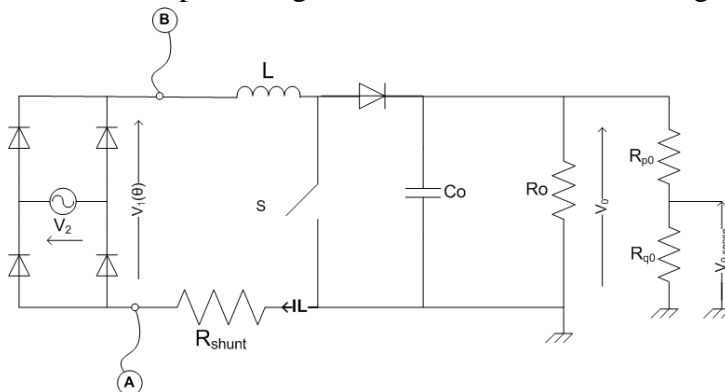


Figure 33: Sensed variables

R_{p0} and R_{q0} allow obtaining V_{Osense} comparable to the value of 4V. Th

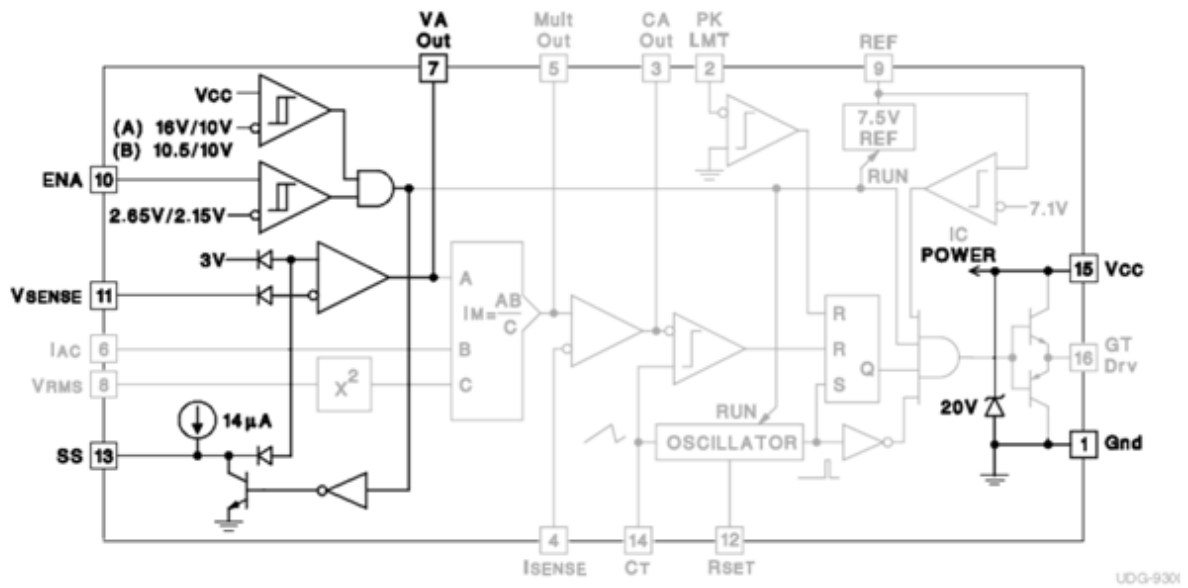


Figure 34: Voltage control elements in the pre-regulator

Our way of thinking is based on Figure 34. Then V_{Osense} has to be sent in order to see if a correction is necessary. That is why we connect our output voltage sensor to the entry of our voltage control (PIN 11). Moreover, the exit of the resistors and capacitors bridge has to be connected to the feedback (PIN 7).

Consequently, the following connection is done in Figure 35:

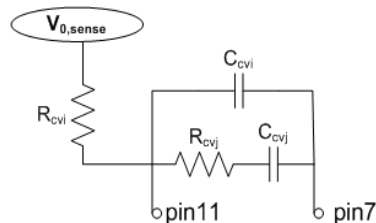


Figure 35: PIN 11 and PIN 7 connections

Then, the voltage controller needs a reference signal to make its work. That is why it is necessary to generate a reference signal by PIN 10. This power allows also the supply of all the preregulator by PIN 15.

With manufacturers, PIN 10 and PIN 15 are connected as shown in Figure 36:

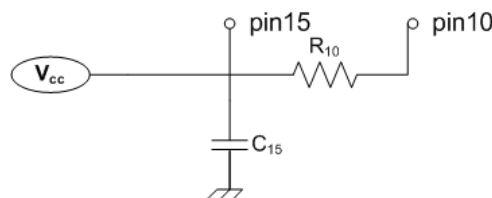


Figure 36: PIN 10 and PIN 15 connection

A small capacitor C_{15} is added to ensure we have V_{cc} as a constant voltage.

The values of the resistor and the capacitor are given by the manufacturer (values in the 7th paragraph).

2) Multiplier

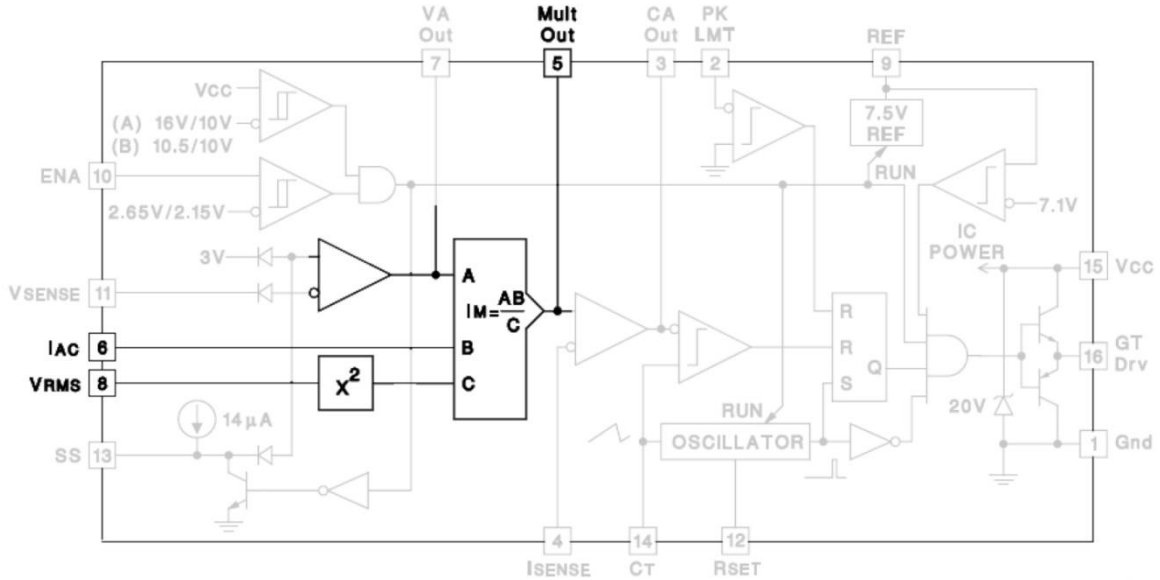


Figure 37: Multiplier operation into the pre-regulator

We sense the current in B and connect it through the Figure 37 to PIN 6 and PIN 8 to what we call the synchronism signal and the effective input voltage feed-forward control.

The input variables are: the voltage controller, the synchronism signal I_{AC} and the effective input voltage feed-forward control V_{ff} . The operation is the following:

$$I_{out}(\theta) = k \cdot \frac{I_{AC}(\theta) \cdot (V_{cv} - 1.5)}{V_{ff}^2} \quad (\text{III.1})$$

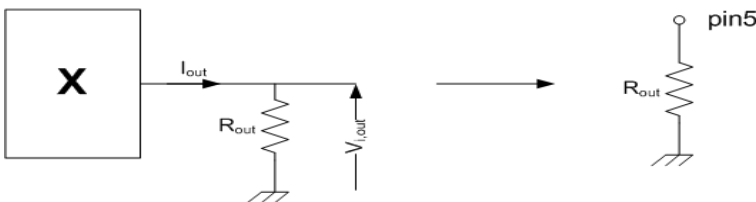


Figure 38: Output of the Multiplier connection

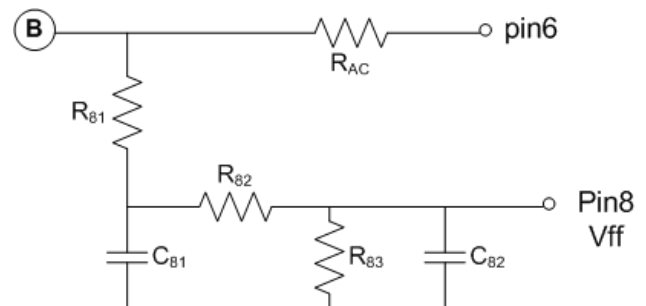


Figure 39: PIN6 and PIN8

We limit the voltage controller at $V_{cv} = 5V$ (datasheet).

Moreover it is given in the datasheet that $V_{ff} = 3V$. So by adopting again k equal to 1, we output peak current from the multiplier is $I_{outpeak} = 7.78 \cdot 10^{-5} A$.

The multiplier output could be a voltage by integrating a resistor (Figure 38). Consequently:

$$R_{out} I_{outpeak} = G_i \cdot I_{input,peak}$$

So: $R_{out} = 33 k\Omega$.

Then this part is connected to PIN 5.

By basic analysis we obtain the following values for the R_{AC} , R_{81} , R_{82} , R_{83} , C_{81} and C_{82} values (Figure39): $R_{81} = 820k\Omega$ (we chose this first value), then:

$$R_{82} = R_{81} \cdot \frac{\frac{V_x}{V_{1med} - V_x} - \frac{V_{ff}}{V_{1med} - V_{ff}}}{1 + \frac{V_{ff}}{V_{1med}}} = 19.38k\Omega$$

$$R_{83} = (R_{81} + R_{82}) \left(\frac{V_{ff}}{V_{1med} - V_{ff}} \right) = 12.91k\Omega$$

$$C_{81} = \frac{1}{2\pi f_{cv} R_{82}} = 0.6845\mu F$$

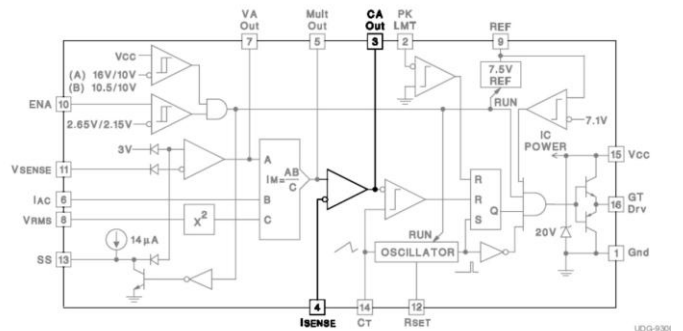
$$C_{82} = \frac{1}{2\pi f_{cv} R_{83}} = 1\mu F$$

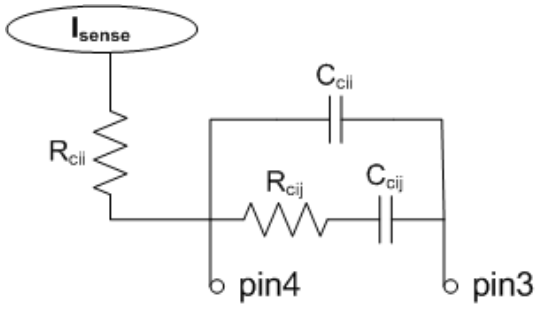
$$R_{AC} = \frac{V_{1med}}{I_{AC}} = 1.55M\Omega$$

Where $V_{1med} = \frac{2}{\pi} V_{1peak}$, $V_x = 3V$ (PIN9 voltage given in the datasheet) and $I_{ACmax} = 200\mu F$ according again to the datasheet of the pre-regulator.

3) Current control

Figure 40: Current elements in the pre-regulator





As previously for the voltage control, the sensed current (or its voltage image) is injected in the current controller (Figure 40).

That is why we connect our resistors and capacitors bridge in PIN4 (enter of the voltage AMPOP) and PIN3 at the exit for this one (Figure 41).

Ultimately, we obtain the current control V_{ci} .

Figure 41 : PIN 3 and PIN4 connections

4) PWM modulator configuration

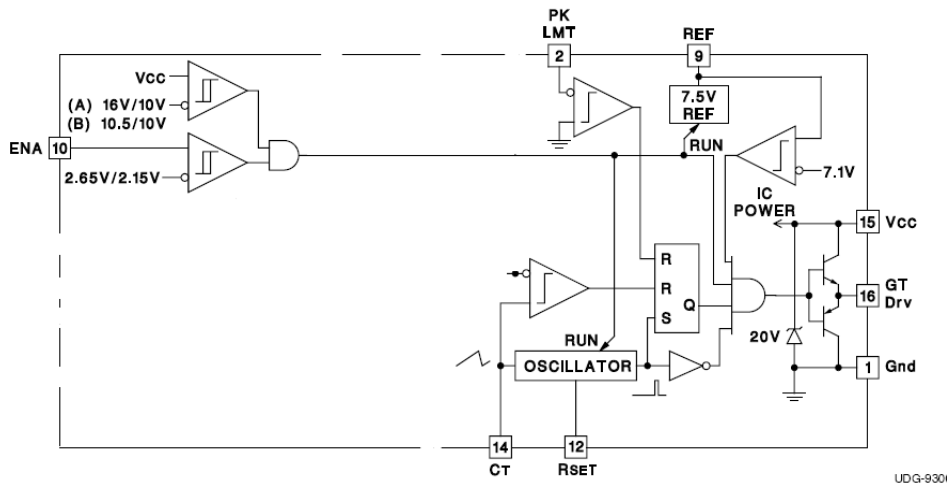


Figure 42: PWM configuration

By the date sheet, we have to integrate a capacitor and a resistor respectively in PIN 14 and PIN 12. Moreover, we have the relation:

$$f_s = \frac{1.25}{C_T \cdot R_T} \quad (\text{III.2})$$

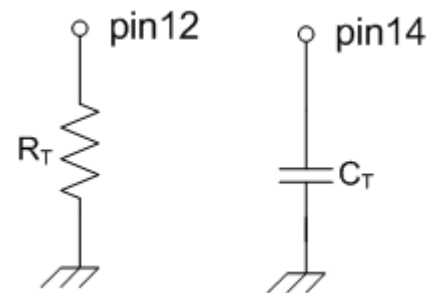


Figure 43: PIN 14 and 12 connection

The manufacturer recommends that $C_T = 1nF$ (7th paragraph).

That is why we can deduce the value of the R_T resistance via **III.2**: $R_T = 31.25k\Omega$.

The connections of these components are available in Figure 43.

5) Over current protection

An over protection is done by this system. If PIN 2 is equal to zero, mosfet pulses will be inactivated. The given over current is configured as shown in Figure 44:

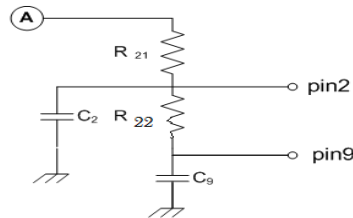


Figure 44: PIN2 and PIN9 connection

This circuit is associated with the resistor R_{shunt} to avoid an over current, characterized by a maximum voltage into this previous resistor [6].

With: $V_{shunt\ max} = R_{shunt} \cdot I_{protection}$

Where $I_{protection}$ is a specification data: $I_{protection} = 1.5 I_{L\ Peak} = 150\% \text{ of } I_{L\ Peak}$

When $V_{shunt\ max}$ is obtained in R_{shunt} , then PIN 2 is equal to zero.

Moreover:

$$\frac{V_{PIN\ 9} - V_{PIN\ 2}}{R_{22}} = \frac{V_{PIN\ 2} - (-V_{shunt\ max})}{R_{21}}$$

Since $V_{PIN\ 2}=0$ then we obtain one resistance in function of the other:

$$R_{21} = \frac{V_{shunt\ max} \cdot R_{22}}{V_{PIN\ 9}}$$

By fixing arbitrarily one we have the other one.

$V_{PIN\ 9} = V_X$ has an internal voltage of about 7.5V (specification data).

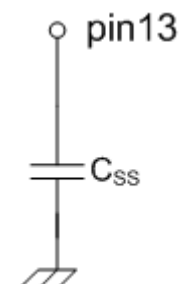
If $R_{22} = 10k\Omega$, then it is obtained: $R_{21} \approx 100\Omega$.

Moreover, for the same reason than previously in PIN 15, two small capacitors C_2 and C_9 are added to make sure we obtain constant voltage in PIN 9 and PIN 2 (values recommended by the manufacturer in the 7th paragraph).

6) Soft-start capacitor

Then, a capacitor in PIN 13 has to be considered in order to define a time which allows the reference signal in PIN 9 to reach its nominal value (Figure 45).

Figure 45: PIN 13 connections



Its value is recommended by the manufacturer (7th paragraph).

7) Manufacturer Recommendations

The manufacturer recommends taking the following values for these different elements:

- $C_2 = 470 \text{ pF}$
- $C_9 = 100 \text{ nF}$
- $C_{15} = 100 \text{ nF}$
- $C_{SS} = 1 \text{ pF}$
- $R_{10} = 22 \text{ k}\Omega$

All the circuit is designed with different and several aspects in order to be in adequacy with the scope statement given by the INEP.

8) Simulation on PSIM

Once all the system is designed, a simulation allows to conclude that the circuit works (see figure 46). Moreover, the resulting waveforms (Figure 47) show that the boost operation and all the control strategy work as expected.

Figure 46: PSIM Simulation

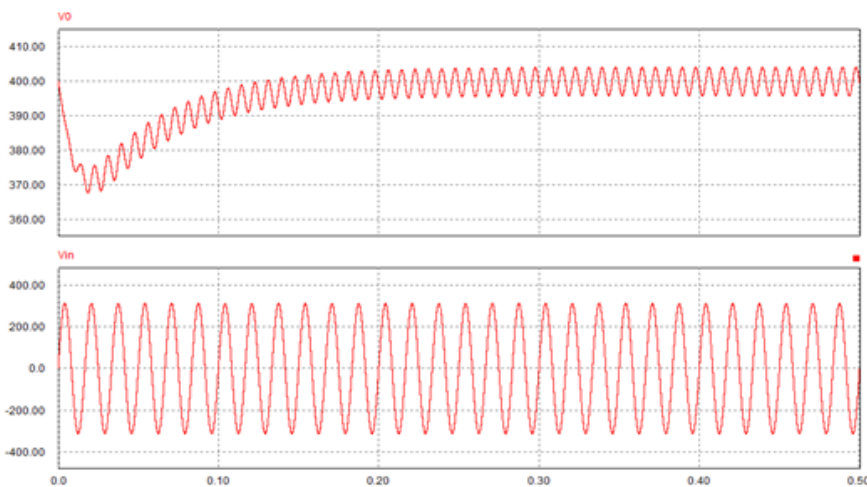
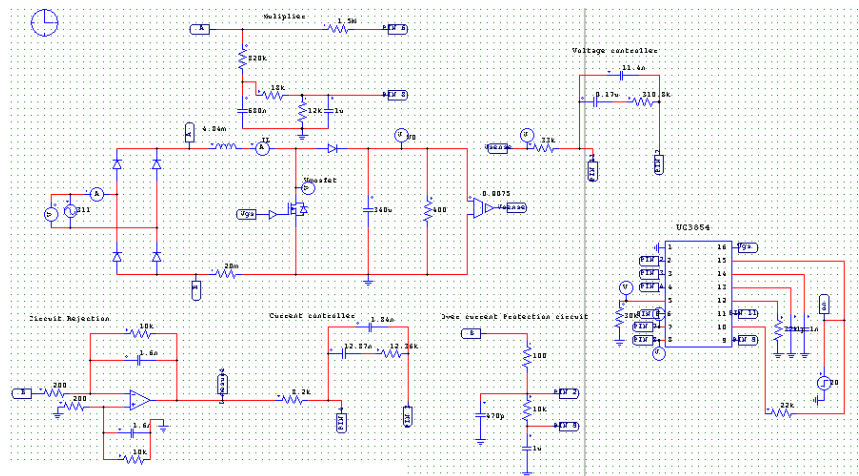


Figure 47: Simulation results

We can conclude that our circuit medullisation is working by the simple fact that the average value of the output voltage increases until 400V very quickly and then stabilizes. Moreover, the other variables were checked to see if the waveforms were as expected: the inductor charges as expected, the mosfet switches...

This validation by simulation enables us to say that all our sizing is correct and that the scope statement is respected.

This step done, we can do the other aspect of the work which is the practical part.

IV- Practical work

All the construction of our circuit is made at the INEP by technicians who are accustomed to make this kind of systems. We have built our own inductor.

Once the prototype ready, this part is describing the tests we made, showing our results and our conclusion on the practical aspect.

1. Circuit construction

Once the circuit designed, it was important that our expectations were well understood by the technicians who made the system. That is why it was necessary to talk with them, using the same language in order to make sure that our needs were well translated (technically speaking). Consequently, we had to make a short scope statement in which we explained the expected system, with detailed figures, table of components (with their quantity, specifications and place in the circuit) and expected results (current through each wire). Currents between each point had been specified: technicians need those values in order to size electrical contacts strips between each component.

Moreover, some variable resistors had been added in our design: some resistor values were not available in the commerce stock. We just need to calibrate the circuit after its construction.

This scope statement is in appendix (Appendix B). A circuit breaker and a fuse had been added in order to prevent any over voltage or over current problem.

2. Inductor construction

Whereas other components are commercial ones, the inductor is made in the INEP.

In the part II.4), the inductor design had been done: materials to use, wire length, number of turns with a selected core etc.

Once the E-55/28/21 core and the conductor wire AW-23 available, from a practical point of view we needed to determin the number of wire, wire length and the number of parallel conductors:

- ✓ $N = 165$ turns
- ✓ $l = 19.1$ m
- ✓ $n_{\text{cond}} = 3$



Figure 48: Inductor construction

Once the component built as shown in Figure 48, it was possible to measure the inductance value. The obtained inductor in Figure 49 is much closer to the expected result: as a matter of fact, the obtained inductance value is about 5.04mH (expected value of 4.85mH).

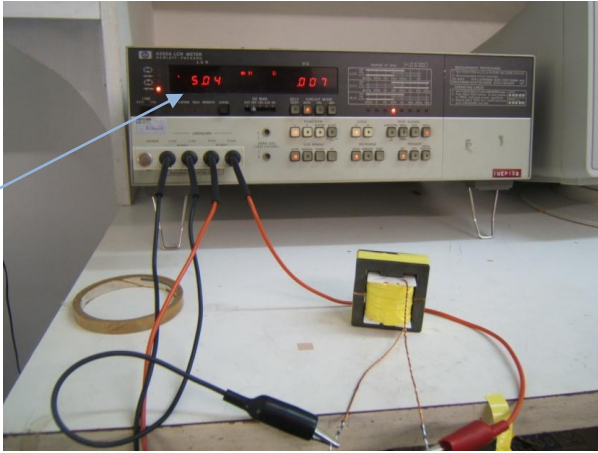
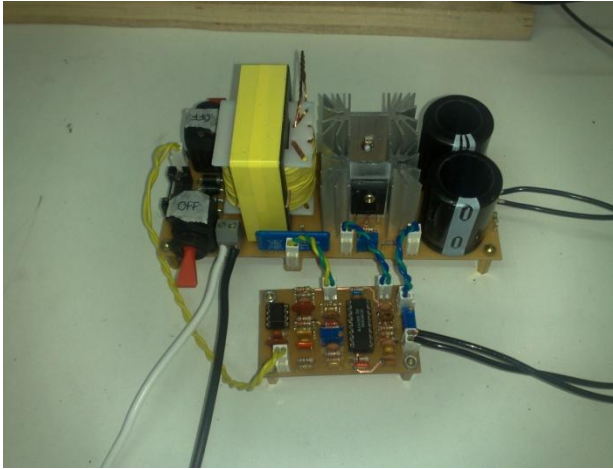


Figure 49 :The experimental obtained inductance

3. Obtained prototype

Once the technician work finished, they gave us two circuits: the first with the boost components and the auxiliary one including the UC3854B pre-regulator, controllers, sensors, over current circuit etc. In addition, we had to build a resistive load of 400W with two switches that allowed us to vary the value of the load. Figure 50 shows the obtained prototype.



} Power stage circuit (Boost circuit)
 } Auxiliary circuit (Control circuit)

Figure 50: The prototype

4. Tests and results

We recorded our tests with an oscilloscope. All the waveforms are recorded thanks to this tool. Moreover, we used an accurate tool (Yokogama digital power meter WT230®) to obtain the power factor values and exact values of the following variables: input and output current, input and output voltage, input and output power. The Figure 51 sums up these measures.

	33% of the nominal load	66% of the nominal load	100% of the nominal load
input current with control	0,681A	1,2821A	1,914A
input voltage with control	215,28V	219,3V	218,64V
input power with control	142,82W	278,27W	415,37W
output current with control	0,347A	0,6845A	1,01A
output voltage with control	402,06V	399,98V	399,98V
output power with control	139,45W	272,1V	404,74W
power factor without control	0,69	0,69	0,72
power factor with control	0,9773	0,9897	0,993
yield with control	0,9764	0,9778	0,9767
THD of the input signal without control	108%	x	x
THD of the input signal with control	10,75%	x	x

Figure 51: Measures

These results are exploited thanks to the software Wavestar.
Figures from 52 to 55 come from Wavestar.

a) Calibration

First of all, we had to calibrate our two sensors: thanks to the output values of these components, we managed this first test which is very important for the following tests. By the same token, we calibrated the third variable resistor.

b) Input signal

In these tests, the goal is to check the waveforms and the values of the input variables. Concerning the input source, we have a sinusoidal signal delivered into our system. The figure 52 shows these input variables for our nominal load of 400W.

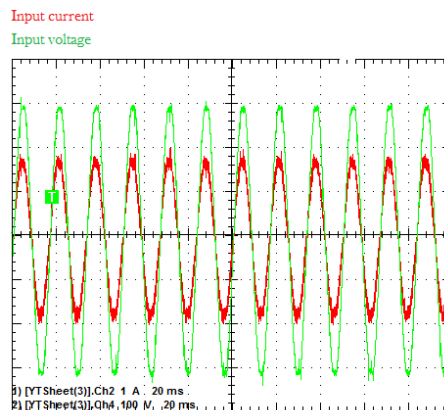


Figure 52: Input signal

We observe that our input signal is sinusoidal as expected and that the input current is in phase with our input voltage.

c) Harmonic aspect

Thanks to the table on the Figure 51, we have values of the Total Harmonic Distortion: it is a distortion factor that shows the variation of a given signal compared to a reference signal (the reference signal here for the input signal is the perfect sinusoidal input signal). Here, with a non-controlled system and 33% of the nominal load the THD is about 108%. For the controlled system, this THD is about 10.75%. So it was divided by ten thanks to the control

circuit. We can conclude that our control system makes our input signal very closed to the perfect sinusoidal signal.

Since our controllers are designed to work for a specific cut-off frequency, they attenuate the weight of the harmonics of rank higher than 1 (which is the fundamental). This aspect is illustrated in figure 53.

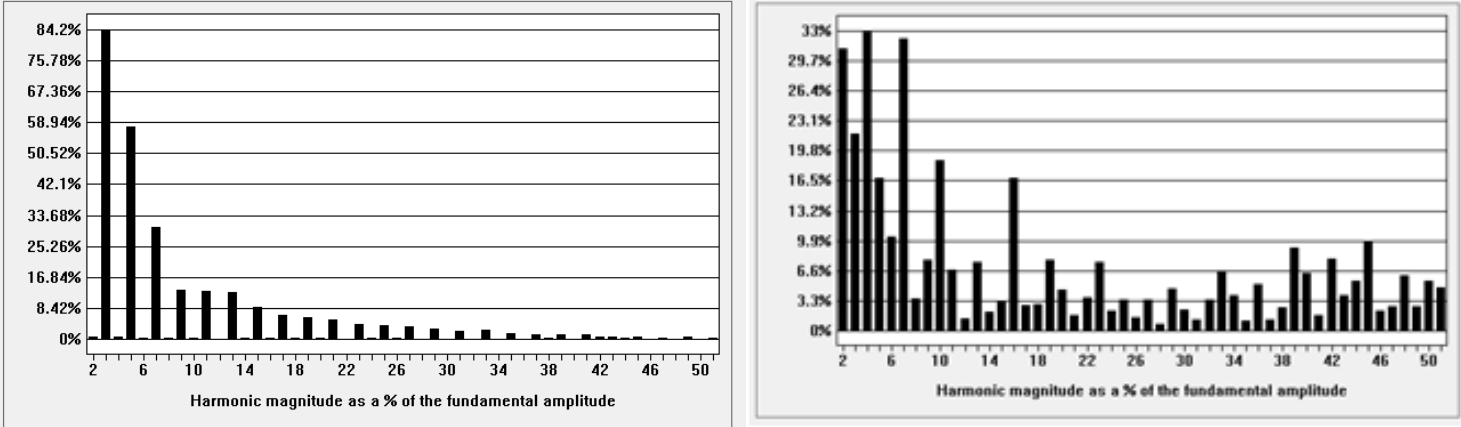


Figure 53: harmonic spectrum of the input current: first without control and then with active filter

This previous figure allows us to conclude that our control circuit (active filter) is acting by letting the weight of the fundamental one and attenuating the weight of the other harmonic magnitude. This aspect allows having an input signal very closed from the one we wanted, which is a sinusoidal one. This aspect is confirmed by the distortion factor which is divided by ten: this factor defines how a signal is closed from the reference. Consequently, we can conclude to a good input voltage.

d) Output voltage

The most important variable to observe is the output voltage. Its control and its final value is the aim of our converter. The following Figure 54 shows the output voltage without control loop and with control loop, in the case of 33% of the load.

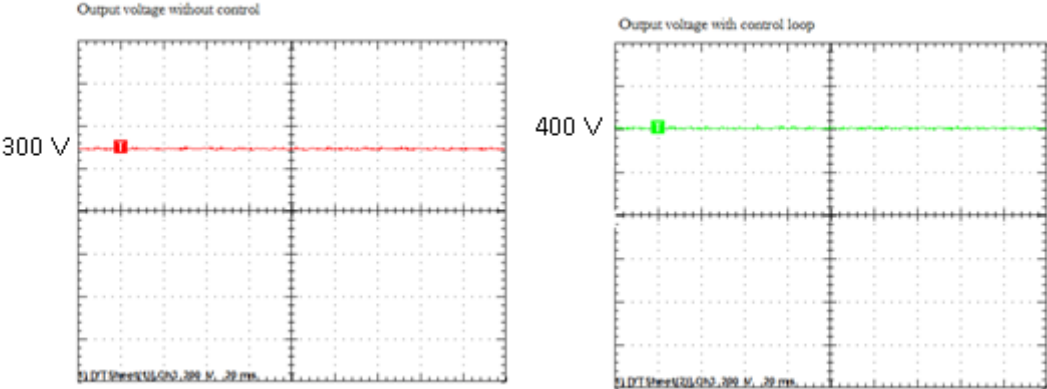


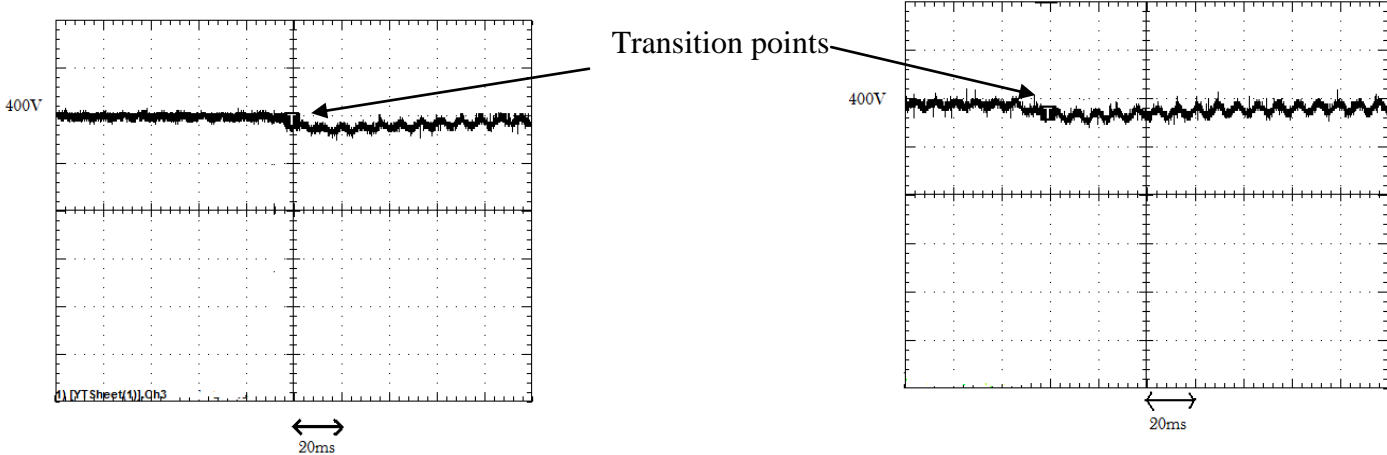
Figure 54: Output voltage without control and with the designed control loop

We can notice that the control loop increases the output voltage from 300V (without control) until 400V, which is the wished value.

Consequently, the auxiliary circuit (including the control components) allows us to close the loop and the control engineering can be done: in the first case (without control), the system reaches a value and that is it. In the second case, the feedback makes it possible to see that the output value is not equal to the expected value of 400V: thus, the controllers command the mosfet on commutations in order to rectify this value. Closing the loop implies a better answer of our system: it reaches the expected value.

e) Dynamic behaviour of the system and yield

Figure 55 shows the experimental transitions between different values of the load: as a matter of fact, the load varies from 33% to 100% of the nominal load. The goal is to see the behaviour of our circuit in a dynamic case about load.



(a) Transition from 33% to 66% of the nominal load

(b) Transition from 66% to the nominal load

Figure 55: dynamic behaviour of the system

These tests shows a little drop of the output voltage at the two transitions (from 33% to 66% and 66% to the full value of the nominal load). Just before these two transitions as (see in Figure 55), the output voltage is equal to 400V. Then, this voltage drops about ten or fifteen volts and increases to the value of 400V: this result allows us to conclud that our system is well controlled by the engineering strategy. The closed loop and the control circuit do the expected operation on the output voltage: it maintains this value of 400V and that even if perturbations happen, with an answer time of about 100 milliseconds.

Our prototype works on dynamic loads: moreover the answer time of our system is about 100 milliseconds, which is a higher performance time.

The figure 56 shows how the power factor evolves according to the load in two cases: the first one with the control and the second one without the control.

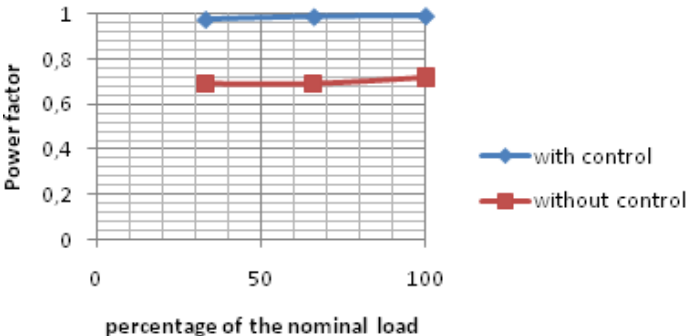


Figure 56: power factor in function of the connected load (percentage of the nominal load)

We observe that connecting our control system enable to increase the value of the power factor from 0.7 to 0.99. We can conclude that, as expected, our control system increases the power factor and thus, makes it possible to reduce the Joule effect as well as the costs.

We define the efficiency as the ratio of the output power by the input power:

$$\eta = \frac{P_0}{P_2}$$

Figure 57 shows the efficiency of our controlled system according to the connected load (percentage of the nominal load).

There are three values of efficiency for three loads (33%, 66% and 100% of the nominal load). We draw a tendency curve from these three points. We can observe that the efficiency increases from 33% to 66%. It continues to increase slightly after 66%, until 70% for a value of 0.9778 for the power factor. Then it drops from 70% to 100% of the nominal load to reach 0.9768.

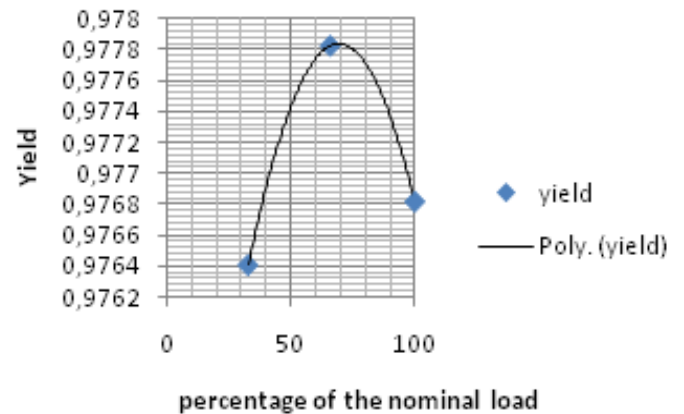


Figure 57: efficiency in function of the load.

First of all, we can conclude that the efficiency is much closer to 1: in all cases it is between 0,976 and 0,978. A good efficiency therefore mea a more economic system.

Then, thanks to previous observations, we can conclude that the optimal efficiency is obtained for a percentage of the nominal load (here 70%). There are no physical for this aspect: the maximum efficiency could happen at any values of the load without predictions.

Conclusion

The Boost converter has been studied in order to show that this kind of system is very useful for many applications: in our case, for electrical warships. The power stage components were also sized to respect a scope statement: the inductor, the output capacitor and the resistor. Moreover, two controllers have been designed in order to follow a control strategy: maintain the output voltage at a constant value of 400V from an input source 220V/60Hz. These two controllers are the following: a current controller and a voltage controller. These components implied many other functions and components: current and voltage sensors, rejection circuit in order to filter noises and disturbances and have a pure measured current, over current protection, PWM configuration. The controllers and their auxiliary functions have been included in only one component: the integrated circuit UC3854/B. This component synthesizes all these functions. All this sizing was done alongside with PSIM, simulation software. This powerful tool allowed us to see and check our results. Once the final simulation was working, we wrote a scope statement to the technicians: they built our system and made the practical part possible.

Our tests began by testing step by step the two circuits in order to make sure that these two previous circuits were as we expected (and that the circuit would not burn when the electrical power came). We then tested the entire system. The first variable we checked was the input signal: we had a sinusoidal signal and the voltage fitted with the current. Moreover, harmonically speaking, the 220V/60Hz source was fine.

Then we checked the most important variable: the output voltage. This one is significantly increased with the control circuit: without it we have an output voltage of 300V. Controlled system made it possible to reach the constant value of 400V.

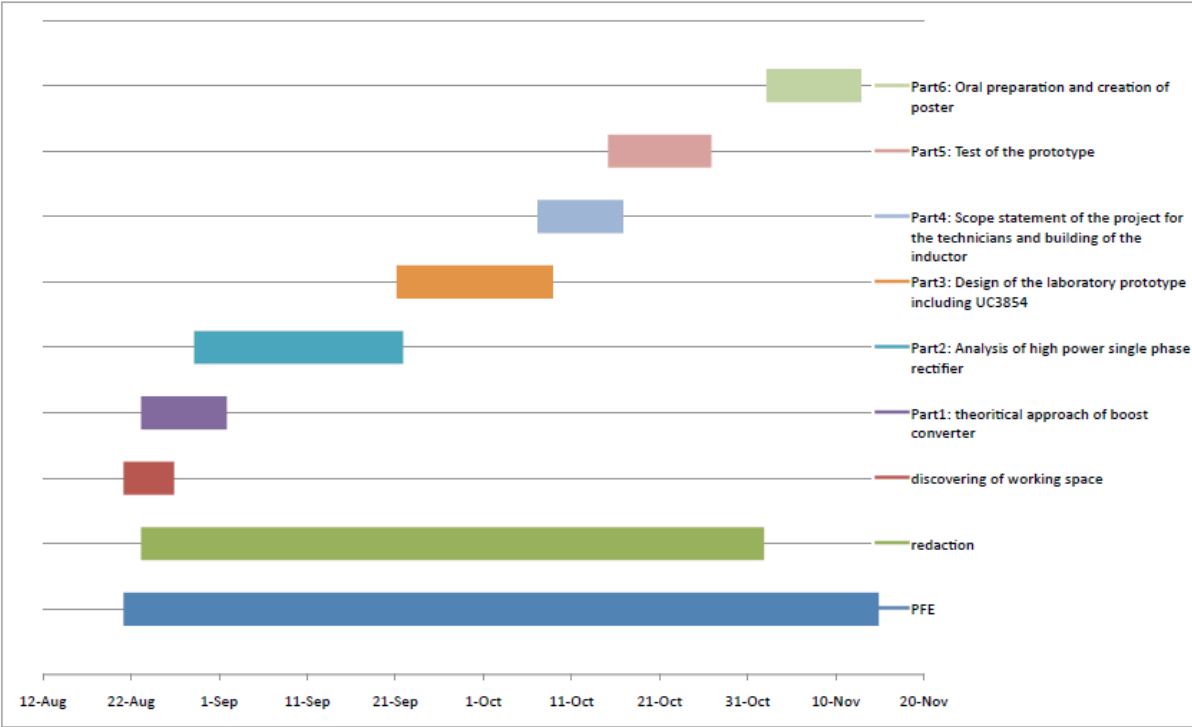
Tests continued with the dynamic behaviour of the system: we varied the value of our load. Results show that the power factor increases a lot with the control circuit: without it, it is about 0.7. Once the control acting, the power factor is about 0.99. The system is more economic when we consider power: moreover, the current is lower and consequently losses are lower. This aspect is confirmed by the fact that the efficiency is very good.

A high power factor allows having the same active power but with lower currents and consequently a better exploitation of the system.

During this project we had to deal with difficulties that engineers have to cope with every project. We had a scope statement and we had to design a system, with the control circuit and the boost circuit. We had to find solutions and select components to design the most effective system. The theoretical part was done alongside with two powerful software: MathCAD and PSIM. Algebra computation and simulations allowed us to size the system and correct some aspects. Then we tested our system with a meticulous protocol. This way of working will be beneficial for us to face difficulties aboard our warships and is applicable to all power electronic systems, which are present in all warships today.

Appendixes

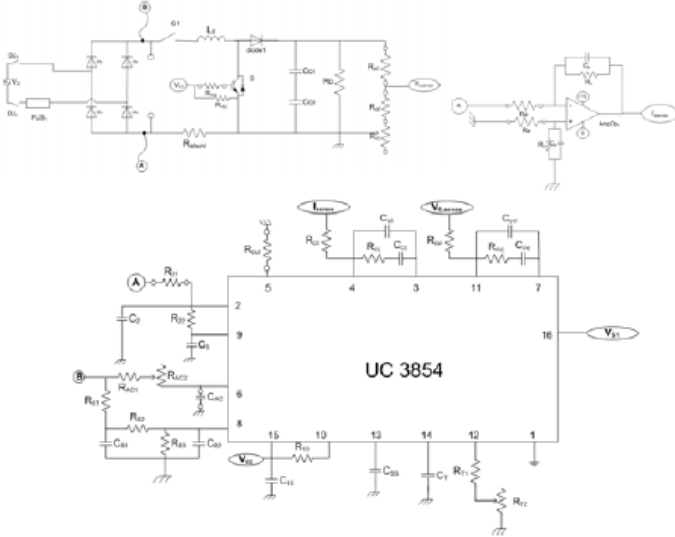
Appendix A: Gant diagram



Appendix B: Scope statement

Scope statement

Schema



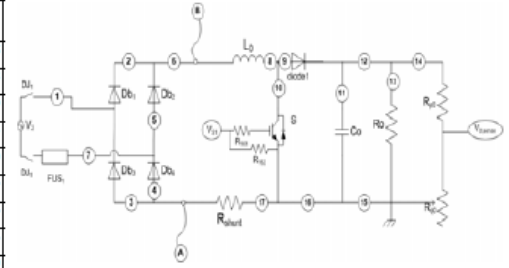
Component	Specification	Value	Quantity
Circuit Breaker - DJ ₁			1
Fuse - FUS ₁		4A	1
Bridge Diodes - Db ₁ , Db ₂ , Db ₃ , Db ₄	1N5404		4
Switch - G ₁	Manual		1
Inductor - L ₂	EE-55/28/21	4.84mH	1
Mosfet - S	IRFP31N50L - International rectifier		1
Diode - Diode1	FFP08H60S hyperfast 2 rectifier		1
Capacitor - C ₀₁ , C ₀₂	Electrolytique	680µF	2
Resistor - R ₀	1/4W, 5% of precision	400Ω	1
Resistor - R ₂₀	1/2W, 5% of precision	470kΩ	1
Resistor - R ₂₁	1/4W, 5% of precision	2.8kΩ	1
Potentiometer - R ₂₂	1/4W	1kΩ	1
Resistor - R ₁₆₁	1/4W, 5% of precision	22Ω	1
Resistor - R ₁₆₂	1/4W, 5% of precision	10kΩ	1
Resistor - R ₁₆₃		20mΩ	1

UC 3854 circuit

Components	Specification	Value	Quantity
Resistor - R ₂₁	1/4W, 5% of precision	100Ω	1
Resistor - R ₂₂	1/4W, 5% of precision	10kΩ	1
Resistor - R ₂₃	1/4W, 5% of precision	820kΩ	1
Resistor - R ₂₄	1/4W, 5% of precision	18kΩ	1
Resistor - R ₂₅	1/4W, 5% of precision	12kΩ	1
Resistor - R ₂₆	1/4W, 5% of precision	1.5MΩ	1
Potentiometer - R ₂₇	1/4W	100kΩ	1
Resistor - R ₁₀	1/4W, 5% of precision	22kΩ	1
Resistor - R ₁₁	1/4W, 5% of precision	33kΩ	1
Resistor - R ₁₂	1/4W, 5% of precision	8.2kΩ	1
Resistor - R ₁₃	1/4W, 5% of precision	12kΩ	1
Resistor - R ₁₄	1/4W, 5% of precision	33kΩ	1
Resistor - R ₁₅	1/4W, 5% of precision	310kΩ	1
Resistor - R ₁₆	1/4W, 5% of precision	27kΩ	1
Potentiometer - R ₁₇	1/4W	10kΩ	1
Capacitor - C ₂	Ceramic	470pF	1
Capacitor - C ₃	Ceramic	100nF	1
Capacitor - C ₄	Ceramic	100nF	1
Capacitor - C ₅	Ceramic	1µF	1
Capacitor - C ₆	Ceramic	100nF	1
Capacitor - C ₇	Ceramic	1pF	1
Capacitor - C ₈	Ceramic	1nF	1

Current values

Core	I _{max} (A)	I _{eff} (A)
1	2.90	1.85
2	2.90	1.28
3	2.89	1.29
4	2.89	1.33
5	2.89	1.30
6	2.90	1.84
7	2.90	1.84
8	2.90	1.84
9	2.89	1.50
10	2.90	1.06
11	1.90	1.12
12	1.01	1.00
13	1.01	1.00
14	0	0
15	1.02	0.99
16	2.9	1.52
17	2.95	1.84



Component	Specification	Value	Quantity
Capacitor - C ₀₁	Ceramic	1.8nF	1
Capacitor - C ₀₂	Ceramic	13nF	1
Capacitor - C ₀₃	Ceramic	11nF	1
Capacitor - C ₀₄	Ceramic	160nF	1

Current low-pass filter circuit

Components	Specification	Value	Quantity
Operational amplifier - AmpOp	LM 358P		1
Resistor - R ₄	1/4W, 5% of precision	200Ω	2
Resistor - R ₅	1/4W, 5% of precision	10kΩ	2
Capacitor - C ₅	Ceramic	1.6nF	2

Bibliography

Classical references

- [1] International Thomson Publishing, Robert W. Erickson, 1997. - Fundamentals of power Electronics
- [2] M. Charpentier lesson, Ecole Navale. – Electronique de Puissance.
- [3] Edition Masson, J.-P. Ferrieux, 1987.-Alimentations à découpage convertisseurs à résonance.
- [4] Edição do autor, Ivo Barbi, 2000. -Conversores CC-CC Basicos Nao Isolados.
- [5] UFSC, INEP, Ivo Barbi, 2004.-Analise da topologia Tradicional do Retificador Boost.
- [6] UFSC, INEP, Silvia Helena Pini, 2010.-Dissertação de Mestrado, Estudo e Implementação de um Filtro Ativo .

Web Sites

- [7] <http://www.thornton.com.br/>: Manufacturer core web site
- [8] http://www.iutenligne.net/ressources/electricite_electrotechnique/Marty/Elec/fichiers/6_Puissances/Elec6.htm: Main power electronic principles
- [9] http://www.smeps.com/Knowledge/Boost_PFC/Boost_PFC.shtml: Boost PFC principles