Modelling, design and control of a DC-DC converter for automotive Thermoelectric-Generator

Tutor
Prof. Marcello Chiaberge

Candidate
Maria Vittoria Morelli

December 2018
Modeling, Design and Control of a DC-DC converter for Automotive Thermo-electric Generator
ABSTRACT

Nowadays the reuse of wasted energy is an important task for the majority of the fields, particularly for the automotive one. This aims at more efficient systems and reduce pollution, making a product compatible with the requests of active regulations, environment, consumers and producers. With regards to this, in the automotive field, the Exhaust Gas Recirculation (EGR) is becoming an important promise in the energy improvement. The present thesis deals with the modelling, design, control and implementation of a DC-DC power converter to recover energy from the exhaust system in a vehicle.

The strategy is to convert the heat wasted energy into electricity by means of a thermoelectric generator (TEG), which is based on Seebeck effect. The TEG, designed by Magneti Marelli, is connected to a DC-DC converter which allows the recharge of the battery. Many DC-DC converter topologies are examined, few of them are acceptable for the considered application: recharge a battery (nominal voltage 12 V) with an input voltage from 5 V to 55 V and a nominal maximum power of 300 W. In case of extremely high temperature from the exhaust, a flap electro-mechanical valve permits the hot gasses to bypass the TEG in order to avoid damages.

Preliminary losses calculations, complexity, number of components and cost lead to focus on the non-inverting Buck-Boost DC-DC converter topology, in the interest of having both step-up and step-down operation modes.

The thermoelectric generator is modeled and simulated in a model-based environment with the defined DC-DC converter. Maximum Power Point Tracking (MPPT) Perturbation and Observation (P&O) algorithm, for the TEG, is designed and implemented. Dual loop control, for the DC-DC converter, is chosen to control output voltage and current. Design, power dissipation estimations and simulations are validated through prototypes.
# Index

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABSTRACT</td>
<td>2</td>
</tr>
<tr>
<td>Introduction</td>
<td>5</td>
</tr>
<tr>
<td>ENERGY SAVING TECHNOLOGY IN AUTOMOTIVE FIELD</td>
<td>5</td>
</tr>
<tr>
<td>AIM AND THESIS LAYOUT</td>
<td>6</td>
</tr>
<tr>
<td>Thermoelectric generator</td>
<td>8</td>
</tr>
<tr>
<td>SEEBECK EFFECT AND PARAMETERS</td>
<td>8</td>
</tr>
<tr>
<td>MODELLING AND SIMULATIONS</td>
<td>10</td>
</tr>
<tr>
<td>DC-DC Converter</td>
<td>16</td>
</tr>
<tr>
<td>DC-DC TOPOLOGIES AND CONSTRAINTS</td>
<td>16</td>
</tr>
<tr>
<td>NON INVERTING BUCK-BOOST CONVERTER (NIBB)</td>
<td>18</td>
</tr>
<tr>
<td>DESIGN</td>
<td>22</td>
</tr>
<tr>
<td>Switching Frequency Selection</td>
<td>22</td>
</tr>
<tr>
<td>Inductor Selection</td>
<td>23</td>
</tr>
<tr>
<td>Duty Cycle Computations</td>
<td>24</td>
</tr>
<tr>
<td>Capacitors Selection</td>
<td>25</td>
</tr>
<tr>
<td>Power dissipations and Efficiency</td>
<td>27</td>
</tr>
<tr>
<td>Control algorithms and simulations</td>
<td>34</td>
</tr>
<tr>
<td>MPPT, ALGORITHMS AND IMPLEMENTATIONS</td>
<td>34</td>
</tr>
<tr>
<td>Maximum Power Transfer Theorem</td>
<td>34</td>
</tr>
<tr>
<td>Perturb and Observe (P&amp;O) algorithm</td>
<td>35</td>
</tr>
<tr>
<td>Incremental Conductance (IC) algorithm</td>
<td>36</td>
</tr>
<tr>
<td>Modified algorithms and implementation</td>
<td>37</td>
</tr>
<tr>
<td>CONTROL DESIGN</td>
<td>43</td>
</tr>
<tr>
<td>CONTROL STRATEGIES AND SIMULINK ARCHITECTURES</td>
<td>44</td>
</tr>
<tr>
<td>First Control Strategy: Current Loop</td>
<td>45</td>
</tr>
<tr>
<td>Second Control Strategy: Dual Loop</td>
<td>47</td>
</tr>
<tr>
<td>Third Control Strategy: MPPT + Current Loop</td>
<td>48</td>
</tr>
<tr>
<td>Future developments, implementation and conclusions</td>
<td>52</td>
</tr>
<tr>
<td>FUTURE DEVELOPMENTS</td>
<td>52</td>
</tr>
<tr>
<td>IMPLEMENTATION</td>
<td>53</td>
</tr>
<tr>
<td>CONCLUSION</td>
<td>57</td>
</tr>
<tr>
<td>References</td>
<td>59</td>
</tr>
</tbody>
</table>
ENERGY SAVING TECHNOLOGY IN AUTOMOTIVE FIELD

“Automotive energy recovery systems, such as RBS, turbocharger, and EGR, hold a significant potential to recover the energy wasted while braking or by exhaust gasses during the combustion process. At present, electric vehicles are gaining importance as a substitute to the conventional vehicle powered by IC engine that runs on non-renewable fossil fuels such as petrol and diesel. EGR is the most promising technology which reduces NOx emission by reusing the exhaust gasses reviled during the combustion process.”

As Eshan Hira of Allied Market Research declares, the increase of electrical and hybrid vehicles represents an important reduction of toxic emissions without renouncing the performance of the whole system. The automotive company Tesla Motors, Inc. in the 2003 was the pioneer of the production of electric vehicles, cars of an improbable future. In the 2018 the company is just one of the several manufacturers: according to “usnews.com”, among the best electric vehicles of the year in U.S. there are cars produced by Tesla and also BMW, Chevrolet, Hyundai, Volkswagen and many more.

Although many advantages are related with the adoption of the electrical motor, lot of constraints are still unresolved such as the battery range or the increasing temperature [1]. Hence another approach in the automotive energy recovery field has been developed: Exhaust Gas Recirculation (EGR) in Internal Combustion (IC) engine vehicles. ICs do not totally convert the chemical energy into mechanical one. In fact, as shown in the Fig.1.1, the greater part of a typical fuel path is wasted in the coolant system and in the exhaust. Only 25% of the energy flows into specific vehicle operations [2].

![Fig. 1.1 Energy flow path in internal combustion engine](image)

According to “Global Automotive Exhaust Gas Recirculation System Market Size, Share, Trends, Growth and Regional Forecast Industry Report” the EGR system market would grow rapidly up to 2023. The technique is based on the reduction of nitrogen oxide in the environment due to the gasoline and diesel engines of automobiles [3]. In fact, as MAHLE GmbH (Germany) believes, “combustion engine will remain an important drive for a long period of time” [4]. Among the companies which produce EGR systems, there are also
For a better understanding and for supporting market analysis, the EGR system market could be organized in different segmentations:

- Product type: EGR valve (Fig. 1.2a), EGR Cooler (Fig. 1.2b), EGR pipe (Fig. 1.2c);
- Vehicle Type: Passenger vehicle, light commercial vehicle, heavy and commercial vehicle;
- Application: Gasoline, Diesel;
- Regions: North America, Europe, Asia, Rest of the world.

Geographically, the market, in the last two years, raised especially in China and Japan. The reason lies in the fact that those countries did not suffer from the global economic crisis of 2008-2009. Instead they demonstrated a continuous development, particularly in the automotive field, which leaded to great sales and more improvements.

**AIM AND THESIS LAYOUT**

The exhaust heat recovering is a potential solution in saving fuel and reducing dangerous emissions, with low costs and remarkable efficiency improving. The thermoelectric conversion, based on the Seebeck effect, represents the beginning of this energy recovering project.

The present work consists in developing a device by means of the usage of a TEG, which is placed on the exhaust, to recharge the battery of a vehicle. The maximum power has to be transferred from the TEG to the load through a DC-DC converter, which aims at regulating parameters into a specific range (nominal voltage of the battery 12 V).

The usage of Maximum Power Point Tracking algorithms allow reaching the best working point, in order to get the best performance of the system.
Hence the objective of the thesis could be explained as well by Fig. 1.3:

![Diagram of energy flow](image)

Fig. 1.3 Thesis objective scheme

Prototypes are realized and tested allowing the observation of the correctness of the design and the power dissipations and efficiency calculations. Moreover in the final project a bypass flap actuator would be connect to the device and implemented in the vehicle in order to shut off quickly dangerous hot gasses from the exhaust, because the purpose is to guarantee the life and reliability of the entire energy recovery system in all situations.

The layout of the thesis is planned in chapters to describe separately the different elements of the entire project. In fact second chapter illustrates the functioning of the TEG, the third the DC-DC converter. Then the control and the simulation parts are explained in details, defining initially the MPPT and its various algorithms and implementations. Conclusions are made with some mentions about future developments.
Thermoelectric generator

SEEBECK EFFECT AND PARAMETERS

Thomas Johann Seebeck discovered in 1822 that two electrical conductors or semiconductors produce a voltage difference in the presence of a temperature gradient. Therefore, the Seebeck effect is the phenomenon consisting in the conversion of a temperature difference into electrical energy. This behavior is widely used with thermocouples for the temperature measurements and generally with thermoelectric generators.

The simplest TEG module is made of a couple of N-type and P-type materials connected at their end. As shown in the figure below (Fig. 2.1), the semiconductors are placed electrically in series and thermally in parallel. When the gradient of temperature $\Delta T$ is high enough, the electrons flow from the hot to the cold side of the materials, generating an electromotive force. Hence, if a circuit is attached, a direct current is generated [5].

![Fig. 2.1 Thermoelectric module - Seebeck effect](image)

The electromotive force $e_{TEG}$ produced is directly proportional to the temperature difference $T_{hot} - T_{cold}$ and it is specified by the Seebeck coefficient $\alpha$, unique for each material used[6]:

$$e_{TEG} = \alpha \Delta T = \alpha (T_{hot} - T_{cold}) \quad (1)$$

As validated from the equation (1), there is a linear correlation between the voltage produced and the Seebeck coefficient $\alpha$. Therefore, to guarantee an appropriate performance of the thermoelectric (TE) material, a significant Seebeck coefficient value is needed. If it is low the voltage across the conductors could be not large enough. Each grade of temperature can produce only few millivolts. To increase the output voltage, a possible solution consists in positioning many TE devices in series. Moreover, if the output current has to be risen, the TE modules have to be placed in parallel (Fig. 2.2).
Other parameters need to be considered. In fact, the efficiency of a TE material depends mainly on the figure of merit $ZT$, based on Seebeck coefficient $\alpha$, heat conductivity $\lambda$ and electrical resistivity $\rho$:

$$ZT = \frac{\alpha^2 T}{\rho \lambda}$$  \hspace{1cm} (2)

Combinations of high Seebeck coefficient, low conductivity and low electrical resistivity permits an adequate value of the figure of merit, the main characterization of TE modules. $ZT$, as depicted in the equation (3) below, assesses the efficiency and the quality of the conversion of the heat into electricity.

$$\eta = \frac{\Delta T}{T_{hot}} \left(\frac{\sqrt{1 + ZT} - 1}{\sqrt{1 + ZT} + T_{cold}/T_{hot}}\right)$$  \hspace{1cm} (3)

An important aspect to consider is also the material used to shape the TE modules. There are several TEG created for several project, thus using very different materials each offering advantages, in terms of efficiency or cost. An improvement could be making the device flexible. Examples are polydimethyloxane (PDMS) based TEGs or SU-8 based TEGs [7]. Instead among more conventional TEG, materials commonly employed are Bismuth Telluride (Bi2Te3) and Lead Telluride (PbTe), which are characterized by figure of merit $ZT = 1$. Nowadays the market is becoming more and more challenging and finding new materials which can have higher figure of merit is the purpose. The research is focused on the increasing of the Seebeck coefficient or reducing the thermal conductivity. In order to achieve those goals it is common to study the nanostructure of the materials.
MODELLING AND SIMULATIONS

The characteristic of the single cell which contains a TE module is shown above in Fig. 2.3. The maximum power and the open circuit voltage are represented as a function of the delta temperature $\Delta T$. The values are extracted from the datasheet, given by Magneti Marelli, and imported in Matlab (first graph Fig. 2.4). Because of the Seebeck effect, it is simple to notice that the higher the gradient of temperature, the higher the output voltage $V_{oc}$ across the TE modules.

From the elaboration of these data, the coefficients of the voltage ramp, function of the temperature, are computed:

$$V_{OC} = m_v \cdot \Delta T + q_v \quad (4)$$

With

$$q_v = -0.039636 \quad m_v = 0.045785$$

From the reconstruction of $V_{oc}$, it is obtained the relation between current and voltage at different temperature gradients (second graph Fig. 2.4). The star points stand for the maximum power at each temperature. The third graph of Fig. 2.4 represents the equivalent resistance characteristic and its linearization. The coefficients are calculated and expressed below:

$$R_{eq} = m_r \cdot \Delta T + q_r \quad (5)$$

With

$$q_r = 1.2111 \quad m_r = 0.0018764$$

Therefore the obtained electrical model of a single cell is defined by equations (4) and (5), that is a voltage source with an internal resistance.
The objective is to transfer the maximum power from the thermoelectric generator to the load. Hence, the output voltage and current of the TEG should be the values corresponding to the star points in the second graph, which are exactly the half of their maximum values (6):

\[ V_{mp} = \frac{V_{DC}}{2} \]  \hspace{1cm} (6)

\[ I_{mp} = \frac{P_{MAX}}{V_{mp}} \]  \hspace{1cm} (7)

Vmp and Imp will be then the inputs of the DC-DC converter described in next chapters.

The results of a single cell would not be enough for an adequate performance of the entire system. In fact, the thermoelectric generator used in the present work, is a combination of 24 cells connected in series and parallel; in order to have a total output voltage less than 60 V the final configuration is 6 in series and 4 in parallel (Fig. 2.5):
Different data are then examined and imported in Matlab. In Fig. 2.6 three different graphs are shown:

- Fig. 2.5 TEG structure used

- Fig. 2.6 TEG characteristic 24 cells on Matlab

The results change because of the addition of the new cells. Take into account that:

\[ ns = \text{number of cells in series} \]

\[ np = \text{number of cells in parallel} \]
The obtained values are upgraded in this way:

\[ V_{OCTot} = V_{OC} \times ns \] (8)

\[ P_{MAXtot} = P_{MAX} \times ns \times np \] (9)

With:

\[ q_{vtot} = -0.23782 \]

\[ m_{vtot} = 0.27471 \]

\[ q_{rtot} = 1.8166 \]

\[ m_{rtot} = 0.0028146 \]

And moreover:

\[ V_{mp} = \frac{V_{OCTot}}{2} \] (10)

\[ I_{mp} = \frac{P_{MAXtot}}{V_{mp}} \] (11)

In next chapters with the annotation Vmp and Imp it is considered the output voltage and current from the TEG with 24 cells.

In the Simulink environment the input of the TEG is represented by the \( \Delta T \), that is the temperature difference between the hot and the cold side of the all TE modules. The behavior is simulated with a constant value or with a slow ramp. Inside the green block (Fig. 2.7) the TEG is then modeled in a circuit as depicted below (Fig. 2.8):
The parameters calculated in the workspace are transferred to the voltage generator and to the variable resistance. To evaluate the correctness of the model, a controlled current source, supplied with a current ramp, is applied at the output of the TEG, to simulate a load (Fig. 2.9):

The performance of the TEG seems to behave as the supposed one. As in Fig 2.10, current and voltage have the opposite orientation and the maximum power is obtained at the half of their maximum values. In this case, with $\Delta T = 150^\circ C$, $P_{\text{max}} = 189 \text{ W}$; $V_{\text{mp}} = 20.6 \text{ V}$; $I_{\text{mp}} = 9.2 \text{ A}$. 
Fig. 2.10 Simulink TEG simulation
DC-DC TOPOLOGIES AND CONSTRAINTS

The present thesis deals with the realization of a DC-DC converter to take advantage of the voltage across the output of the TEG. DC-DC converters can regulate an unregulated input voltage for a specific load [8] (Fig. 3.1).

![Diagram of DC-DC converter regulation scheme]

- Efficiency is generally high, from 70% to 95%;
- They can be designed with very small components: on one hand high efficiency requires less coolers, on the other hand high frequencies permits the usage of reduced components. These converters are usually 80% lighter with respect to the analogous linear supplies [10];
- Smaller components correspond to smaller packaging and then lower costs;
- They could have multiple outputs which could be higher or lower values with respect to the input and also with different polarity.

**Drawbacks** are also presented:

- DC-DCs are complex to design and especially to control;
- They produce high electromagnetic noise;
- They are slow systems: with a reduced bandwidth if the load changes, the output voltage changes slower.

The device to be realized has to be light, easy to implement on a car, cheap and functional. DC-DC converters are adequate for this purpose, and the various limitations presented above could be considered negligible compared to the amount of advantages regarding this choice.

Below in the Table 1 there is a list of the main DC-DC converters which could be considered to be connected to the output of the TEG. The table shows the electrical circuit of each topology, the operation mode (Vi = input voltage; Vo = output voltage) and the TEG eligibility. Green point approves the converter for the present task; yellow point does not mean to be the optimal solution; red point considers the converter not suitable.
Buck converter is a basic topology which permits to step down the input voltage to a specific output voltage reference. Boost converter, instead, is a step-up converter: it increases the voltage input to a higher value. Buck-Boost converter represents the perfect combination of the above converters, allowing the working mode in every situation. Therefore, only three of the aforementioned topologies in the table are acceptable for the application: Boost and Buck cascaded, Non Inverting Buck-Boost (NIBB) and SEPIC. Considering preliminary losses calculation, complexity, number of components and cost, the development is focused on the non-inverting buck-boost topology.
NON INVERTING BUCK-BOOST CONVERTER (NIBB)

The electrical circuit depicted in the Fig. 3.2 represents the basic topology from which the modelling started.

![Fig. 3.2 Non inverting Buck-Boost converter electrical circuit](image)

This is a **two-switch non inverting Buck-Boost converter**, modeled with two switches, a single inductor, input and output capacitors and two diodes. Then the converter is characterized by diode (or nonsynchronous) rectification. Two different parts are distinguishable: the first stage is the Buck leg, consisting of switch SW1 and diode D1; while the second stage is defined Boost leg, made of switch SW2 and diode D2. The voltage source represents the input of the DC-DC and thus the output of the TEG. The load, now is considered as a simple resistor; next it will be replaced by a battery.

Considering the Buck leg, when SW1 is closed (Fig.3.3), hence when it is ON, the current flow and passed through the inductor L also because the diode D1 stop it (it is in OFF mode).

![Fig. 3.3 NIBB circuit SW1 on; D1 off](image)

When SW1 is OFF, diode D1 is ON and it permits the flowing of the current from the down branch to the inductor (Fig.3.4):
The pictures above illustrated the full Buck operation mode of the circuit. Main parameters are summarized in Table 2:

<table>
<thead>
<tr>
<th>Operation Mode</th>
<th>Input Voltage</th>
<th>SW1</th>
<th>D1</th>
<th>d1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Full Buck</td>
<td>$V_{in} &lt; (V_{out} - p)$</td>
<td>ON</td>
<td>OFF</td>
<td>100 %</td>
</tr>
<tr>
<td></td>
<td></td>
<td>OFF</td>
<td>ON</td>
<td>0 %</td>
</tr>
</tbody>
</table>

Table 2

Same reasoning could be applied to the Boost leg: when SW2 is ON, D2 is OFF (Fig. 3.5)

Fig. 3.4 NIBB circuit SW1 off; D1 on

and when SW2 is OFF the D2 is considered in ON mode (Fig. 3.6).
Modeling, Design and Control of a DC-DC converter for Automotive Thermo-electric Generator

In Table 3 full Boost operation mode is defined:

<table>
<thead>
<tr>
<th>Operation Mode</th>
<th>Input Voltage</th>
<th>SW2</th>
<th>D2</th>
<th>d2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Full Boost</td>
<td>( V_{in} &gt; (V_{out} + p) )</td>
<td>ON</td>
<td>OFF</td>
<td>100 %</td>
</tr>
<tr>
<td></td>
<td></td>
<td>OFF</td>
<td>ON</td>
<td>0 %</td>
</tr>
</tbody>
</table>

Table 3

Consequently, the important aspect of this circuit topology is the synchronization of the switches (so the duty cycle) because their opening and closing can cause different working operations. Taking into consideration the value of the input voltage, the Buck-Boost operation mode in this circuit topology is possible when (Table 4):

<table>
<thead>
<tr>
<th>Operation Mode</th>
<th>Input Voltage</th>
<th>SW1</th>
<th>D1</th>
<th>D2</th>
<th>d1</th>
<th>d2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Buck-Boost</td>
<td>( (V_{out} - p) &lt; V_{in} &lt; (V_{out} + p) )</td>
<td>switching</td>
<td>ON and OFF</td>
<td>variable</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 4

The peculiar condition of the Buck-Boost mode is based on the switching of both SW1 and SW2, hence on a variable (= var in Table 4) duty cycle. Then also diodes turn ON and OFF very fast.

It is necessary to remember that the input voltage depends on the maximum power of the TEG \( (V_{in} = V_{mp}) \), hence on the temperature gradient. Different temperatures means different operation modes. In our case the load to be charged is a battery of 13.5 V \( (V_{out} = V_{batt}) \). From the dataset given by Magneti Marelli, when \( V_{mp} \) is approximately close to 13.5V the delta temperature is around 100°C. Buck-Boost region is defined with the parameter \( p \), then a little up and a little down the voltage of the battery.

With \( p = 2 \), the distinction of the three regions are shown in the table (Table 5) and figure below (Fig. 3.7):

<table>
<thead>
<tr>
<th>Working Mode</th>
<th>Input Voltage [V]</th>
<th>Input Temperature [°C]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Boost</td>
<td>( V_{mp} &lt; 11.5 )</td>
<td>( \Delta T &lt; 85 )</td>
</tr>
<tr>
<td>Buck</td>
<td>( V_{mp} &gt; 15.5 )</td>
<td>( \Delta T &gt; 113 )</td>
</tr>
<tr>
<td>Buck-Boost</td>
<td>( 11.5 &lt; V_{mp} &lt; 15.5 )</td>
<td>84 &lt; ( \Delta T &lt; 114 )</td>
</tr>
</tbody>
</table>

Table 5
Fig. 3.7 Operation mode Zones
DESIGN

Switching Frequency Selection

The first task in designing a DC-DC converter is to set the switching frequency $f_{sw}$. The value of $f_{sw}$ takes into account the power level of the system and the specifications of all the components. This choice involves the management of reaching some compromises. In fact the higher is the frequency the smaller are the filter components and faster is the response of the system [11]. On the other hand, at high frequencies the power dissipations of the switches become no more negligible and the correct behavior of the system could not be guaranteed. Moreover the inductor would be larger and heavier.

DC-DC converters are generally characterized by two operating mode, according to the current which flows through the inductor:

- Continuous Conduction Mode (CCM): the inductor current $i_L$ fluctuates but never reaches zero; (Fig. 3.8 represents a general inductor current in a Buck converter in CCM)

![Fig. 3.8 Inductor current Buck converter CCM](image)

- Discontinuous Conduction Mode (DCM): there is an interval in which $i_L$ is equal to zero. (Fig. 3.9 represents a general inductor current in a Buck converter in DCM)

![Fig. 3.9 Inductor current Buck converter DCM](image)
**Inductor Selection**

The selection of the inductor should be based on various factors [12]:

- Peak-to-peak inductor ripple current (*ripple*), which is set at 0.3;
- RMS currents;
- Saturation current;
- Size, weight and cost.

The equations to choose the inductor are:

\[
L_{\text{Buck}} = \frac{V_{\text{batt}}(V_{mp} - V_{\text{batt}})}{\text{ripple} \cdot I_{out}V_{mpfsw}} \quad (12)
\]

\[
L_{\text{Boost}} = \frac{V_{mp}^2(V_{\text{batt}} - V_{mp})}{\text{ripple} \cdot I_{out}V_{battfsw}^2} \quad (13)
\]

In the Buck-Boost region the value of the inductance is maintained constant. Fig. 3.10 represents the inductance as a function of the delta temperature $\Delta T$ for different values of switching frequency:

![Inductor L](image)

Fig. 3.10 Inductance at several frequencies

Selecting the inductor is the fundamental step for DC-DC design but could be critical because it defines also the operating mode of the converter. Specifying the chosen inductor with $L_{\text{dcdc}}$:

\[
L_{\text{dcdc}} > L_{\text{Buck}} = CCM
\]

\[
L_{\text{dcdc}} < L_{\text{Buck}} = DCM
\]
The same reasoning is applied to $L_{\text{boost}}$:

\[
\begin{align*}
L_{\text{dc}} & > L_{\text{Boost}} = \text{CCM} \\
L_{\text{dc}} & < L_{\text{Boost}} = \text{DCM}
\end{align*}
\]

Choosing to stay in CCM mode, it is convenient to pick an inductor with higher values with respect to the ones represented by the plots for each frequency. For this reasoning and considering also the size and the cost of the component, the inductor chose is $8.2 \mu\text{H}$. Then the switching frequency is set at $100 \text{kHz}$.

**Duty Cycle Computations**

The previous equations illustrates how the inductance depends on duty cycle. The parameter in fact is associated to the selection of the majority of the components and it affects the behavior of the current which flows in the circuit. There are different ways to calculate the duty cycle because of the operation mode in which the converter is working. Table 6 depicts the equations in CCM, instead Table 7 is related to DCM:

<table>
<thead>
<tr>
<th>Converter</th>
<th>CCM</th>
</tr>
</thead>
<tbody>
<tr>
<td>Buck</td>
<td>$d_{\text{CCM}} = \frac{V_{\text{batt}}}{V_{\text{mp}}}$</td>
</tr>
<tr>
<td>Boost</td>
<td>$d_{\text{CCM}} = 1 - \frac{V_{\text{mp}}}{V_{\text{batt}}}$</td>
</tr>
</tbody>
</table>

Table 6

<table>
<thead>
<tr>
<th>Converter</th>
<th>DCM</th>
</tr>
</thead>
<tbody>
<tr>
<td>Buck</td>
<td>$d_{\text{DCM}} = \sqrt{\frac{4L_{\text{dc}}f_{\text{sw}}I_{\text{out}}V_{\text{batt}}}{V_{\text{mp}}(V_{\text{mp}} + 2V_{\text{batt}})}}$</td>
</tr>
<tr>
<td>Boost</td>
<td>$d_{\text{DCM}} = \sqrt{\frac{2L_{\text{dc}}f_{\text{sw}}I_{\text{out}}(V_{\text{batt}} - V_{\text{mp}})}{V_{\text{mp}}}}$</td>
</tr>
</tbody>
</table>

Table 7

At 100 kHz, duty cycles are computed and compared to the value of the input voltage, so to the operating mode of the converter.
The plots in Fig. 3.11 are obtained in Matlab also through some considerations about the distinction between CCM and DCM mode. The criteria used to define the two modes are these:

\[
\frac{i_{\text{ripple}}}{2I_{mp}} > 1 =\Rightarrow \text{Boost DCM} \quad (14)
\]

\[
\frac{i_{\text{ripple}}}{2I_{out}} > 1 =\Rightarrow \text{Buck DCM} \quad (15)
\]

The equations express that if the half ripple current of the inductor is greater than the average current of L, the working mode change from CCM to DCM. The aim is to be restricted in the CCM zone.

**Capacitors Selection**

Both input and output capacitor selection should not be underestimated because they are very stressed components.

For a Buck converter the selection of the input capacitor is critical because it can keep the electromagnetic noise due to the high current which flows from the source.

For a Boost converter, instead, the output capacitor is very stressed. In fact, when the switch is ON, all the current comes from the output capacitor to the load. (Buck converter, differently, delivers always current to the load from the inductor)
In any case some electrical specifications are required when selecting the right capacitors in a DC-DC design [13]:

- RMS current: it is the main parameter to be considered. There are many way to compute it. The general equation, valid for every topology of RMS current is the (16):

\[
I_{RMS} = \sqrt{\frac{I_{min}^2 + I_{min}I_{MAX} + I_{MAX}^2}{3}} \tag{16}
\]

If the current is a triangle wave, it is sufficient to follow the equation (17):

\[
I_{RMS\Delta} = \frac{I_{peak}}{\sqrt{3}} \tag{17}
\]

Another way consists in the RMS itself: Root, Mean and Square the current. Otherwise, one common method is the usage, when possible, of the flat top approximation.

In the Fig. 3.12 the RMS currents of input and output capacitors are revealed, results of Matlab computations. The capacitor have to handle surely 3.5 A.

- Voltage applied on capacitor (from 12 V for the battery to 25 V from the source)
- Capacitor impedance ESR, which has to be as low as possible.

All those considerations lead to select capacitors of 330 µF with ESR equal to 41 mOhm.
Power dissipations and Efficiency

This subsection is related to various computation in order to obtain the general overview of the power dissipations of the DC-DC converter and thus its efficiency. From each component the main losses are computed.

**INDUCTOR**

The component is characterized by two different power losses: conduction loss and magnetic core loss. Only conduction loss would be considered and it depends on the ESR resistance which is due to the windings of the inductor. ESR value is directly proportional to the inductance; while it is inversely proportional to the case size. As the equations illustrate in Table… the power loss is equivalent to the square of the current and it is not affected by duty cycle. It means that the higher the average current of the inductor, the higher power dissipation [14] and it is well depicted in Fig. 3.13.

<table>
<thead>
<tr>
<th>Converter</th>
<th>Power dissipated from the Inductor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Buck</td>
<td>$P_L = \left( \frac{I_{\text{out}}^2}{12} + \frac{I_{\text{ripple}}^2}{12} \right) L_{\text{ESR}}$</td>
</tr>
<tr>
<td>Boost</td>
<td>$P_L = \left( \frac{I_{\text{mp}}^2}{12} + \frac{I_{\text{ripple}}^2}{12} \right) L_{\text{ESR}}$</td>
</tr>
<tr>
<td>Buck-Boost</td>
<td>$P_L = \left( \frac{I_{\text{mp}}^2}{12} + \frac{I_{\text{ripple}}^2}{12} \right) L_{\text{ESR}} + \left( \frac{I_{\text{out}}^2}{12} + \frac{I_{\text{ripple}}^2}{12} \right) L_{\text{ESR}}$</td>
</tr>
</tbody>
</table>

Table 8

![Inductor Conduction Losses](image)

Fig. 3.13 Inductor Conduction Losses
CAPACITORS
The general power dissipation model adopted for the capacitors is defined by the equations…:

\[ P_{\text{cin}} = I_{\text{RMS}}^2 C_{\text{cin}} \text{ESR} \]  \hspace{1cm} (18)

\[ P_{\text{cout}} = I_{\text{RMS}}^2 C_{\text{cout}} \text{ESR} \]  \hspace{1cm} (19)

The losses are calculated multiplying the square of the RMS current with the ESR of the capacitors. The picture below (Fig. 3.14) represents the stress of the input and output capacitors. The first one is more stressed in Buck mode, the output one usually works more in Boost region, but at high temperatures, so high powers, it is stressed as before.

![Input Capacitor Power losses](image1)

![Output Capacitor Power losses](image2)

Fig. 3.14 Capacitors Losses

DIODES
The selection of a proper diode depends on some parameters:

- Average current \( (I_{\text{ave}}) \);
- Maximum peak current;
- Maximum reverse voltage \( (V_f) \) (which is extracted from the diode characteristic, obtained from the datasheet of the selected component in Fig. 3.15).
The characteristic shows the reverse voltage as a function of the instantaneous forward current on a logarithmic graph. Therefore the power dissipated is then computed:

\[ P_{D1} = V_{f1}I_{ave1} \]  

\[ P_{D2} = V_{f2}I_{ave2} \]

And the result is depicted in the pictures below (Fig 3.16):

---

**Fig. 3.15 Diodes Characteristic**

**Fig. 3.16 Diodes Losses**
The second diode, on the Buck leg, seems to dissipates 15 W, it is extremely high loss just for one diode. A possible solution consist in put two output diodes in parallel, in order to dissipates at least half of this power.

**SWITCHES**

Switches losses are the most relevant dissipations and they depend mostly on the duty cycle, thus on the different operation modes of the DC-DC converter. In the following tables are represented the power losses equation of both SW1 and SW2, in Buck, Boost or Buck-Boost region. The main dissipations are due to the conduction (duty cycle = 1) and switching (duty cycle = var) conditions. Moreover also the output recharging capacitance (Coss) is considered.

<table>
<thead>
<tr>
<th>LOSSES</th>
<th>SW1</th>
<th>SW2</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Conduction loss</strong></td>
<td>$P_{con} = \left( I_{out}^2 + \frac{I_{ripple}^2}{12} \right) R_{ds on} d_{Buck}$</td>
<td>$P_{con} = 0$</td>
</tr>
<tr>
<td><strong>Switching loss</strong></td>
<td>$P_{sw} = V_{mp} I_{out} (t_R + t_F) \frac{f_{sw}}{2}$</td>
<td>$P_{sw} = 0$</td>
</tr>
<tr>
<td><strong>Output Capacitance Coss loss</strong></td>
<td>$P_{Coss} = C_{oss} V_{mp}^2 \frac{f_{sw}}{2}$</td>
<td>$P_{Coss} = 0$</td>
</tr>
</tbody>
</table>

Table 9

<table>
<thead>
<tr>
<th>LOSSES</th>
<th>SW1</th>
<th>SW2</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Conduction loss</strong></td>
<td>$P_{con} = \left( I_{mp}^2 + \frac{I_{ripple}^2}{12} \right) R_{ds on} d_{Boost}$</td>
<td>$P_{con} = \left( I_{mp}^2 + \frac{I_{ripple}^2}{12} \right) R_{ds on} d_{Boost}$</td>
</tr>
<tr>
<td><strong>Switching loss</strong></td>
<td>$P_{sw} = V_{batt} I_{mp} (t_R + t_F) \frac{f_{sw}}{2}$</td>
<td>$P_{sw} = V_{batt} I_{mp} (t_R + t_F) \frac{f_{sw}}{2}$</td>
</tr>
<tr>
<td><strong>Output Capacitance Coss loss</strong></td>
<td>$P_{Coss} = C_{oss} V_{batt}^2 \frac{f_{sw}}{2}$</td>
<td>$P_{Coss} = C_{oss} V_{batt}^2 \frac{f_{sw}}{2}$</td>
</tr>
</tbody>
</table>

Table 10

<table>
<thead>
<tr>
<th>LOSSES</th>
<th>SW1</th>
<th>SW2</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Conduction loss</strong></td>
<td>$P_{con} = \left( I_{out}^2 + \frac{I_{ripple}^2}{12} \right) R_{ds on} d_{Buck}$</td>
<td>$P_{con} = \left( I_{mp}^2 + \frac{I_{ripple}^2}{12} \right) R_{ds on} d_{Boost}$</td>
</tr>
<tr>
<td><strong>Switching loss</strong></td>
<td>$P_{sw} = V_{mp} I_{out} (t_R + t_F) \frac{f_{sw}}{2}$</td>
<td>$P_{sw} = V_{batt} I_{mp} (t_R + t_F) \frac{f_{sw}}{2}$</td>
</tr>
<tr>
<td><strong>Output Capacitance Coss loss</strong></td>
<td>$P_{Coss} = C_{oss} V_{mp}^2 \frac{f_{sw}}{2}$</td>
<td>$P_{Coss} = C_{oss} V_{batt}^2 \frac{f_{sw}}{2}$</td>
</tr>
</tbody>
</table>

Table 11
Modeling, Design and Control of a DC-DC converter for Automotive Thermo-electric Generator

<table>
<thead>
<tr>
<th>Boost in Buck-Boost region</th>
<th>SW1</th>
<th>SW2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Conduction losses</td>
<td>$P_{\text{con}} = \left( I_{\text{out}}^2 + \frac{I_{\text{ripple}}^2}{12} \right) R_{\text{dson dBoost}}$</td>
<td>$P_{\text{con}} = \left( I_{\text{mp}}^2 + \frac{I_{\text{ripple}}^2}{12} \right) R_{\text{dson dBoost}}$</td>
</tr>
<tr>
<td>Switching Losses</td>
<td>$P_{\text{sw}} = V_{\text{mp}} I_{\text{out}} (t_R + t_F) \frac{f_{\text{sw}}}{2}$</td>
<td>$P_{\text{sw}} = V_{\text{batt}} I_{\text{mp}} (t_R + t_F) \frac{f_{\text{sw}}}{2}$</td>
</tr>
<tr>
<td>Coss</td>
<td>$P_{\text{Coss}} = C_{\text{oss}} V_{\text{mp}} ^2 \frac{f_{\text{sw}}}{2}$</td>
<td>$P_{\text{Coss}} = C_{\text{oss}} V_{\text{batt}} ^2 \frac{f_{\text{sw}}}{2}$</td>
</tr>
</tbody>
</table>

As seen from the tables, the Buck-Boost region is divided in two parts, evaluating the different duty cycles whether up or down from the battery reference (13.5 V).

Following pictures shows the Matlab losses computation results:

![HighSide Switch (SW1) Conduction Losses](image1)

![LowSide Switch (SW2) Conduction Losses](image2)

Fig. 3.17 Switches Conduction Losses
Additionally another value has to be esteemed: the gate charging loss in equation (22):

\[ P_{gate} = Q_{gs} V_{gs} f_{sw} * 2 \quad (22) \]
In conclusion, the amount of the overall power dissipation of the DC-DC converter is made of all the previous contributions, as depicted in the equation (23):

\[ P_{\text{diss}} = P_L + P_{\text{cin}} + P_{\text{cout}} + P_{D1} + P_{D2} + P_{\text{con}1} + P_{\text{sw}1} + P_{\text{Coss}1} + P_{\text{con}2} + P_{\text{sw}2} + P_{\text{Coss}2} + P_{\text{gate}} \]

Knowing that

\[ P_{\text{out}} = P_{\text{in}} - P_{\text{diss}} \quad (24) \]

It is possible to calculate the efficiency:

\[ \eta = \frac{P_{\text{out}}}{P_{\text{in}}} \times 100 \quad (25) \]

And to observe the final values on the figures of Fig. 3.20.

Fig. 3.20 Pin/Pout, Dissipated Power and Efficiency
Control algorithms and simulations

MPPT, ALGORITHMS AND IMPLEMENTATIONS

Maximum Power Transfer Theorem

The primary objective of the project is to realize a device capable to recharge the battery through the heat dissipated from the exhaust of a vehicle. To ensure enough energy and to guarantee the best performance of the entire system, it is necessary to transfer the maximum power from the TEG to the load. As seen in the previous chapter, the TEG can be definitely modelled as a voltage generator with a resistor in series, which depends on the material used for the TE modules. Therefore it is possible to apply the maximum power transfer theorem. It states that the maximum power can be transferred when the total resistive value of the load is equal to the internal resistance of the source. The output voltage and current of the TEG reveal a linear relation between them. The output power has a quadratic behavior, a parabolic trajectory with a distinguishable maximum point. (Fig.4.1) Those values are functions of the temperature difference of the TE modules.

![Fig. 4.1 Power parabolic behavior - Matlab](image)

For each curve at given temperature there is one maximum power point, consequently, Maximum Power Point Tracking Techniques (MPPT) have to be developed, modified and applied to our system. For any specific condition, the MPPT can give a different reference, important for the DC-DC control implementation.

The literature illustrates two main fundamental algorithms: P&O (Perturbation and Observation) and IC (Incremental Conductance) algorithms. They will be described briefly in these pages, because from them the modified algorithm would be computed.


**Perturb and Observe (P&O) algorithm**

Among the MPPT methods, P&O represents the simplest algorithm to enhance the performance in a simple way. The basic procedure lies in perturbing the voltage or the current of the TEG and then observing the results on the power generated [15].

\[
dP/dV = 0 = MPP
\]
\[
dP/dV > 0 = left of MPP
\]
\[
dP/dV < 0 = right of MPP
\]

For each cycle the previous power and voltage are compared to the new one; the effect consists in decreasing or increasing continuously the voltage reference. Thus, the working point is incessantly moving toward the MPPT to guarantee the maximum performance.
Instead of having a reference voltage, it is possible to have a duty cycle reference and apply it directly to the control of the DC-DC converter.

The behavior of the algorithm could be improved by changing some specifications, for example the step of the reference (voltage or duty cycle) or the sampling period.

**Incremental Conductance (IC) algorithm**

The algorithm is developed to improve the previous one in some limitations, especially convergence speed and steady-state error [16]. As shown by the diagram of Fig. 4.3 and the equations of the box, it is used the concept of the derivative of the power. The result could be seen in a faster response and precision.

\[
\frac{dP}{dV} = 0
\]

\[
P = V I
\]

\[
\frac{dP}{dV} = \frac{d(VI)}{dV} = I + V \frac{dI}{dV}
\]

\[
0 = I + V \frac{dI}{dV}
\]

\[
\frac{dI}{dV} = -\frac{I}{V}
\]

When:

\[
\frac{dI}{dV} = -\frac{I}{V} = MPP
\]

\[
\frac{dI}{dV} > -\frac{I}{V} = \text{left of MPP}
\]

\[
\frac{dI}{dV} < -\frac{I}{V} = \text{right of MPP}
\]
The usage of those equations permits to choose the best following operating point.

**Modified algorithms and implementation**

As shown in the Fig. 4.4 the Simulink environment is made by three blocks: TEG block (green), Measuring block (red) and MPPT block (blue). Instead of having directly the DC-DC converter, the load of the system is represented by a variable resistor. Therefore, the modified MPPT algorithms would give a resistance reference, from which in next chapters a current reference will be developed for the current control loop.

Now it is important to affirm the correctness of the algorithms and it is made by representing the converter as a simple resistor.

![Fig. 4.4 MPPT Simulink simulation](image)

The modified algorithms realized are three and they are based on the model of the P&O and IC algorithms.

**P&O_1 modified algorithm**

The first algorithm is well depicted in the flowchart of Fig. 4.5. By measuring the power and the voltage of the TEG, it compares the new with the old values and the result is just a step up or down of the resistance. It is always a perturbation because the resistance cannot be constant. The precision of this model is confirmed when, as the maximum power theorem states, the value of the variable resistor reaches the source resistance.
The Matlab function adopted is illustrated in the figure Fig. 4.6.

```matlab
function R = P&O_1_function(R_step, R_old, P_new, V_new, V_old)
    if (P_new > P_old)
        if (V_new > V_old)
            R = + R_step;
        else
            R = - R_step;
        end
    else
        if (V_new > V_old)
            R = - R_step;
        else
            R = + R_step;
        end
    end
end
```

Fig. 4.6 P&O_1 Matlab algorithm
P&O_2 modified algorithm

The flowchart which explained graphically the second modified algorithm is in Fig. 4.7. Because of the resistance reference could not be constant, it would be equal to the positive or negative value of the step for each cycle. For example, if the new power is higher than the old one and if the previous resistance have been increased, it means that the maximum power point is not reached yet. Hence it is necessary to reduce the current (thus to have higher resistance) and to increase the voltage.

As for the previous algorithm, below there is the Matlab algorithm (Fig. 4.8):
The third modified algorithm is just an adjustment of the IC one. The principle again lies on the concept of the maximum power. If the reached point is not considered the maximum the reference has to move. The step is positive or negative with respect to the new value of voltage. The relative flowchart and Matlab function are illustrated below in Fig. 4.9 and Fig. 4.10.

```matlab
function R = P_O_modify(R_step, P_old, P_new, R)
    if (P_new > P_old)
        if (R > 0)
            R = + R_step;
        else
            R = - R_step;
        end
    else
        if (R > 0)
            R = - R_step;
        else
            R = + R_step;
        end
    end
end
```

Fig. 4.8 P&O_2 Matlab algorithm
Fig. 4.9 IC modified flowchart

```matlab
function R = P_O_modify(R_step, P_old, P_new, V_new, V_old)
    if ((P_old-P_new)/(V_old-V_new)) >= 0
        R = +R_step;
    else
        R = -R_step;
    end
```

Fig. 4.10 IC modified Matlab algorithm

This algorithm would be applied to the control. Given a ramp reference, the simulations in Simulink (Fig. 4.11 and Fig. 4.12) shows that after 2 ms the obtained resistance follows the source one.
From the resistance, by applying simple calculations, it is easy to acquire a current reference through equation (26):

$$I_{ref} = \frac{V_{mp}^2}{RV_{batt}} \quad (26)$$
CONTROL DESIGN

To ensure a proper control of the system, it is necessary to control output current and voltage through feedbacks. In order to make it, the right transfer functions should be obtained.

First of all the duty cycle represents the control input of the converter, voltage and current are the outputs (Fig. 4.13):

![Basic control principle scheme](image)

Fig. 4.13 Basic control principle scheme

The picture above depicts two inputs: Buck and Boost duty cycles. In fact the basis of the control of the present system, lies on the differentiation of the two converters. This is due to the aim to simplify the control algorithms, transfer functions and simulations.

The DC-DC converter is made of non-linear element, such as switches and diodes. Therefore the DC-DC is a Non Linear Time Variant system (NLTV). Transfer functions can be found only with stable systems [9].

![From NLTV to LTI system scheme](image)

Fig. 4.14 From NLTV to LTI system scheme

As shown in the Fig. 4.14, the model needs to be averaged and then linearized, for this reason it is used the state space averaging method (known also as Middlebrook method).

The transfer functions for each converter related to the current loop are shown in Table 13, the others related to the voltage loop in Table 14:

<table>
<thead>
<tr>
<th>Mode</th>
<th>Current Loop ( G = \frac{i_{\text{out}}}{d} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Buck</td>
<td>( G_{di} = \frac{R(1 + sC_{\text{out}} \text{ESR})}{(1 + sRC_{\text{out}})} ) (20)</td>
</tr>
<tr>
<td>Boost</td>
<td>( G_{di} = \frac{R(1 - d)}{2} \left( 1 - \frac{sL_{d\text{dc}}}{R(1 - d)^2} \right) \left( 1 + sC_{\text{out}} \text{ESR} \right) \frac{1}{(1 + sRC_{\text{out}}/2)} ) (22)</td>
</tr>
</tbody>
</table>

Table 13

<table>
<thead>
<tr>
<th>Mode</th>
<th>Voltage Loop ( G = \frac{v_{\text{out}}}{d} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Buck</td>
<td>( G_{dv} = \frac{V_{mp}(1 + sC_{\text{out}} \text{ESR})}{(1 + sL_{d\text{dc}}/R + s^2L_{d\text{dc}}C_{\text{out}})} ) (21)</td>
</tr>
<tr>
<td>Boost</td>
<td>( G_{dv} = \frac{V_{mp}}{(1 - d)^2} \left( 1 - \frac{sL_{d\text{dc}}}{R(1 - d)^2} \right) \left( 1 + sC_{\text{out}} \text{ESR} \right) \frac{1 + sRL_{d\text{dc}}}{(1 - d)^2 + s^2L_{d\text{dc}}C_{\text{out}} / (1 - d)^2} ) (23)</td>
</tr>
</tbody>
</table>

Table 14
CONTROL STRATEGIES AND SIMULINK ARCHITECTURES

The initial control approach consists in a dual control loop, made of an inner current loop and an external voltage loop, as depicted in Fig. 4.15.

The controller used in the simulations are PI controllers, which are characterized by the below equation:

\[ P = K_c \left(1 + \frac{K_i}{s}\right) \]  

Defining with \( P_2 \) the function of the external controller, with \( P_1 \) the function of the current controller; the resulting control transfer functions are achieved:

For the Current Loop

\[ TF_1 = \frac{P_1 * G_{di}}{1 + P_1 * G_{di}} \]  

For the Voltage Loop

\[ TF_2 = \frac{P_2 * P_1 * G_{dv}}{1 + P_2 * P_1 * G_{dv}} \]

The right proportional and integral parameter, \( K_c \) and \( K_i \), are the objectives for a proper control. Because of the complexity of the converter and of the control itself, the procedure should be organized in steps:

- Step1: divide the control study in two separated parts: one for Buck and the other for Boost converter;
- Step2: guarantee an accurate and fast current control for each converters;
- Step3: develop the voltage control, slower than the current one, for each converter;
- Step5: compare the different parameters, find the right working zone for both converters and choose the final \( K_c \) and \( K_i \).

First of all, it is important to ensure a fast and reactive behavior of the initial part of all the control strategies: the current loop.
First Control Strategy: Current Loop

The picture in Fig. 4.16 depicts the current control loop in the Simulink environment. The input (in red) is a current and initially it is a constant reference. The output current which comes from the output of the DC-DC (in green) is filtered and then compared to the reference. The difference, thus the error between them is the input of the controller. The output of the PI (in yellow) represents the duty cycle of the converter.

For Buck converters the parameters selected for the PI controller are:

\[
Kc_{Buck} = 0.01
\]

\[
Ki_{Buck} = f_{p_{Buck}} = \frac{1}{2\pi R_{out} C_{out}} = 689.3953
\]

The integral parameter is chosen equal to one pole of the transfer function of the Buck converter. The Fig. 4.17 illustrates how the output current follows the reference, it is zoomed in Fig. 4.18.
For Boost converters the parameters selected for the PI controller are:

\[ Kc_{Boost} = 1 \]
\[ Kf_{Boost} = \frac{f_{p_{Boost}}}{2\pi RC_{out}} = 1378.79 \]

The result is shown in the figures below (Fig. 4.19 and Fig. 4.20)
Second Control Strategy: Dual Loop

The second strategy is based on the complete dual control loop. The inner current loop, now, follows an external voltage loop, hence the output of the first PI controller (in orange) gives a current reference input for the second PI. The output voltage is compared to the battery voltage, the error would be the input of the voltage PI.

Choosing the parameter below

\[ K_{ci} = 0.05 \]
\[ K_{ii} = 1000 \]

For the PI current controller, and

\[ K_{cv} = 2 \]
\[ K_{iv} = 10 \]

For the PI voltage controller, with \( \Delta T=180^\circ C \), in the Simulink environment the results obtained are illustrated in the following plots.

Fig. 4.22 compares the voltage reference (the red ramp) with the output voltage, which after some seconds can reach the voltage reference (chosen at 15 V). In Fig. 4.23 there is the current comparison. It is satisfying, the output current is fast in following the red reference. Moreover it is less oscillating than the reference.
Third Control Strategy: MPPT + Current Loop

The third control approach is made of MPPT and the current control loop (Fig.4.24).

The explanation requires the description of the different tools which are used in the Simulink environment:

- The TEG is the left green scheme and it is defined by the temperature gradient $\Delta T$. Its output current and voltage go directly to supply the DC-DC and to the MPPT block. In addition, also the equivalent resistance value is sent to the MPPT, in order to perform the algorithm;
- The MPPT block is the blue one. Inside the IC modified algorithm is implemented and adjusted with few calculations to obtain a reference current instead of just a resistance;

- The result of the MPPT would be the input of the current control loop, represented by the orange block. Among the inputs of the control there is also the output current of the converter, in order to compare it to the MPPT reference. The output is the duty cycle for the high side (Buck leg) or the low side switch (Boost leg);

- The DC-DC converter is represented by the red block. It receives the parameters from the TEG and the duty cycle from the control block.

The MPPT algorithm applied is the IC modified one and the parameters selected for the PI current controller are always:

\[
K_c = 0.05 \\
K_i = 1000
\]

In the figures below there are some interesting results after the simulation ran at \( \Delta T = 180^\circ C \).

Fig. 4.25 Output/Reference current with error in Simulink using MPPT+Current Loop (180°C)

Fig. 4.26 Output/Reference current Zoom (180°C)

Fig. 4.25 represents the comparison between the reference current (red plot) realized by the MPPT and the output current (blue plot) which is supposed to follow the reference, and actually it can do it. In Fig. 4.26 the view is zoomed and it can be observed that the little red steps are followed by the output with light oscillations. The value of the output current should be approximately around 19 A, in the simulation it is a little lower than 18 A.
The prove that explains the right functioning of the MPPT is demonstrated in Fig. 4.27:

![Fig. 4.27 Source and Load Resistances comparison (180°C)](image)

The load resistance (in blue) is going down to the source resistance reference. It stops before, but it places in stability in a little range. The Buck duty cycle obtained from the control block is depicted in red in Fig. 4.28.

![Fig. 4.28 Duty Cycles obtained at 180°C with MPPT+Current Loop](image)

Also the Boost mode of the converter is tested and controlled, in fact next picture (Fig. 4.29) is related to the Simulink simulation at $\Delta T=50^\circ C$. From previous calculation, the output current should be 1.71 A, the simulation shows a similar behavior a little lower, focused in Fig. 4.30.

In Fig. 4.31, instead, is illustrated the comparison of the source reference resistance (in red) to the load one (in blue), computed by the IC modified algorithm. Even though they are not the same, the result is quite satisfactory.
Fig. 4.29 Output/Reference current with error in Simulink using MPPT+Current Loop (50°C)

Fig. 4.30 Output/Reference current Zoom (50°C)

Fig. 4.31 Source and Load Resistances comparison (50°C)
FUTURE DEVELOPMENTS

The previous control strategies are based on the partition of the Non Inverting Buck-Boost converter in separated Buck and Boost. Thus only one duty cycle is obtained and it is referred to the relative operation mode of the DC-DC. One possible solution is the addition of a new block (Fig. 5.1), which inside (Fig. 5.2) there is a function capable to split one duty reference in two suitable values for SW1 and SW2.

The principle lies in creating two different carriers to be compared to the duty cycle reference [17], from which two driving signals are achieved (Fig. 5.3):
Then the different operation mode (Table 15) is defined by the value of the reference \(d\), which corresponds to the output of the control block and the input of the new one.

<table>
<thead>
<tr>
<th>Operation Mode</th>
<th>Reference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Buck</td>
<td>(-1 &lt; d &lt; V_L)</td>
</tr>
<tr>
<td>Boost</td>
<td>(V_L &lt; d &lt; V_H)</td>
</tr>
<tr>
<td>Buck-Boost</td>
<td>(V_H &lt; d &lt; 1)</td>
</tr>
</tbody>
</table>

And for specific conditions due to the carriers values, the duty cycle of each switch is obtained and shown in Table 16:

<table>
<thead>
<tr>
<th>Duty Cycle</th>
<th>Condition</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>Buck</td>
<td>(d &lt; -1)</td>
<td>(d_{\text{Buck}} = 0)</td>
</tr>
<tr>
<td></td>
<td>(-1 \leq d \leq V_H)</td>
<td>(d_{\text{Buck}} = \frac{1 + d}{1 + V_H})</td>
</tr>
<tr>
<td></td>
<td>(d &gt; V_H)</td>
<td>(d_{\text{Buck}} = 1)</td>
</tr>
<tr>
<td>Boost</td>
<td>(d &lt; V_L)</td>
<td>(d_{\text{Boost}} = 0)</td>
</tr>
<tr>
<td></td>
<td>(V_L \leq d \leq 1)</td>
<td>(d_{\text{Boost}} = \frac{d - V_L}{1 - V_L})</td>
</tr>
<tr>
<td></td>
<td>(d &gt; 1)</td>
<td>(d_{\text{Boost}} = 1)</td>
</tr>
</tbody>
</table>

**IMPLEMENTATION**

The prototypes realized in laboratory are four switch Non Inverting Buck-Boost converters (Fig. 5.4 and Fig. 5.5).

![First lab prototype](image-url)
The choice to adopt the synchronous rectification is made after some considerations about efficiency. In fact, as seen in previous chapters, the diode losses could be very significant in high current applications, they can completely damage the total efficiency of the converter. Replacing two MOSFETs instead of two diodes (basic scheme in Fig. 5.6) would lead to the reducing of conduction losses.

Because of at lower input voltage more conduction losses are generated, the result would be an efficiency improvement especially at small values of Vin.

The circuit, however, it is made of four switches. It means that in Buck-Boost mode each of the four components dissipates switching and conduction powers. Then as a result its efficiency is lowest in Buck-Boost operation mode. What is lost in this case is achieved with the reduction of the conduction losses caused by the diodes, then it is possible with the four switch Buck-Boost converter to reach higher levels of efficiency.
The prototypes have been tested, closing two of the possible switches, in order to calculate the power dissipated, hence the efficiency, of the Buck and of the Boost operation mode.

For every test:

- Specific voltage and current are applied to the input from a power supply;
- Duty cycle is adjusted for the purpose;
- Output voltage and current are measured from a resistive load;
- Input, output and dissipated power are calculated;
- Efficiency is obtained.

Table 17 and Table 18 show the measured values of both operation modes. Then the efficiency is represented as a function of the input voltage in Fig. 5.7 and it is compared to the efficiency calculated previously through the given dataset.

<table>
<thead>
<tr>
<th>Boost Measured Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>---------</td>
</tr>
<tr>
<td>3.3</td>
</tr>
<tr>
<td>4.1</td>
</tr>
<tr>
<td>4.2</td>
</tr>
<tr>
<td>5</td>
</tr>
<tr>
<td>5.6</td>
</tr>
<tr>
<td>6</td>
</tr>
<tr>
<td>6.5</td>
</tr>
<tr>
<td>7</td>
</tr>
<tr>
<td>7.5</td>
</tr>
<tr>
<td>8</td>
</tr>
<tr>
<td>8.45</td>
</tr>
<tr>
<td>9.1</td>
</tr>
<tr>
<td>9.4</td>
</tr>
<tr>
<td>10</td>
</tr>
<tr>
<td>10.5</td>
</tr>
<tr>
<td>11</td>
</tr>
<tr>
<td>11.5</td>
</tr>
</tbody>
</table>

Table 17
### Buck Measured Values

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>14.65</td>
<td>4.5</td>
<td>65.925</td>
<td>11.02</td>
<td>5.2227</td>
<td>57.5547</td>
</tr>
<tr>
<td>14.99</td>
<td>4.8</td>
<td>71.952</td>
<td>11.79</td>
<td>5.5877</td>
<td>65.8787</td>
</tr>
<tr>
<td>15.73</td>
<td>6.5</td>
<td>102.245</td>
<td>12.26</td>
<td>7.4303</td>
<td>91.0955</td>
</tr>
<tr>
<td>16.73</td>
<td>6.8</td>
<td>113.764</td>
<td>12.92</td>
<td>7.8303</td>
<td>101.1675</td>
</tr>
<tr>
<td>17.67</td>
<td>6.5</td>
<td>114.855</td>
<td>12.98</td>
<td>7.8667</td>
<td>102.1093</td>
</tr>
<tr>
<td>18.95</td>
<td>7.4</td>
<td>140.23</td>
<td>12.88</td>
<td>9.6119</td>
<td>123.8018</td>
</tr>
<tr>
<td>19.78</td>
<td>7.1</td>
<td>140.438</td>
<td>12.89</td>
<td>9.6194</td>
<td>123.9941</td>
</tr>
<tr>
<td>20.8</td>
<td>8.6</td>
<td>178.88</td>
<td>12.82</td>
<td>11.8704</td>
<td>152.1781</td>
</tr>
<tr>
<td>21.77</td>
<td>8.2</td>
<td>178.514</td>
<td>12.83</td>
<td>11.8796</td>
<td>152.4156</td>
</tr>
<tr>
<td>22.73</td>
<td>9.9</td>
<td>225.027</td>
<td>12.57</td>
<td>15.1446</td>
<td>190.3673</td>
</tr>
<tr>
<td>24.74</td>
<td>9.7</td>
<td>239.978</td>
<td>12.73</td>
<td>15.3373</td>
<td>195.2445</td>
</tr>
<tr>
<td>24.74</td>
<td>9.5</td>
<td>235.03</td>
<td>12.84</td>
<td>15.4699</td>
<td>198.6333</td>
</tr>
</tbody>
</table>

Table 18

---

**Fig. 4.7 Real and Computed Efficiency Comparison**
CONCLUSION
The final proposed solution, consists in the Non Inverting Buck-Boost converter control by the usage of TI DSP control card (in Fig. 5.7).

Considering also the presence of the carrier board, which is populated with CAN transceivers (for communication), analog connections (i.e. temperature sensors) and signal conditioning circuits, the entire device could be imagined like in the Fig. 5.8.
References


[17] Ilan Aharon, Doron Shmilovitz, Alon Kuperman “Robust output voltage control of multimode non-inverting DC-DC converter”, 2015