UNIVERSITY OF PADUA

DEPARTMENT OF INFORMATION ENGINEERING

DEGREE IN ELECTRONIC ENGINEERING

Loss estimation in a voltage source inverter for electrical drives

Student:
Fabiano FOSSATO

Supervisor:
Dott. Simone BUSO

Academic Year 2013/2014

To my parents.

Contents

Chapter 1

Inverter

In this chapter we will talk about the theoretical analysis of an inverter, analysing the different configurations, the losses, the choice we have done and the models of the losses that we have used to define which heat-sink to use.

1.1 Half Bridge

The half-bridge configuration is a type of configuration that can be used to realize an inverter. The main advantage of this kind of configuration is its simpleness. Only few components are indeed necessary: two transistors in conjunction with two free-wheeling diodes. On the other hand, a three-wires DC source is required or more specifically, two DC voltage sources in input with a common wire.

A configuration like this can be obtained by means of a single DC input voltage realized with two input capacitors connected as shown in Fig.1.1 to obtain a common floating connection. An half-bridge configuration, however, has the disadvantage to require a control of the switching devices in order to guarantee the same voltage across the input capacitors. A full-bridge configuration allows to overcome this limitation. This is one reason why we decided to use a full-bridge configuration in this project.
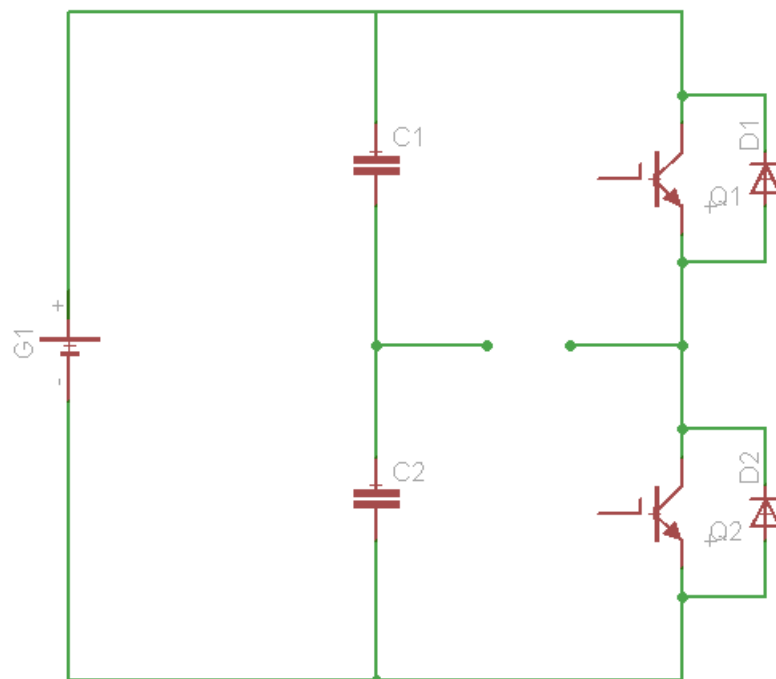
Figure 1.1: Half-bridge configuration

How an half-bridge works

In this section we are going to discuss how an half-bridge works as it is essential to understand how a full-bridge works. Let us consider that the load has both a resistive and a inductive part. The inductance may be provided by the load (e.g. electric motor) or by a filtering device, used to filter the current harmonics on the load.

Analysis: From the diagram in Fig. 1.1 it can be seen that there are two diodes connected in anti-parallel to each switching device. These diodes, called "free-wheeling diodes", assure a reclosing way for the current in every situation. In the case where the load is purely resistive these diodes are useless, but if the load is reactive they are fundamental to prevent a failure of the device. Usually, indeed, the switching devices are current unidirectional so, if the current is forced by the load in the wrong direction, the devices will be destroyed. Thus, free-wheeling diodes are preferably used as they assure a reclosing way for the current in every operating condition. These diodes in power devices are currently included in the package of each switch and usually they are not normal diodes, but the manufacturers take advantage to the internal structure of the switch. For example, thinking about the MOS-structure, there is a p-n junction between the body and the drain, this de facto represents the anti-parallel diode that we need. Designing accurately this junction, the manufacturers assure that this intrinsic diode can tolerate the same current that pass through the MOS in the forward direction. In this way, the device becomes current bidirectional even if the current flow is controllable only when it flows from the drain to the source.

Many power devices have an internal structure like this, so usually it is not necessary to use an external diode.

1.2   Full Bridge

The full-bridge structure is composed by two half-bridge blocks connected to the load. This configuration has many advantages compared to the half-bridge structure. First of all, it does not require a complex control: in fact it is not necessary to control the voltage between the input capacitors. This makes completely free the way with which you can drive the transistors, except for the two transistors of the same half-bridge. The reason is that, if both the transistors of the same arm are on at the same time, there will be a short-circuit at the input. Another important advantage is the range of the output voltage that with this configuration is $2V_{in}$, the double of the previous case.

The main disadvantage is an higher hardware complexity of the driving systems, that must drive four different transistors. Furthermore, the losses are increased because the devices in series at the load are always two: this means that the current flows through two devices that have a non-zero voltage drop, so the power dissipation is the double of the half-bridge configuration. As you can see from the circuit schematic in Fig. 1.1, the anti-parallel diodes are always present to guarantee a reclosing way for the current. The evident advantages make this configuration often preferable to the half-bridge and, for this reason, it will be used in this project. Another motivation for choosing this solution in this study is that we use an high voltage source, so the higher voltage drop across the two transistors is not so relevant. Consequently, one of the biggest disadvantage of this configuration can be avoided.

1.3   PWM modulation

Above we have described the inverter configuration we are going to use in this project. The next step is necessary to define the type of control we want to drive the transistors.

The most common driving system for the inverters, and also for a lot of other electronic devices, is the Pulse Width Modulation (PWM). This type of modulation consists in comparing a reference signal with a modulation signal, usually a high frequency sawtooth waveform. When the reference signal is higher than the modulation signal, the PWM output is at high logical level. Conversely, if the reference signal is lower than the modulation signal the PWM output is at low logical level. The PWM switching frequency has to be much faster than what would affect the load, which is to say the device that uses the power. In this case the output main frequency is set to 50Hz. With this setting for the main frequency, the corresponding frequency of the PWM usually has a range from few kHz to tens of kHz.
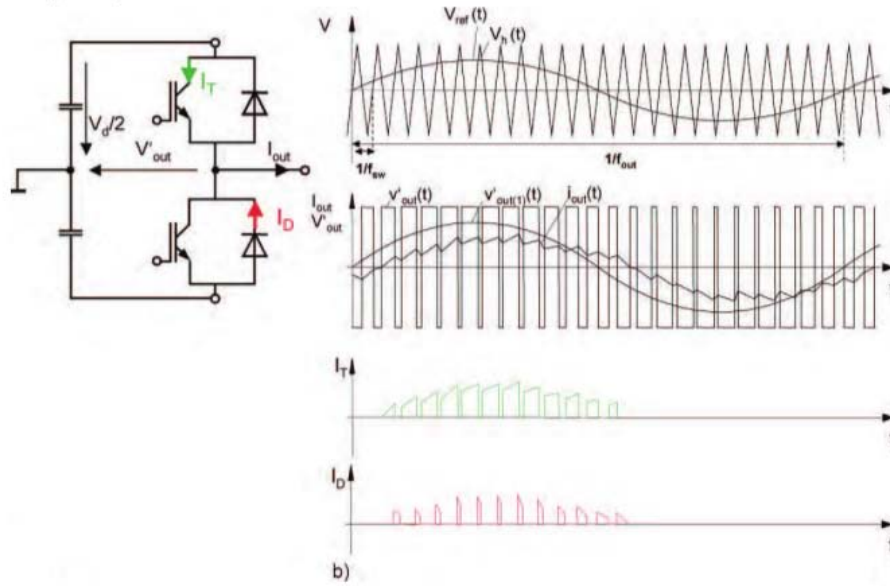
Figure 1.2: Example of a pwm modulation: On the left, the circuit representing an half-bridge. On the right, a representation of the signals involved: the modulation, the reference and the output of the PWM. [1]

This type of modulation assures to have a digital output with a fixed frequency and with a duty-cycle proportional to the ratio between the reference and the modulation signal.

Principle: The Pulse Width Modulation has as output a rectangular waveform whose duty cycle is modulated with the instantaneous value of the waveform, more precisely is modulated proportionally to the amplitude of the waveform.

Let us now take a look in a math way, considering a pulse waveform $f(t)$ with period T, low value $y(min)$ and high value $y(max)$, the mean value of the waveform is:

$$\bar{y}(x) = \frac{1}{T} \int_0^T f(t)\, dt. \tag{1.1}$$

We will try now to relate the mean value with the duty cycle D, that is defined as the percentage of one period where a signal is active. In this case, the waveform has high value (y_{max}) for $0 < t < DT$ and low value (y_{min}) for $DT < t < T$. With this new consideration, the formula becomes:

$$\begin{aligned}
\bar{y}(x) &= \frac{1}{T} \left(\int_0^{DT} f(t)\, dt + \int_{DT}^T f(t)\, dt \right) \\
&= \frac{T \cdot D \cdot y_{max} + T \cdot (1-D) \cdot y_{min}}{T} \\
&= D \cdot y_{max} + (1-D) \cdot y_{min}.
\end{aligned} \tag{1.2}$$

Considering that usually $y_{min} = 0$ because of the digital output, the expression can be simplified as $\bar{y} = D \cdot y_{max}$.

With this technique it is so possible to have any type of output signal. In fact, it is only necessary to choose an appropriate reference signal and the modulation will generate a digital signal. This signal, connected to the switching devices of our inverter, will force the output to follow the desired signal.

To characterize a type of PWM modulation, relating the reference signal with the modulation signal, there are two important parameters:

m$_f$: It is defined as the ratio between the frequency of the modulating signal (f_m) and the frequency of the carrier signal

$$m_f = \frac{f_c}{f_m}. \tag{1.3}$$

For high values of this parameter it could not be integer: in this case the PWM is called asynchronous and at the output are present some low level sub-harmonics. Vice versa, if the parameter is integer, the PWM is synchronous and the output has no sub-harmonics. This means that if this parameter if not so high it must be integer, otherwise the presence of the sub-harmonics is remarkable. In our case this is not a problem, f_m is fixed at 50Hz and f_c around 10kHz, so m_f is sizeable and it is also integer.

m_a: The modulation factor is defined as the ratio between the amplitude of the modulation signal (V_m) and the amplitude of the carrier signal (V_c)

$$m_a = \frac{V_m}{V_c} \tag{1.4}$$

If $m_a < 1$ (under-modulation) the modulation is linear and the amplitude of the fundamental changes linearly with m_a. In addition to the fundamental there are harmonics of value m_f, $2m_f$, $3m_f$ and others, centered on these. In this range, the higher the modulation factor and the longer the high side switch is on, the higher is the average load voltage. We can see this with the next formula:

$$V_{out} = m_a \cdot \frac{V_d}{2}, \tag{1.5}$$

where V_d is the input voltage of the inverter.

If $m_a > 1$ (over-modulation) the modulation is non-linear and there are more harmonic components, such that the fundamental could not be the biggest one.

Bipolar PWM: The bipolar PWM is so called because the switches 1-4 and 2-3 are driven in pair, the output in the first arm is exactly the opposite of the second arm, this means that:

$$V_{Bo}(t) = -V_{Ao}(t) \tag{1.6}$$

this means that the output voltage could be written in this way:

$$V_o(t) = V_{Bo}(t) - V_{Ao}(t) = 2V_{Ao}(t) \tag{1.7}$$

and the voltage peak of the fundamental harmonic is then:

$$V_{o1}(t) = m_a V_d, \ for \ m_a < 1 \tag{1.8}$$

1.4   Switching Devices

Commercially the switching devices you can find are a lot, and the range of values they can cover is considerable. Essentially, there are four main devices used in power electronics as switches:

Power MOSFET: These devices share with the normal MOS only the principle of operation, that is the "Field-Effect", but technologically the normal MOSFETs and the power MOSFETs are completely different devices. The power MOSFETs are able to stand current of hundreds of Amperes and voltage till over 1000V. Anyway, the power MOSFETs share with the normal MOSFETs two important features: they have the big advantage that they have no steady state losses due to the driving and they have a fast switching time. On the other hand, they are more expensive and more sensitive to the overvoltage.

IGBT: This kind of devices are done mixing the structure of a MOSFET and of a BJT and for this reason usually an IGBT is schematized like in figure 1.3. This combination of elements permits to control the input like a FET and to use a bipolar transistor as a switch. This means that this device is gate-driven so like a FET does not need a current to maintain its static condition but, like a bipolar transistor, it is capable of an high-current and a low-saturation-voltage. For this reason,

these devices, fairly recent invention, are used in the medium-high power applications, usually in a range of current till more then $1500A$ and for switching voltage up to $3000V$.
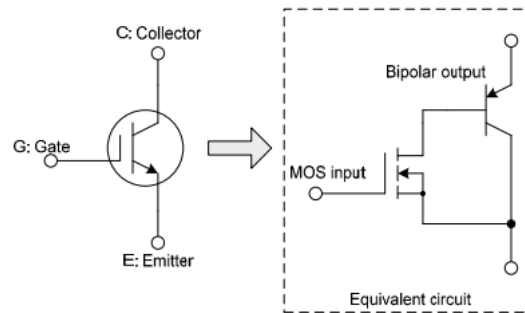
Figure 1.3: Symbol of an IGBT and equivalent circuit [2]

GTO, SCR: These components are used exclusively for very high power applications, with current till more then $3000A$ and voltage up to $7000V$. They are, respectively, controllable at the switch on and not controllable during the switch off. This means that the SCRs can not be switched off by a controlled signal, but they switch off only when the current that flows through stops by external reasons. These devices can allow high current and high voltage, but they have the big disadvantages of a long switching time. This limits the frequency they can operate. For high power applications there are no others choices, so the only thing that you could do is to limit the frequency.

In applications like in this thesis we do not need to manage very high electrical quantities, so we can use power MOSFETs or IGBTs. The IGBTs are a bit cheaper than the MOSFETs, they are less sensitive to the overvoltages and they have also a lower voltage drop during the steady state conditions. On the other hand, they have a commutation time a bit longer than the MOSFETs, anyway in our case this is not so important because the switching frequency is not very high, for all these reasons we decided to use the IGBTs.

Part I

# Theoretical Analysis

Chapter 2

Modelling of the losses

In this chapter we will talk about the losses in an IGBT and about two models for the evaluation of the losses.

2.1 General view

In power electronics, and particularly in the design of electronic converters, devices like IGBT, diodes, MOSFET and others are used as switches. The reason of this is that, ideally, a switch has no losses considering that it has either zero current or zero voltage, so the power dissipation is always zero. By the way, these devices, even if used as switches, are not ideal switches: this means that they generate power losses. These inevitable losses occur both in static and switching conditions and, furthermore, there are also power losses due to the driving of the switching devices.

The losses generated by the these devices used like switches are the most responsible of the losses in an electronic converter: they may increase the temperature till destroying completely the device. For this reason, it is very important to evaluate these losses to assure that the temperature of the junction of the device remains under the maximum junction temperature specified in the data-sheet.

The total losses in a switching device can be divided in three main categories and then in some other sub-categories. The following graph explain these differences.
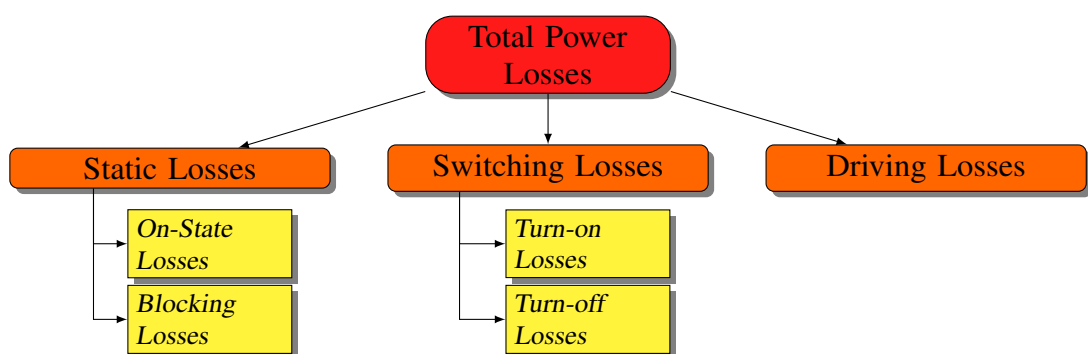
These categories have not the same importance to the losses estimation: for example, often, the driving losses are neglected since their contribution is a minimal part of the power losses. In power electronics this is reasonable since the purpose of this field of electronic is, somehow, to change the size of the units of measurement like the voltage, the current and the frequency and doing that, trying to reduce the losses as much as possible, so it makes sense that the driving losses are neglected compared to the other ones.

The other two big categories are the static losses and the switching losses or rather the losses during the conduction time interval and the losses during the commutation interval. In this thesis we use an IGBT device, so let us analize the losses specially for this kind of device.

IGBT: As discussed above, the driving losses can be neglected because they do not have a big influence in the calculation of the total losses: furthermore, also the blocking losses are usually neglected. This because when the IGBT is on, its voltage drop is not zero, but when it is off the leakage current is really close to zero and so the power during the off-state. For this reason, normally, except for some extreme applications, both the driving and the blocking losses are neglected. That is also what we are going to do in this case.

Let us now take a deeper look to the remaining categories, analysing their dependencies.

Regarding the on-state condition, an IGBT device has a behaviour similar to a bipolar device. This means that an electrical equivalent scheme of the device can be a constant voltage source – that represents the $V_{ce(sat)}$ – with, connected in series, a resistance – representing the resistivity of the material – that correlate the losses with the current. For this reason we can say that the on-state power dissipation ($P_{cond(T)}$) is dependent on:

- The load current over output characteristic $V_{CE(sat)} = f(I_c, V_{GE})$;

- The junction temperature;

- The duty cycle.

Concerning the switching losses, they are due to the not instantaneous changing of the electrical characteristics of the device: in fact, the variation is gradual and requires a delta-time to be completed, so in this time interval the power losses can be very high. For this reason, it is important that the switching device changes its state as fast as possible. Furthermore, this kind of losses restricts the maximum switching frequency: in fact, if it is too high the switching losses can be more relevant than the on-state losses and also strongly compromise the efficiency of the system. We have to say that is not easy to estimate the switching losses, because the waveform of the current and of the voltage during a commutation is very complicated and also dependent of various parameters. Anyway, at given control parameters (R_G, V_{GG}) and neglecting parasitic effect (L_s, C_{load}), turn-on and turn-off losses (P_{on}, P_{off}) are mainly dependent on:

- The load current and the electric load type (ohmic, inductive, capacitive);

- The DC-link voltage;

- The junction temperature;

- The switching frenquecy.

With this new consideration, the total losses are now composed as follows:

$$P_{tot(T)} = P_{cond(T)} + P_{on} + P_{off} \qquad (2.1)$$

Obviously, this consideration is true only for a single IGBT, if there are more than one IGBTs you have to multiply this formula for the number of the devices to have the total losses.

Freewheeling diode: Let us now analyse the losses in the freewheeling diodes. Like in the previous paragraph, where we decided to neglect the blocking losses for the IGBTs, for the same reasons the total blocking losses may be neglected also in this case.

So far, the components of the power dissipation in the freewheeling diodes are three like for the IGBTs. Nevertheless, in this case, the turn-on losses may be neglected as well: in fact the turn-on power dissipation is caused by the forward process and, as for fast diodes, this share of the losses may be neglected.

Like above, let us now analyse which are the variables that influence the on-state power dissipation. The on-state power dissipation ($P_{cond(D)}$) is dependent on:

- The load current (over output characteristic $V_f=f(If)$);

- The junction temperature;

- The duty cycle.

At given control parameters of the IGBT commutation with the diode, and neglecting parasitic effects (L_s), turn-off losses (P_{rr}) are dependent on:

- The load current;

- The DC link voltage;

- The junction temperature;

- The switching frequency.

The total losses for each freewheeling diode are composed as follows:

$$P_{\text{tot(D)}} = P_{\text{cond(D)}} + P_{\text{rr}} \tag{2.2}$$

Also in this case the previous formula concerns only a single diode, so to find out the right formula for the total losses of a module we have to sum the total losses of an IGBT and of a diode and to multiply that for the number of the switches presents in the device. The resulting formula, as follows:

$$P_{\text{tot(M)}} = n(P_{\text{tot(T)}} + P_{\text{tot(D)}}) \tag{2.3}$$

2.2 Thermal equivalent circuit

Now we will treat how we can model the heat flow using an equivalent electrical model and how to use that for our purposes. First of all, we are going to introduce the thermal resistance and the thermal capacitance:

- The thermal resistance is a property that every object has: it represents a measurement of a temperature difference by which an object resists a heat flow;

- The thermal capacitance is a measurement of the physical quantity of heat energy required to change the temperature of an object by a given amount.

The thermal equivalent circuit is used to simplify the treatment of the heat flow through a power device and its heat-sink: the heat flow is represented by current, the temperatures are represented by voltages, the heat source is represented by constant current sources, the absolute thermal resistances are represented by resistors and the thermal capacitances by capacitors.

The quantity of power dissipation P_{tot} due to the heat losses is expressed by the temperature difference between the device – usually this temperature is referred to the temperature of the junction, so it is called T_j – and the heat-sink (T_s):

$$\Delta T_{\text{j-s}} = T_j - T_s. \tag{2.4}$$

The thermal resistance between the junction and the heat-sink ($R_{\text{th(j-s)}}$) is defined as follow:

$$R_{\text{th(j-s)}} = \frac{T_j - T_s}{P_v}. \tag{2.5}$$

The thermal resistance is appropriate only during the stationary conditions, but regarding the transient condition the thermal resistance become a thermal impedance $Z_{\text{th(j-s)}}$. Often, there is not only one thermal resistance to consider: for example, if the device has a base plate in the data-sheet usually there are two thermal resistances, one between the junction and the case $R_{\text{th(j-c)}}$ and the other one between the case and the flat plate $R_{\text{th(c-s)}}$:

$$R_{\text{th(j-s)}} = R_{\text{th(j-c)}} + R_{\text{th(c-s)}}. \tag{2.6}$$

Obviously, if we have to consider also the transients, we have to take impedances and not resistances:

$$Z_{\text{th(j-s)}} = Z_{\text{th(j-c)}} + Z_{\text{th(c-s)}}. \tag{2.7}$$

In power modules with base plate, the transistor and the free-wheeling diode are soldered together on a common copper substrate, thus being thermally coupled. In figure 2.1 are shown both the models: the equivalent thermal model with and without the base plate. The base plate matches the temperature of the IGBT and the freewheeling diode. This means that, for example, only the IGBT will heat the base plate also the diode will be warmed up, even if the diode itself does not contribute to the heating up of the plate.
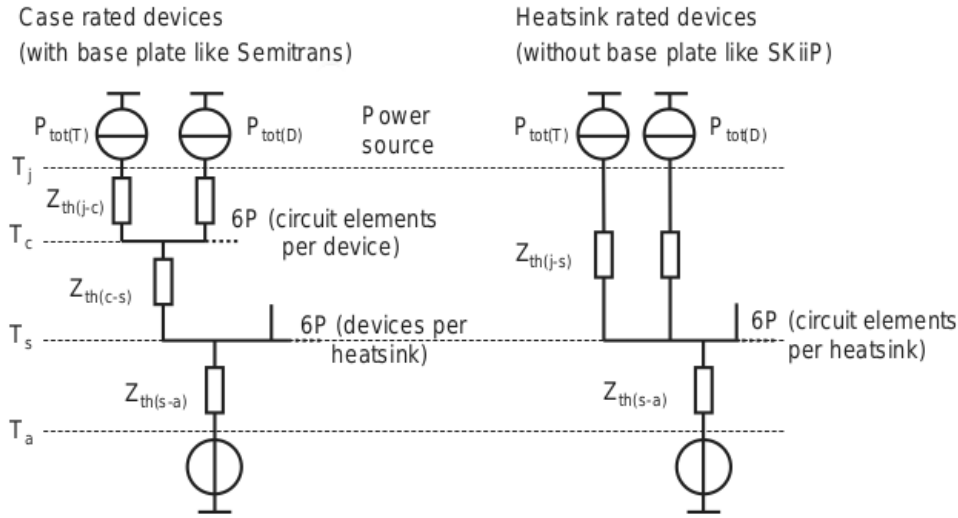
Figure 2.1: In this figure there are two simplified thermal circuit diagrams of IGBT and freewheeling diode in a power module. On the left, a model for a module with copper base plate, on the right a module without a base plate. [1]

In the figure 2.1 we may find several temperatures, called: T_j (temperature of the junction), T_c (temperature of the case), T_s (temperature of the substrate) and T_a (temperature of the ambient). As we can see on the right picture there is not a case temperature, in fact it has not a base plate so, for the left picture the total thermal impedance will be:

$$Z_{th(j\text{-}a)} = Z_{th(j\text{-}c)} + Z_{th(c\text{-}s)} + Z_{th(s\text{-}a)}; \tag{2.8}$$

vice versa, on the right picture the case does not exist, so the formula is:

$$Z_{th(j\text{-}a)} = Z_{th(j\text{-}s)} + Z_{th(s\text{-}a)}. \tag{2.9}$$

To obtain the chain-type equivalent circuit diagram, the factor time constants for the exponential function are determined by means of formula manipulation.

$$\begin{aligned}
Z_{th(x\text{-}y)} &= R_{th1}\left(1 - \exp\frac{-t}{\tau_{th1}}\right) + R_{th2}\left(1 - \exp\frac{-t}{\tau_{th2}}\right) + ... \\
&= \sum_{v=1}^{n} R_{thv}\left(1 - \exp\frac{-t}{\tau_{thv}}\right)
\end{aligned} \tag{2.10}$$

The parameters R_{thv} are usually specified in the data sheets of the devices, like the parameters τ_{thv}: so it is possible to use the formula above and estimate correctly the temperature of the junction and choose the correct heat sink.

Junction temperature: To evaluate the temperature of the junction it is first necessary to calculate the losses of the device: after this is possible, having all the thermal impedances of the heating system and the ambient temperature, to find out the temperature of the junction. This, obviously, is necessary to check if the specified device could work in that condition (power losses/heat sink) or not. To calculate correctly the temperature of the junction considering the evolution of the system with time it is necessary to use thermal impedances. Otherwise, with thermal resistances we can calculate the temperature under stationary operation conditions: this means that we have also to use stationary characteristics of the system. The order in which you have to proceed in this calculation is "from the outside to the inside", starting with the ambient temperature and the thermal impedance of the heat-sink and then proceeding step by step until reaching the junction temperature. The formula used for each step is: $T_i = T_o + P_v \sum Z_{th}$, where T_i is the inner temperature and T_o the outer temperature.

The scheme 2.2 is an example of the procedure to follow to calculate the temperature of the junction of our device.
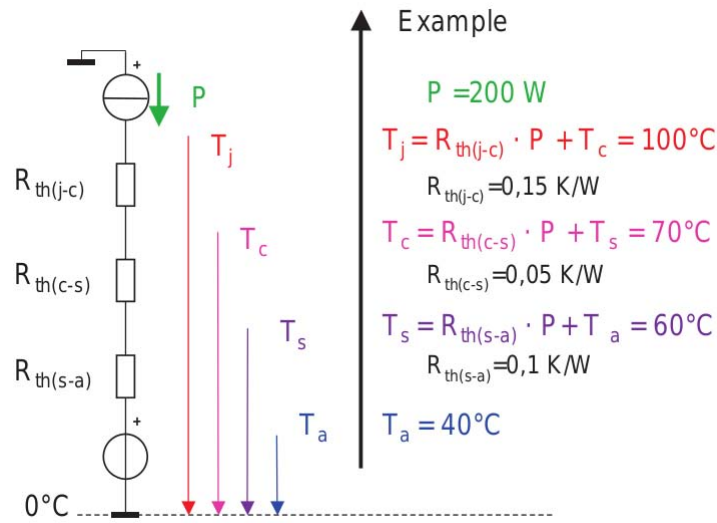
Figure 2.2: Calculation of the junction temperature under stationary condition [1]

Actually this scheme is very simple and the losses source is only one, but if there is more than one source of losses, for example in a power switching device with IGBTs and freewheeling diodes, the formula changes in this way:

$$T_{\text{i}} = T_{\text{o}} + (P_{\text{tot(T)}} + P_{\text{tot(D)}}) \sum Z_{\text{th}}, \tag{2.11}$$

where $P_{\text{tot(D)}}$ and $P_{\text{tot(T)}}$ are, respectively, the power losses of the diode and of the transistor. In case of the "couples" transistor/diode should be more than one the above formula will change in accord to the contribution of every couple:

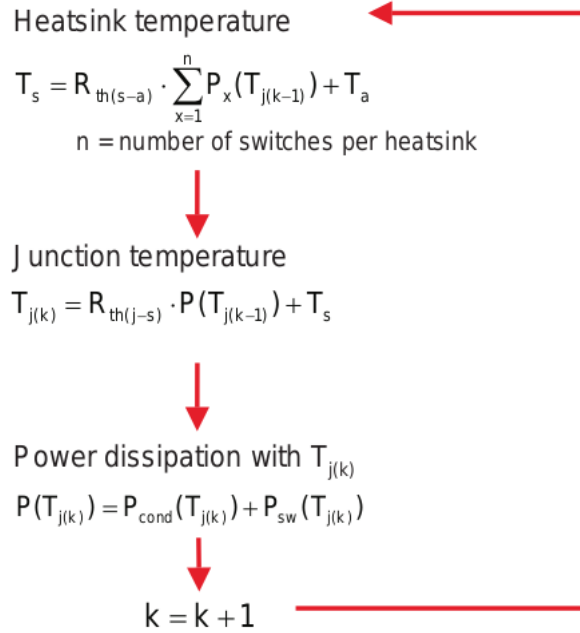$$T_{\text{i}} = T_{\text{o}} + n_1 (P_{\text{tot(T)}} + P_{\text{tot(D)}}) \sum Z_{\text{th}}. \tag{2.12}$$

The semiconductor has a strong matching between the temperature and the losses, this means that the temperature influences the characteristics of the semiconductor. Considering this, it is not trivial to calculate the losses: the easiest way is to calculate then considering the temperature of the junction at the maximum temperature. This ensures to take a conservative amount of the losses because the most of these increase with the temperature. A more accurate way to calculate the losses is taking account the temperature to determinate the losses at a starting temperature (for example the ambient temperature) and then iteratively calculate the temperature and the losses until they will reach a stable value. This loop can be easily solved by a computer and allows us to estimate more accurately the final temperature and the losses of our device. The figure 2.3 explain in a graphic way how this technique works.

2.3 Average estimation of the losses

We will now analyse the losses using the average characteristics of the system. Anyway, to calculate correctly the losses we have to consider which kind of modulation we are going to use in our converter, that is PWM, already explained previously. In figure 2.4 you can see the half-bridge configuration with some graphics that explain how the PWM works compared with the output quantity: in particular, you can see that the output current is not exactly in phase with the output voltage and thus – considering the output current positive – the current flows in the IGBT when it is on and, when the IGBT is off, it flows in the opposite direction through the diode.

The configuration in the figure is an half-bridge, whereas we are going to use a full bridge, but the other two switches (IGBT and relative diode) are driven complementary, so it is enough to consider an half bridge to estimate the losses. Furthermore, the two IGBTs of each half-bridge are on respectively only in the positive or negative semi-sine, this means that we can analyse the losses only for one switching device and then extend our considerations to the other three switches.

Even if we are considering an average model of the losses, the temperature of the junction is constant only when the steady-state condition is reached: instead, during the transient it depends on different

Heatsink temperature

$$T_s = R_{th(s-a)} \cdot \sum_{x=1}^{n} P_x(T_{j(k-1)}) + T_a$$

n = number of switches per heatsink

Junction temperature

$$T_{j(k)} = R_{th(j-s)} \cdot P(T_{j(k-1)}) + T_s$$

Power dissipation with $T_{j(k)}$

$$P(T_{j(k)}) = P_{cond}(T_{j(k)}) + P_{sw}(T_{j(k)})$$

$$k = k + 1$$

Figure 2.3: Loop used to estimate the temperature and the losses in a semiconductors [1]

parameters, principally the AC output frequency, because it establishes how long each IGBT has to be in on-condition continuously. This means that a lot of parameters vary in time and could be difficult to estimate every single variation of these parameters. So we are going to define some simplification to allow us to calculate the instantaneous losses without considering useless parameters but, anyway, having an acceptable estimation. The following simplification are assumed:

- Transistor and diode switching times are neglected;

- Junction temperatures are in time constant;

- Linear modulation ($m \leq 1$);

- The switching frequency ripple of the AC current (sinusoidal current) is neglected;

- $f_{sw} \gg f_{out}$.

We suppose that V_{inv}, the output voltage of the inverter, correspond to V_o, the voltage across the load (that in our case is supposed to be a machine). This means that the output impedance has a low drop at the given frequency. The following expression represents the voltage across the load:

$$V_o(t) = \hat{V}_o \sin(\omega t) \implies V_o(\theta) = \hat{V}_o \sin(\theta). \tag{2.13}$$

Consequently, the average output voltage and the average voltage across the load can be forced to be the same. We can so deduce the expression for the duty cycle related to the V_o:

$$\overline{V}_{INV} = V_{DC}D(\theta) - V_{DC}(1 - D(\theta)) = V_{DC}(2D(\theta) - 1) \overset{!}{=} V_o(\theta) \tag{2.14}$$

$$D(\theta) = \frac{V_o(\theta)}{2V_{DC}} + \frac{1}{2} = \frac{V_o(\theta) + V_{DC}}{2V_{DC}} \tag{2.15}$$

Using this expression of the duty cycle, we can proceed to calculate the average current that flows through the IGBT and consequently the power dissipation. In fact, the average current can be calculated as follows:

$$\overline{I}_{S1} = D(\theta)\overline{I}_L(\theta) = \frac{V_0(\theta) + V_{DC}}{2V_{DC}} \hat{I}_L \sin(\theta + \phi) \tag{2.16}$$
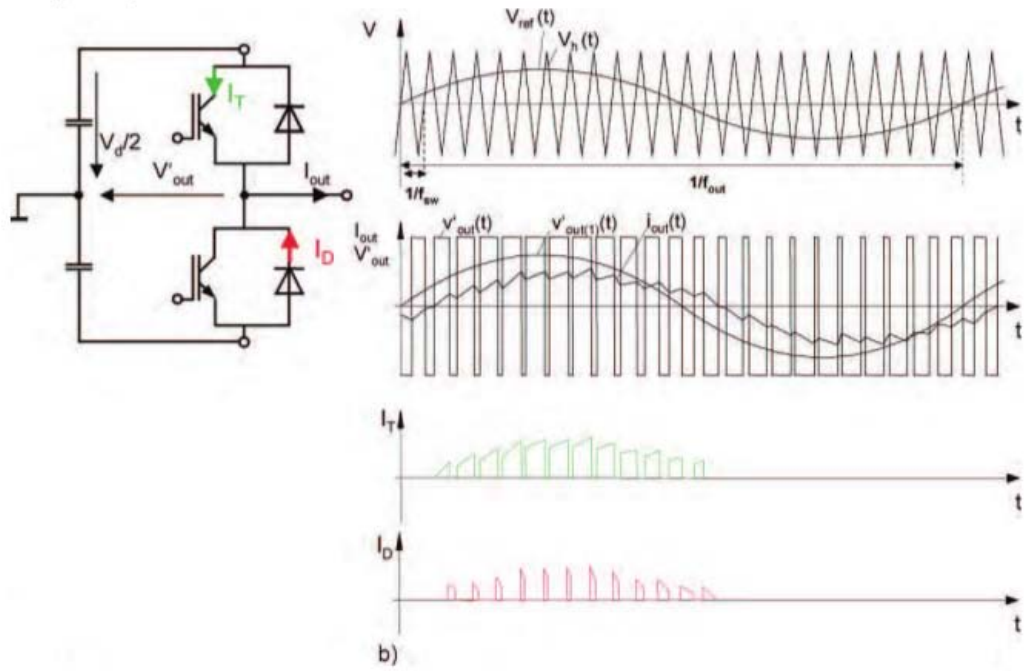
Figure 2.4: Half-bridge configuration with the driving and output waveform [1]

This equation represents the average value of the current that flows through the IGBT S1 and S2 in the first half period of each sine, so from an angle 0 to π. In the second half period, in fact, these two IGBTs are off because the current is negative. This calculation is, in any case, a simplification because we have considered that if the average current is positive only the IGBT S1 and S2 can be on. This is not true , in fact, if the average current is close to zero, the instantaneous current can be negative even if the average is positive. Anyway this condition happen only for a limited time interval, so it is reasonable to neglect it.

According to the previous formula the power losses of one IGBT is:

$$P_{S1}(\theta) = V_{on}\overline{I}_{S1}(\theta) + R_{on}I^2_{S1RMS}(\theta) \tag{2.17}$$

Where $\hat{I}_{S1RMS}(\theta)$ it is the RMS current that flows through the IGBT in one switching period. Consequently the power losses can be calculated like:

$$P_{S1} = \frac{1}{2\pi} \int_0^{2\pi} P_{S1}(\theta)\, d\theta = \frac{1}{2\pi} \int_0^{\pi} P_{S1}(\theta)\, d\theta \tag{2.18}$$

The $I_{S1RMS}(\theta)$ is related by the current that flows through the load by the formula:

$$I_{S1RMS}(\theta) = \sqrt{D}I_{LRMS}(\theta) \tag{2.19}$$

In turn, the RMS current of the load can be decomposed as follows:

$$I_{LRMS}(\theta) = \sqrt{I_L(\theta) + \frac{1}{12} * \Delta I^2_{LPP}(\theta)} \tag{2.20}$$

The $\Delta I_{LPP}(\theta)$ is the contribution of the ripple component and can be calculated as:

$$\Delta I_{LPP}(\theta) = \frac{V_{DC}{}^2 - V_0{}^2}{2Lf_{sw}V_{DC}}. \tag{2.21}$$

Normally this contribution represent a negligible part of the current.

According with the formulas we have just mentioned, and considering when the current can flow through each device, it is possible to estimate the average losses of the IGBTs and the diodes.

2.4 Instantaneous estimation of the losses

It is possible to realize a more accurate model considering the instantaneous values of the characteristics of our inverter. Particularly, during the on-state condition the output voltage – and consequently the output current – change a lot during each sine, this means that also the temperature of the junction could have a purposeful variation during a period that can not be observed with the previous model. For this reason is necessary to realize another model to check that this temperature rise does not exceed, also for a small time inteval, the temperature limit imposed by the datasheet.

In this model the on-state losses will be calculated just multiplying the instantaneous values of the current with the associated value of the V_{ce} across the IGBT and similarly, using the V_f, for the diode. The V_{ce} and the V_f are parameters strongly related with the junction's temperature of the device and with the current that flows through it. For this reason it is necessary to realize a logic scheme to understand when actually each device is active in each moment.

Below, we can find the half-bridge schematic of the switching-device and the relative logic scheme that explains how each device switches related to the sign of the output current and the PWM signal. This logic scheme is shown in table below on the left and, on the right, the circuit diagram:

Device	I_{OUT}	PWM_1	PWM_2
$IGBT_1$	> 0	1	/
$DIODE_1$	< 0	/	0
$IGBT_2$	< 0	/	1
$DIODE_2$	> 0	0	/
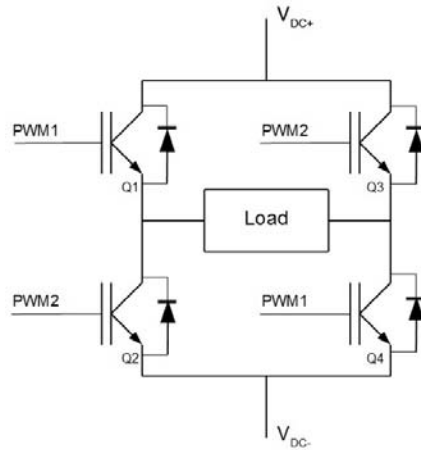
Regarding the switching losses, as previously explained, it is difficult to estimate them correctly, but we can consider the energy needed to change the logic state and multiply it for the switching frequency to obtain the power losses. Anyway, the switching energy losses are not a constant value, but change with the temperature, the current and the voltage across the device. So, it is necessary to take into account this. By the way, using the datasheet it is possible to extrapolate these data and calculate properly the correct value of the energy. The formulas we are going to use are these:

$$P_{sw(I)} = f_{sw} * E_{(on+off)}(T_j, I_{out}, V_{cc}) \tag{2.22}$$

$$P_{sw(D)} = f_{sw} * E_{rr}(T_j, I_{out}, V_{cc}) \tag{2.23}$$

Chapter 3

Calculation of the losses

We have just analysed two ways to estimate the losses in a switching device, now we will see how to do this calculation and the relative results.

First of all we have to choose the switching device we want to use in this project: to do this we have to fix some main parameters of the project, like:

- DC input voltage

- Output power

- Switching frequency

We have decided to design an inverter with an input voltage of about 300V, an output power of about 9kW and a switching frequency of 10kHz. The output voltage will be, according to the full-bridge structure, at most 300V with a maximum output current, in any case related to the others characteristics, around 30A.

Once fixed this parameter, is possible to choose the switching device: in this case, we choose the **INFINEON F4-30R06W1E3**. It has four IGBTs connected internally in a full-bridge configuration and each IGBT has his own freewheeling diode. Furthermore, there is also a NTC temperature sensor to check the real temperature of the device so, in case, it is possible to switch-off the converter preventing a failure of it. According to the data-sheet the IGBTs can tolerate a *"Nominal current (in DC condition)"* of 30A with a *"Repetitive peak current"* of 60A and the *"Maximum V_{ce} voltage"* allowed is 600V. The characteristics of the freewheeling diodes are almost the same of the IGBTs, they can tolerate the same current, but the *"Maximum repetitive reverse voltage"* is 600V.

3.1 Average calculation of the losses

Now we are going to calculate the losses using the model explained in the section 2.3 and to make this calculation we used SIMULINK®. First of all, we speak about the temperature model. It is modelled using the datasheet: in fact, for each IGBT and diode there are four different thermal resistances from the temperature of the junction till the base plate, each one with its own value of thermal resistance and thermal capacitance. The values are reported as follows:

	IGBT			
i:	1	2	3	4
$r_i[K/W]$	0,126	0,274	0,637	0,514
$\tau_i[s]$	0,0005	0,005	0,05	0,2

	Diode			
i:	1	2	3	4
$r_i[K/W]$	0,233	0,398	0,719	0,5
$\tau_i[s]$	0,0005	0,005	0,05	0,2

These reported above are the thermal impedances of the switching devices, but to our model we have to add also the thermal impedance of the heat-sink. We fixed the value of it to $0.5K/W$, with a τ of 0.1s. We have to say that, in this case, the parameter τ is not fundamental because the important thing is the maximum temperature reached by the junction and not in how much time. For this reason it can be changed to make the simulation faster.

To calculate the losses, we used the formulas in section 2.3 and to get the correct value of the parameters we used some look-up tables, extrapolating the data from the data-sheet. We used these tables for the parameters:

- $V_{ce0}(T_J)$

- $r_{ce0}(T_J)$
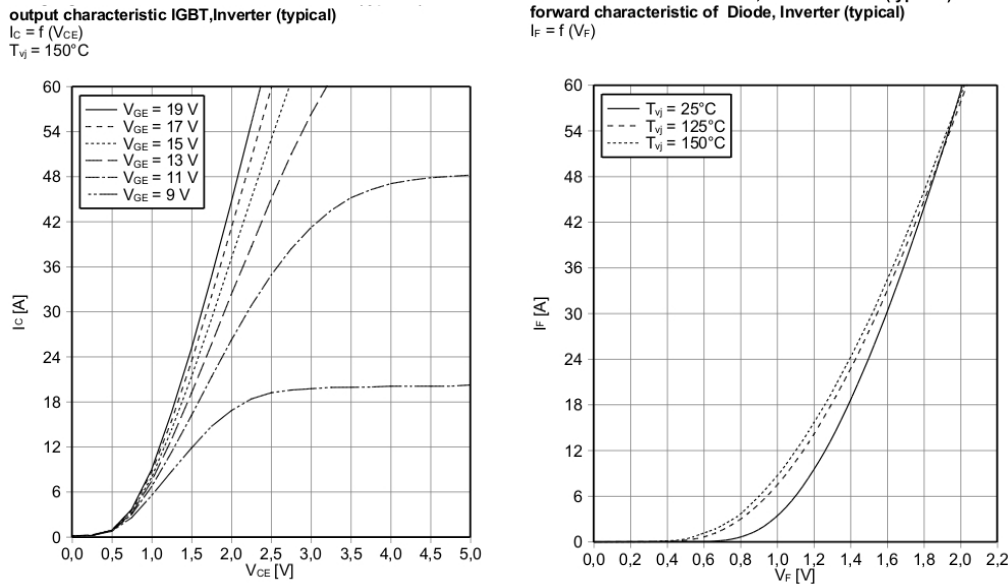
- $V_{f0}(T_J)$

- $r_{f0}(T_J)$

Figure 3.1: Characteristic of the switching device. [3]

3.2 Instantaneous calculation of the losses

The instantaneous model scheme has two main blocks: one block for the calculation of the losses and one for the thermal model. The scheme is reported in the graph 3.2. In particular, regarding the instantaneous calculation of the on-state losses, we used the same graphs above to extrapolate the data, but considering that, in this case, the V_{ce} change instantaneously and the formula is simpler, because we do not need to integrate the losses in one complete period.

Instead, concerning the switching losses we extrapolate from the data-sheet how the switching energy change with the temperature and the current. We have to say that the curves are mapped only for a temperature of $125°C$ and $150°C$, the typical operating point, and there is only a value for the energy losses at the ambient temperature, $25°C$, so we interpolate these curves with that point trying to find a correct value of the energy losses also during the transient or for a operating point with a lower temperature. Unfortunately these curves are not mapped for different values of the V_{ce} so it is not possible to build-up a look-up table for this parameter. Then, we used the same parameter K_v used in the other model.

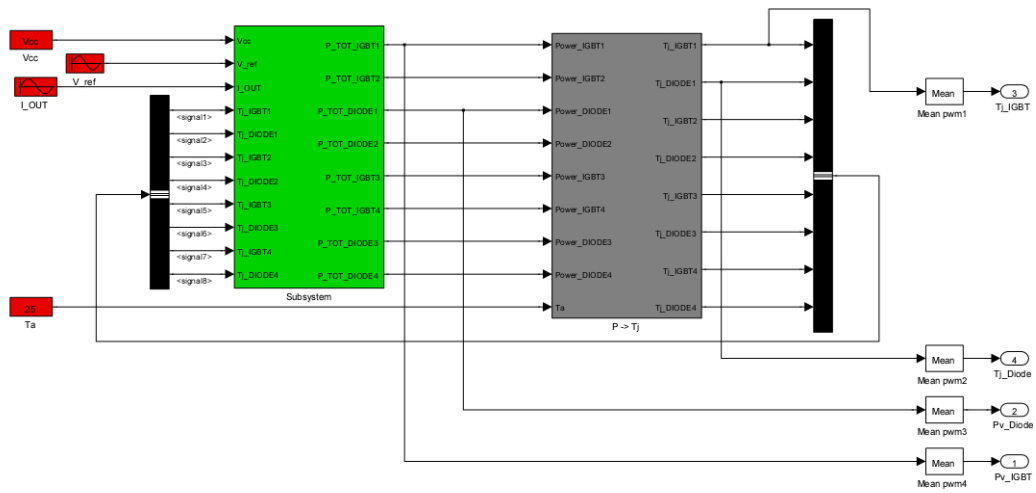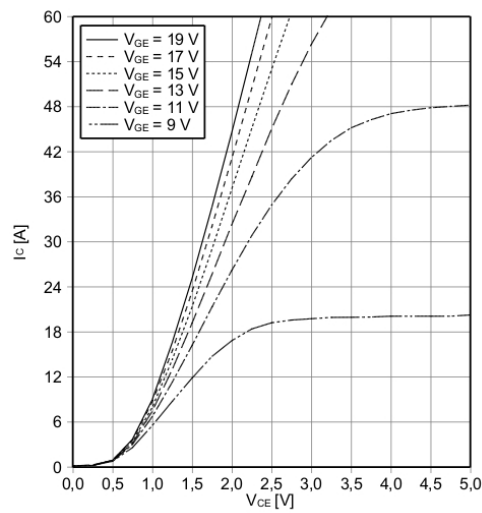Here there are the graphs that relate how the energy change with the temperature and the current:

Figure 3.2: Global view of the instantaneous losses model implemented with SIMULINK®
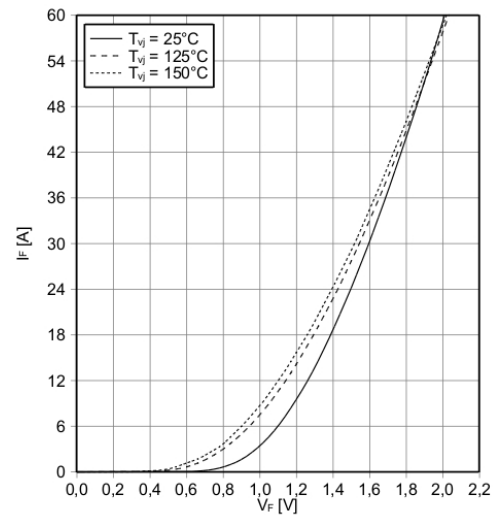

Figure 3.3: Characteristic of the switching device. [3]

3.3 Simulations

In this section we will compare the various models, and we will discuss the results. In addition to the two models realised we are going to compare the results with another model. In fact in the INFINEON website there is a simulation tool with which you can simulate the temperature of the junction and the power losses, just choosing your device and some main parameters like the switching frequency, the input DC voltage, ecc. Anyway, we can not choose which kind of output we would like to have, so it is possible to compare this model only for a limited type of data.
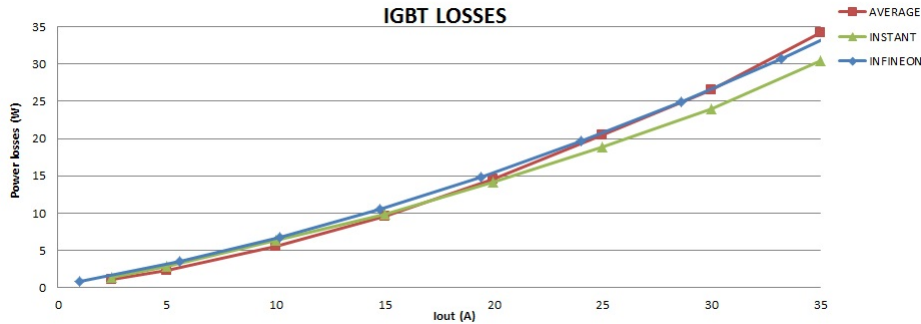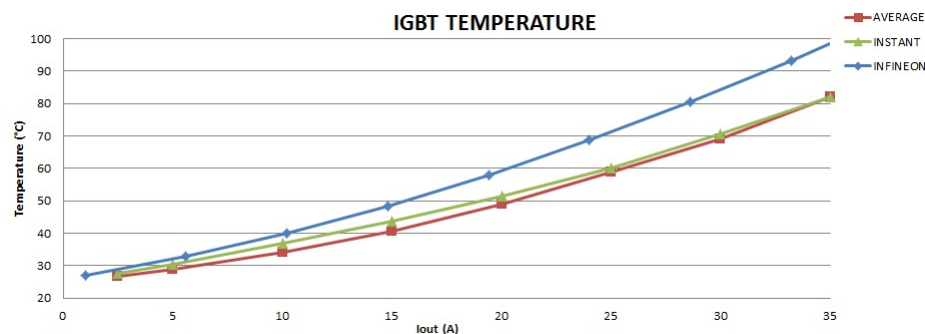

Figure 3.4: IGBT's power losses function of the current
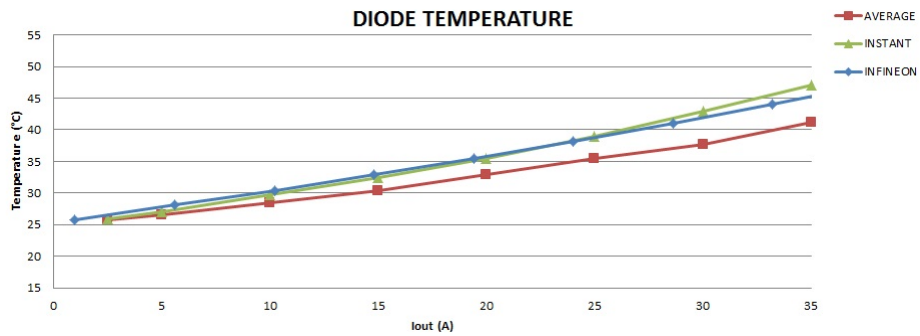Fixed parameters: $m = 0.7, V_{DC} = 300\text{V}, cos(\varphi) = 0.9, T_a = 25C$

Figure 3.5: Diode's power losses function of the current
Fixed parameters: $m = 0.7, V_{DC} = 300\text{V}, cos(\varphi) = 0.9, T_a = 25C$

Figure 3.6: IGBT's temperature function of the current
Fixed parameters: $m = 0.7, V_{DC} = 300\text{V}, cos(\varphi) = 0.9, T_a = 25C$

Figure 3.7: Diode's temperature function of the current
Fixed parameters: $m = 0.7, V_{DC} = 300\text{V}, cos(\varphi) = 0.9, T_a = 25C$

As we can see in the graph above the three models give similar results, except for the temperature of the IGBT. In fact the temperature of the INFINEON model is significantly higher than in the other two models. This because the INFINEON model considers, for the estimation of temperature, the maximum temperature reached. Instead, in our models, the temperature in each point is calculated, also in the instantaneous one, as the average value in one period. In this way it is possible to compare the results of the two models we realized.

Anyway, it is possible to compare the temperature of the instantaneous model and of the INFINEON model printing a graph with the temperature's evolution in one period, once the steady state condition is reached. The graphs that follow represent the comparison of the temperature's evolution between the simulink model and the INFINEON model at the nominal conditions ($V_{in} = 300\text{V}$, $I_{out} = 30\text{A}$, $m = 1$). In this graphs we can see that the temperature's trend of the IGBT and of the diode are really similar for both models, and also the average temperature of the IGBT is similar. The average IGBT temperature of the simulink model is around $75°\text{C}$ and around $70°\text{C}$ for the INFINEON model, instead, regarding the diode, also if the trend is very similar the average temperature is a bit more different, with a value around $43°\text{C}$ for the simulink model and around $35°\text{C}$ for the INFINEON model.
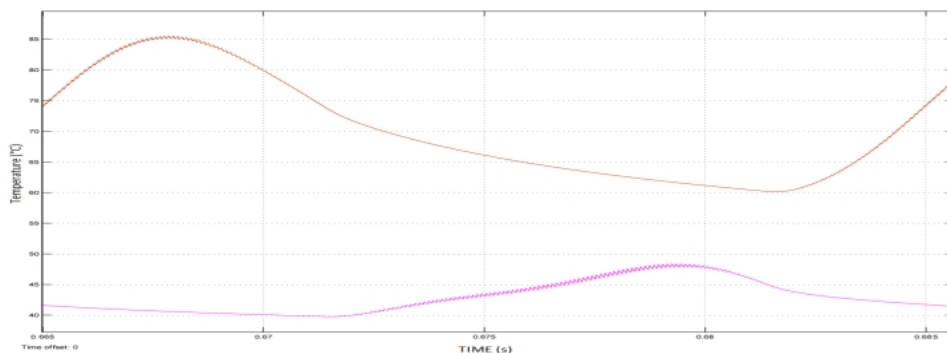
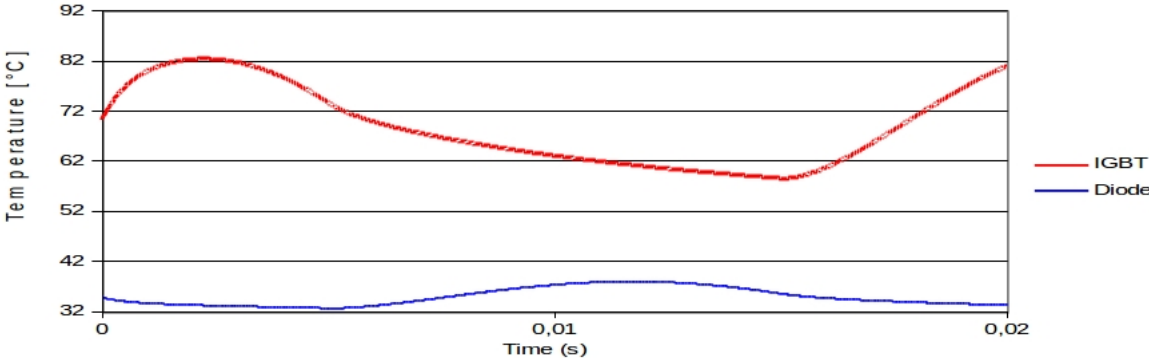**Figure 3.8:** Instantaneous trend of the IGBT and diode's temperature during one period using the SIMULINK model

Figure 3.9: Instantaneous trend of IGBT and diode's temperature during one period using the INFINEON model

Part II

Practical Analysis

Chapter 4

Choosing the components

As said previously, the switching device we are going to use is an **INFINEON F4-30R06W1E3**, so now we need to choose the other components to realise our inverter. Usually, to drive an inverter, there are two possibilities: use a microcontroller or an external PWM generator. In the second case the inverter will not be completely independent, so we decided to use a microcontroller.

4.1 The Microcontroller

Commercially, there are many types of microcontrollers for every kind of application, in particular some of these have also outputs designed specifically to generate a PWM signal. For this reason, it is a good idea to choose a component with this characteristic. Furthermore, the clock frequency is important, in fact our inverter works with a switching frequency of $10kHz$, so the microcontroller must work faster than this frequency to assure that every process will be completed in, at most, $100\mu s$. This actually is not a problem: in fact, commercially, the clock frequency of the microcontroller is usually higher that $1MHz$, often many hundreds. For these reasons, it is not difficult to find a proper microcontroller.

We chose a microcontroller produced by *Texas Instrument* called **TMS320F28335**. It has twelve different outputs suitable to work with a PWM signal: furthermore, the clock frequency is $150MHz$, more than what we need for this project. Anyway, this project, besides the PWM pins, requires some input/output pins that this microcontroller has. In particular, using the inputs it is possible also to connect some sensors: for example, to check the output current or the temperature of the switching device. It is also possible to use these pins, as we can see later, to simplify the testing of the board or to disable quickly the inverter in case of necessity. The microcontroller we have used is mounted on a board "ready to use", where every useful pin should be connected – means a connector – to the other devices, and all the other pins are managed by the board.

Regarding the software it is possible to program this kind of microcontroller using SIMULINK and then, through MATLAB, to generate the *C-code* for a specific microcontroller. Once the *C-code* is generated it is finally possible to load it in the microcontroller or to launch a debug session.
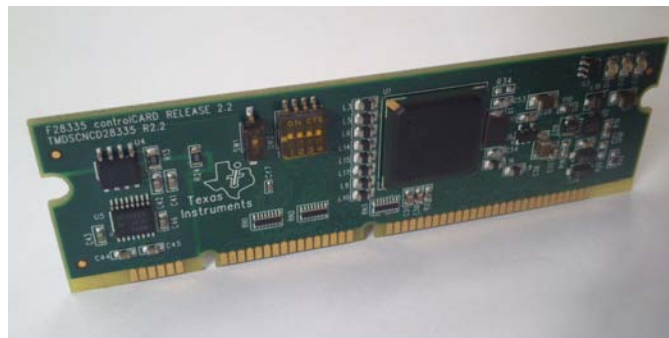
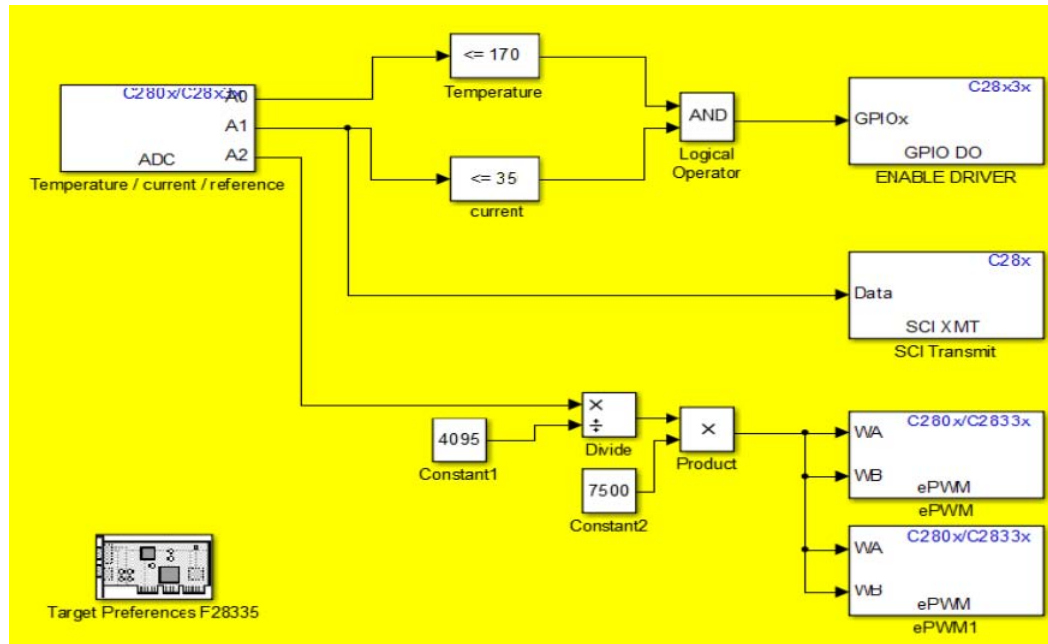**Figure 4.1:** Board of the microcontroller TMS320F28335

Figure 4.2: Scheme of the software

Software

In figure 4.2 there is the SIMULINK scheme of the software, as we can see it appears very simple: on the bottom left there is a block called "Target preferences F28335", used to set some parameters related to the microcontroller and to allow the compiler to generate the code for the specific microcontroller.

On the top left, there is a block that represents the analog inputs. In this case we have used three analog inputs: one for the temperature of the switching device, one for the current and one for the reference signal. The first two signals are connected to two comparators and then to an "*and*" operator to check if the temperature and the current are both under their respective limits. The output of this simple logic is a digital output of the microcontroller. This output is physically connected to the enable pin of the two drivers. This is a simple way to protect the inverter, in fact, if the temperature or the current is too high the inverter will be switched off till this two parameters will return under their limits.

Under the block to enable the drivers there is a block for the data transmission. In this case we decided to transmit only the value of the current, but it is possible to transmit also the temperature or other internal parameters.

The last part of the scheme represents the generation of the PWM signals. The reference signal from the input is normalized from a maximum value of 4095 to 7500, as required by the following blocks, and then connected to the two blocks that generate the PWM signals. These two blocks will generate a PWM signal comparing the reference input signal with an internal sawtooth we have conveniently set. The relationship between the four output signals can be set as we want. Particularly, in this case we set the output to work as two couples with opposite phase and with a proper dead-time.

We have to say that the external reference signal can be easily replaced by an internal signal generator. This block – during the compilation – will be replaced with a data-vector and then saved in the memory of the microcontroller. We realized also a software with an internal sine reference and it works without any problems, but for practical reasons we have used the external reference signal. In fact, with an external reference, it is possible to change easily the parameters of the reference signal like amplitude, offset or frequency; otherwise, it is necessary to modify the software and to recompile it, wasting a lot of time during the testing of the board.

4.2   Driver

As written previously, the microcontroller generates the PWM signals we need to drive our inverter, anyway there is an electrical incompatibility. Indeed, the output of the microcontroller has a maximum voltage of 3.3V and a current of few milliampere. The input of the switching device needs, instead, an input voltage around 15V with a peak current that should be some amperes. For this reason, it is

necessary to use a device that smooths out these differences. This kind of devices are called drivers because their role is to drive the switching device, when an appropriate input signal is supplied.

In this project we decided to use a *dual IGBT driver IC* produced by INFINEON, called **2ED020I12-FI**. This device, as we can understand from the name, can drive two IGBTs, in particular two IGBTs of the same half-bridge. In fact it has a special protection to prevent short circuits. If both the input signals to drive the two IGBTs are on, the output will be disabled till when this situation will be changed. In addition to this special characteristic the driver can tolerate an output voltage of ± 1.2kV, an output current of $+1/-2$A and a power supply from 14 to $18V$.

The IGBTs need two different voltages – one positive and one negative – to assure a fast switching. This driver has not a double input power supply: anyway, using a trick it is possible to realise this configuration. The two outputs of a driver are in fact totally insulated because the emitters of the two IGBTs are connected to two different points of the circuit. Using this characteristic it is possible to realise the configuration in figure 4.3 to obtain two driving voltage.
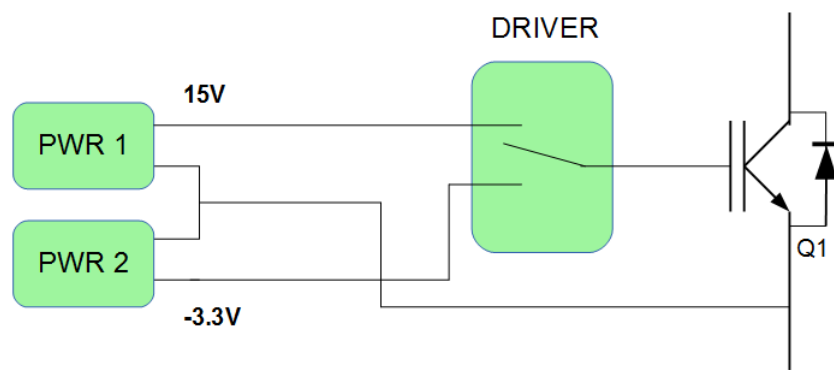

Figure 4.3: Switching management of one IGBT

4.3   Power supply

As explained in the previous section, it is necessary to use an external power supply for the drivers. Each IGBT must be driven with two different voltages: one positive to switch-on the IGBT and one negative to switch it off. The positive voltage usually is around 15V and must be higher then a minimal value to guarantee a correct switch-on of the IGBT. The negative voltage instead is not strictly necessary because to switch-off an IGBT is sufficient to short-circuit the gate with the emitter, removing the charge in the gate-body capacitor. Anyway, to guarantee a fast switch-off of the device – necessary to reduce the losses – usually the IGBTs are switched-off with a negative voltage. In this project we fixed the positive voltage to 15V and the negative to -3.3V. Remembering the full-bridge configuration we can easily see that every IGBT is connected to a different point of the circuit. This means that to guarantee a galvanic isolation it is necessary to use a different power supply for each IGBT, or rather two: one for the switch-on and one for the switch-off. Considering this, we need eight different power supplies to drive correctly our IGBTs.

Every IGBT has the same behaviour and we are also going to use the same circuit configuration for each one, so it is sufficient just to design the power supply for one IGBT and then to use the same configuration for the other switches.

We proceed calculating the energy required to switch one IGBT. An IGBT can be considered as a capacitor, and the value of this capacitor is reported in the datasheet: in particular, for this IGBT the value of the equivalent capacitor is estimated to be 0.3μF. Let us now calculate the power required to switch one IGBT:

$$E = CV^2 = 0.3\mu\text{F}(15\text{V} - (-3.3\text{V}))^2 = 0.3\mu\text{F}(18.3\text{V}))^2 = 0.1\text{mJ} \tag{4.1}$$

$$P = E * f_{sw} = 0.1\text{mJ} * 10\text{kHz} = 1\text{W} \tag{4.2}$$

The power we found is the average value of the power required in a period, but the peak value can be much higher then 1W: for this reason we put a big capacitor (of 10μF) at the output of the converters.

Now, knowing the value our converters must be able to supply we choose two converters produced by TRACO POWER, called TME0515S and TME0503S, respectively to generate 15V and $-3,3\text{V}$.

4.4   Current sensor

We decided also to install a current sensor for two reasons. First of all to prevent a damage of the device: for example, if the load is too small and so the peak current if too high, it is possible to disable instantaneously the IGBT preventing a failure of the device. The second reason is because with this component it is possible to control the inverter with a current loop, improving the behaviour of our inverter in every condition, specially when any parameter change over time.

We decided to use a *LEM CASR 25-NP*. It has a current measuring range between -85 to 85A, it needs a power supply of 5V and an output voltage from $0,375\text{V}$ to $4,625\text{V}$ with a reference voltage of $2,5\text{V}$ at 0A.

Chapter 5

Designing the board

We have just discussed the components we are going to use to realise our inverter. In this chapter we will talk about how the board was designed, initially talking about the electrical schematic and then the printed circuit (PCB).

5.1   Electrical schematic

To realise the schematic circuit we used the software EAGLE. A picture of the electrical schematic of our project is reported in picture 5.1. On the left side there are 3 connectors: The one on the top is a connector for a serial protocol communication system that can be used to pass some information to an external device; under this there are another two connectors: one for the 5V power supply and another one for the DC voltage supply of the inverter.

Between these connectors there is a circuit with a diode used to easily understand when then logical part of the board is powered. As we can note the ground signals of the connector for the communication system and the for the 5V power supply are called differently. This because on one hand the digital signals are very fast and they can influence the values of the analog signals changing over time the value of the ground and, on the other one, the current flow of the analog current can make the voltage level of the ground different around the board and this can compromise the digital communications.

The big connector called *TI_DSP* is the connector of the microcontroller. There are connected the PWM signals, the power supply, others analog inputs and some other signals necessary to program the microcontroller. These last signals are connected to another connector reported under it, that is necessary to connect the programmer. In this case, like in some other part of the schematic, the connections are not made by a wire, but using some labels to simplify the view of the schematic. On the right of the microcontroller's connector there is another small one used to connect some external signals to the microcontroller. These extra signals can be useful during the test of the board, for example to use an external reference signal for the PWM, as we did.

On the right of all these connectors there are some blocks that represent the 2 drivers (the bigger ones) and the relative power supply. The drivers have in input the PWM signals and a signal to enable/disable the drivers and so the inverter. At the output there are the signals to drive the IGBTs. In addition there are the various power supply for the drivers, provided in input with 5V and in output with the $+15/-3.3$V of the *DC/DC converters*, represented by the four small blocks over and above each driver.

The output signals of the drivers are connected by some resistances to the switching block. The resistance are three connected in series for each IGBT, this because the values of the resistances are commercially standardized, so in this way is easier to have the exact value of resistance we want.

On the right side of the switching device there is the AC output of the inverter. One of the two output wires is connected directly to a connector and the other one passes through the sensor current and then to the connector on the right side of the schematic.

5.2   Printed circuit

In this project there are different kinds of signals (analog, digital and power) and every one has different necessity: for this reason we used a four layer board. Two layers are completely dedicated to the power supply and this means that: the 5V supply for the microcontroller, the DC input of the inverter
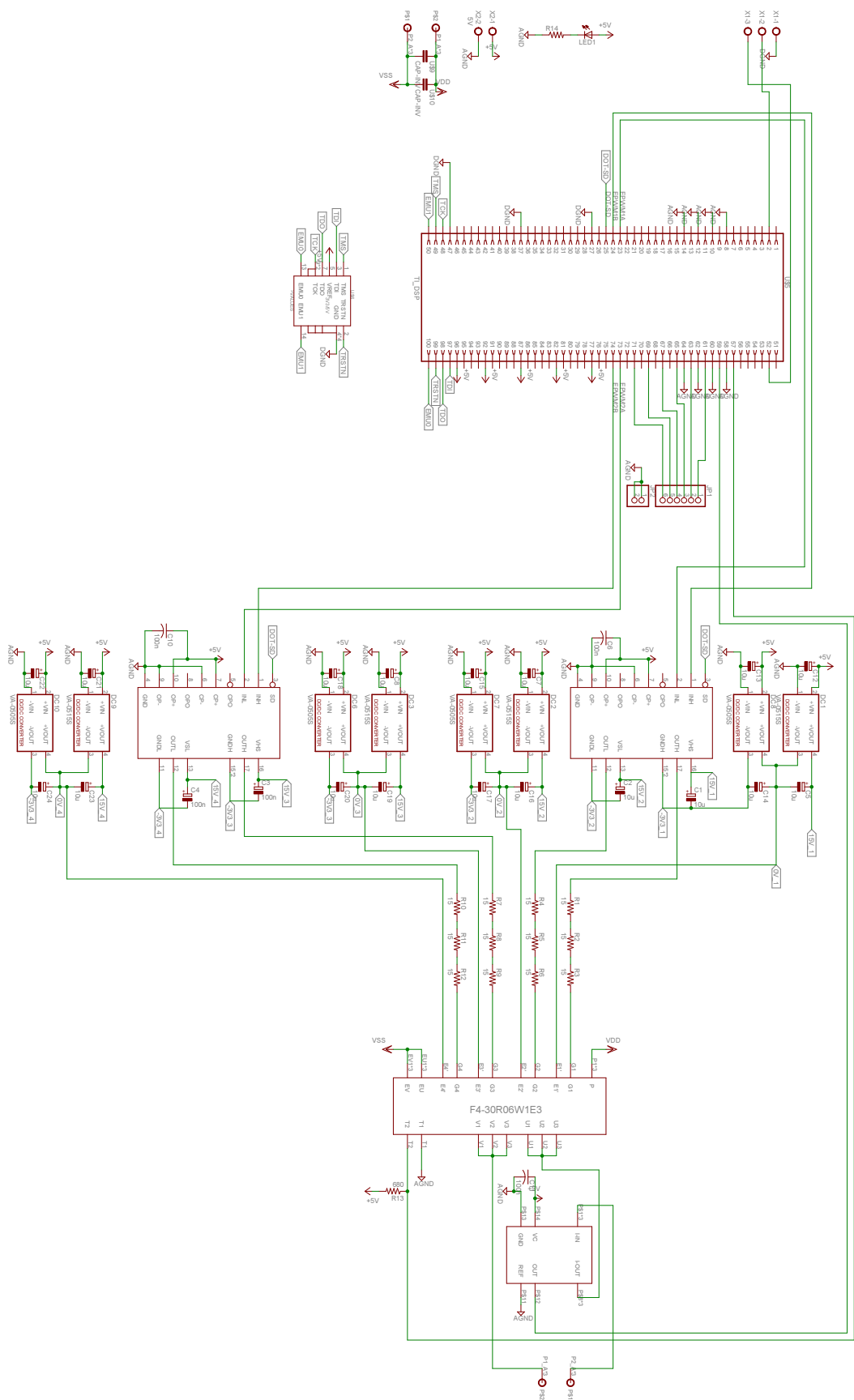
Figure 5.1: Electrical schematic of the inverter

and the AC output wires are realised in the same two layers. These 2 layers are placed in the middle of the board to prevent any dangerous situation for users. The other 2 layers are used for all the signals and also for the power supply of the drivers. In picture 5.2 we can see the final project of the board with all the layers.

The big pink layer represent the 5V power supply of the board and the light blue one the relative ground. These two layers gave us the big advantage to easily power the various devices. In fact, if the layer passes under the specific point where we need power, the connection will be realised just using a via.

On the bottom left part of the board there are the two wires of the DC input power, specifically the pink one is the positive voltage and the light blue the ground. Instead on the bottom right, precisely after the switching device, there is the AC output voltage of the inverter. As we can see the two wires are as much width as possible because of the big current they have to tolerate. Also the thickness of these layers is bigger than for the other two for the same reason. In particular the manufacturer of the board suggests to use a wire at least 0.1mm width for each Ampere that has to flow in that wire. All the devices were mounted on the top side of the board, except for the switching device that is mounted on the bottom side because is necessary to fix it on the heat-sink.

Designing the board we tried, when it was possible, to follow some rules, in particular:

- Minimize the length and the distance of the wires that connect the drivers and the switching device to reduce the parasitic inductances.

- Use vias only for slow signals.

- Separate the board in a "signal area" and in a "power area".

- Minimize the distance of the filtering capacitor to the devices.

As we can see the board is divided in two main areas. On the bottom part there is the "power area" with, starting from the left: the DC input connector, the capacitor of the DC-link, the switching device, the current sensor and the AC output connector.

Instead, on the upper part there is the "signal area" with, starting from the left: the 2-pin connector for the 5V power supply and the 3-pin connector for the serial communication port, the strip connector for the analog input of the microcontroller, the big connector of the microcontroller, the 2 drivers with the filtering capacitors and the relative power supply and, on the right, the connector to program the microcontroller.

On the board are present 6 holes, 4 on the corners to fix the board to a stand and 2 to fix the switching system to the heat-sink.

In picture 5.3 we can see the board without the four power layers, so we can appreciate better the signal wires. As we can see the most of them are on the top of the board, and all the vias are used for slow signals. Something that we can not see from these pictures is that the analog and the digital ground are connected together in one point corresponding to the 3-pin connector. This assure that the digital ground signal is not influenced by the current that flows through the analog ground.
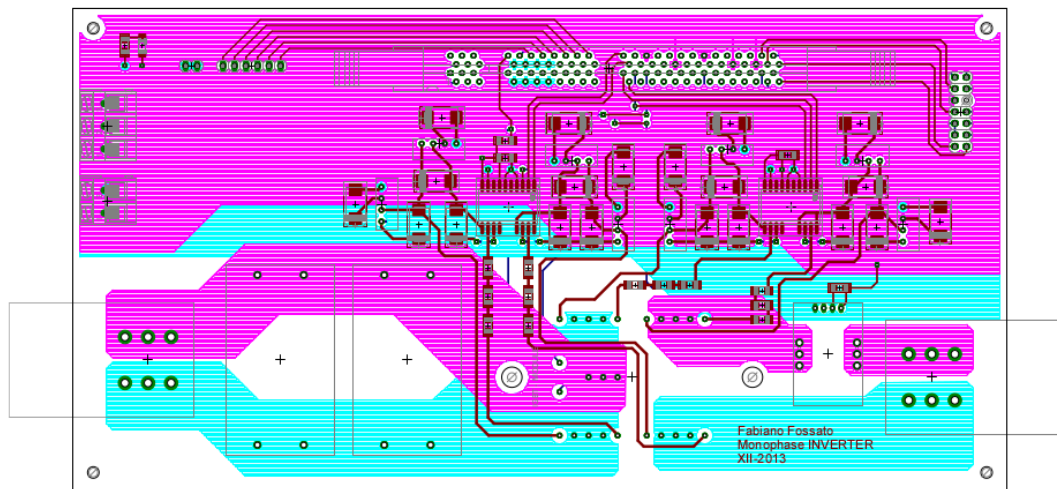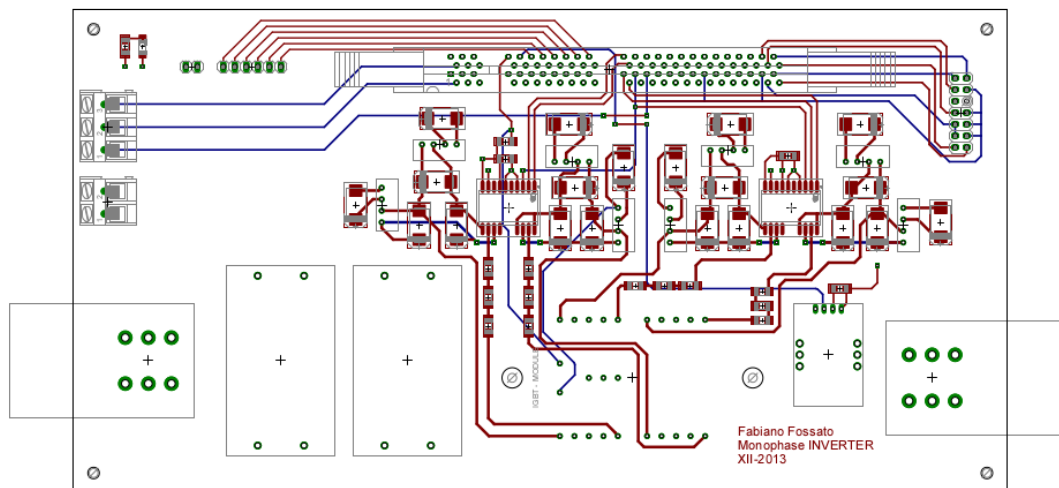
Figure 5.2: Printed circuit

Figure 5.3: Printed circuit without the power layers

5.3   Realising the board

The board was produced by an external producer and then assembled in the university laboratories.
On the top view picture (fig: 5.3) we can see the wires and the pad of the SDM devices.

Initially, as we can see in the picture 5.5, the board was tested without the heat-sink. This to check
with low power if everything was fine. The board was tested in this condition with a voltage less then $10V$
and a current less than $1A$. Once we checked some basic characteristics – like the dead-time – the board
was equipped with an heat-sink provided of a fan and put inside a "safety box". The safety box must be
used for several reasons. In fact, this inverter works with high values of voltage and of current and also
the temperature of the DC-link capacitors and of the heat-sink can be high enough to hurt. Furthermore,
we have to remember that the maximum power of this inverter is around $9kW$ and if something goes
wrong – for example a short circuit due to a dead-time too short – the device could explode or catch fire.

Here below we can find some picture of the realising process. In order: the board without any
component, the board during the soldering process, the board completely assembled and the board with
the heat-sink inside the safety box.
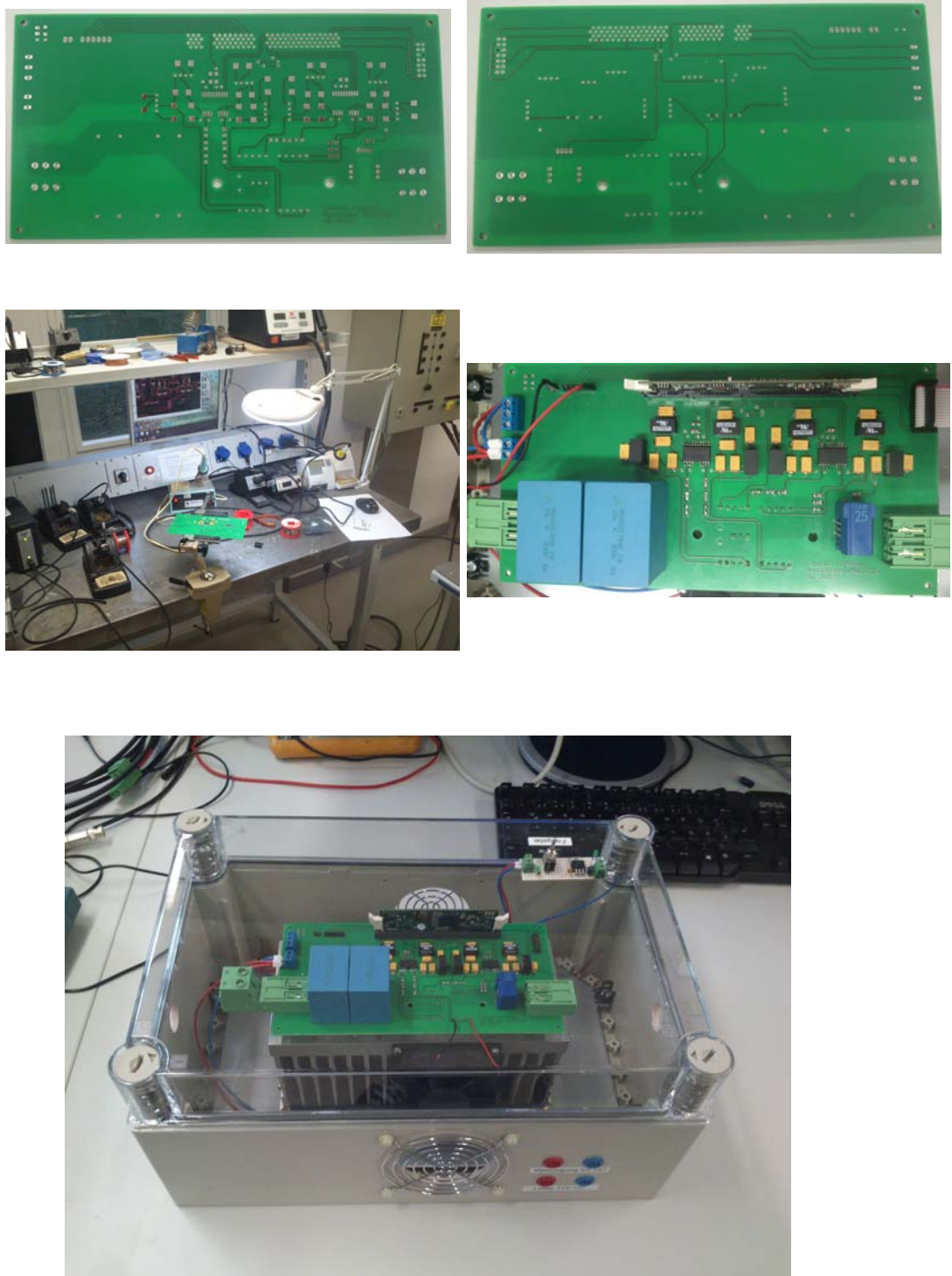
**Figure 5.4:** Board completely mounted in the safety box

5.4 Testing the board

As written above initially the board was tested without the heat-sink as we can see in picture 5.5. The first step was to test the inverter without any external load and check the output voltage starting with a low value of the input and increasing it till the nominal value. We did this test to check if the dead-time was long enough. In fact, if the dead-time is not sufficient, the output voltage will have some negative peaks during each switching due to a temporary short-circuit of one arm. If these voltage peaks are too big they can cause a big current peak, increasing very quickly the temperature of the junction and decreasing the efficiency or, in the worst case, destroying the component. To fix a value of the dead-time

we proceeded starting with a very big dead-time (around $10\mu s$) and decrementing it till few μs. The picture 5.6 represent the output voltage of the inverter with an input voltage of $300V$ and no load. As we can see there are some peaks but they are not so big. We have in fact also to consider that there is not any capacitor at the output to maintain a stable value, so a little bit of "noise" is understandable. For this reason we decided that this kind of peaks was acceptable and we stopped the decrementing of the dead-time at $1,36\mu s$.

Once we have fixed the dead-time we initially tested the inverter (still without the heat-sink) with a low power and only with a resistive load, then with a ohmic-inductive load: precisely with 25Ω in series to an inductance of $3.13mH$. In picture 5.7 we can see the trend of the output current during the first test without heat-sink. We can see that the signal is pretty similar to a sinusoid but when the value of the current is close to zero it results deformed. This is due to the very low value of the current we were forced to use because of the absence of the heat-sink. In this case the peak value of the current is around $160mA$ (the probe attenuation is 20). In fact, if the current that flows through the IGBTs it is too low they do not work in the saturation zone, but in the linear one, deforming the sinusoid. Anyway, except for this problem the inverter seems to work fine, so we proceeded to build the heat-sink and the safety box to test the board at the nominal condition.

The final test-bench we used is in picture 5.8. On the right of the picture there is the big resistance that can tolerate a maximum current of $30A$ and that has a maximum value of 5Ω. Under the table there is – connected in series with the resistance – the inductance, with a value of $3.13mH$. On the table, starting from the right there is the inverter, the power supply for the DC input of the inverter, a signal generator, a device for the current probe and the oscilloscope. The signal generator was used to supply the reference signal for the PWM modulation. In fact, even if we can save the reference signal in the memory of the microcontroller, during the testing of the board we preferred to use an external reference. In this way we can change instantaneously the reference signal without the necessity to reload the program in the microcontroller.
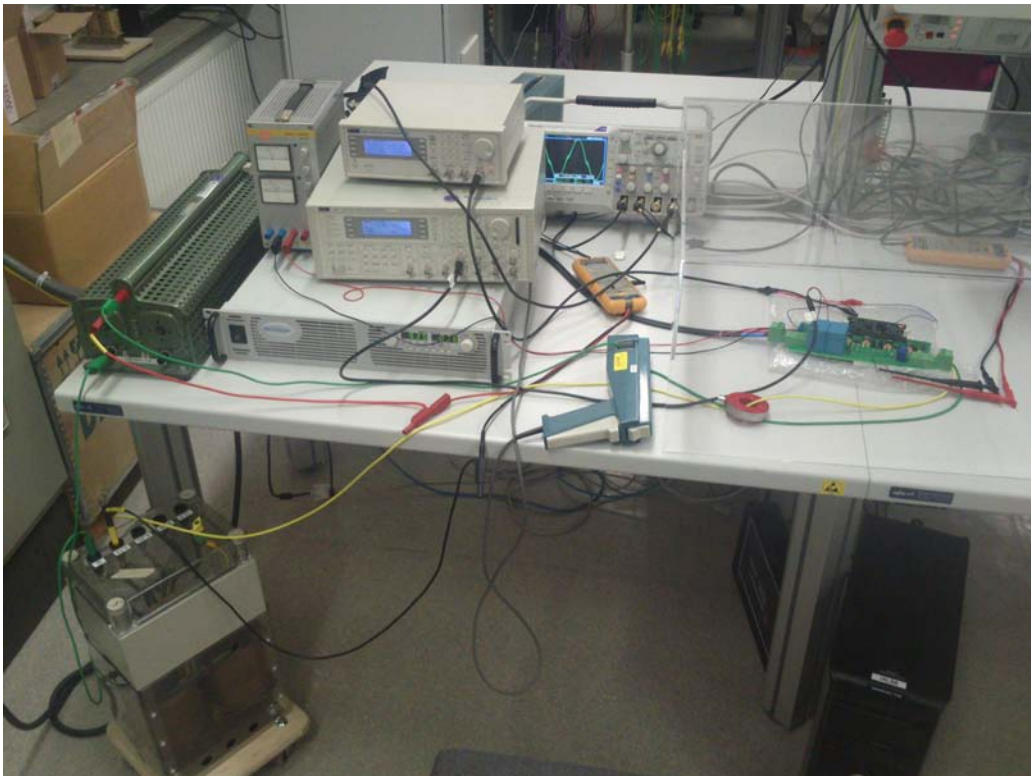
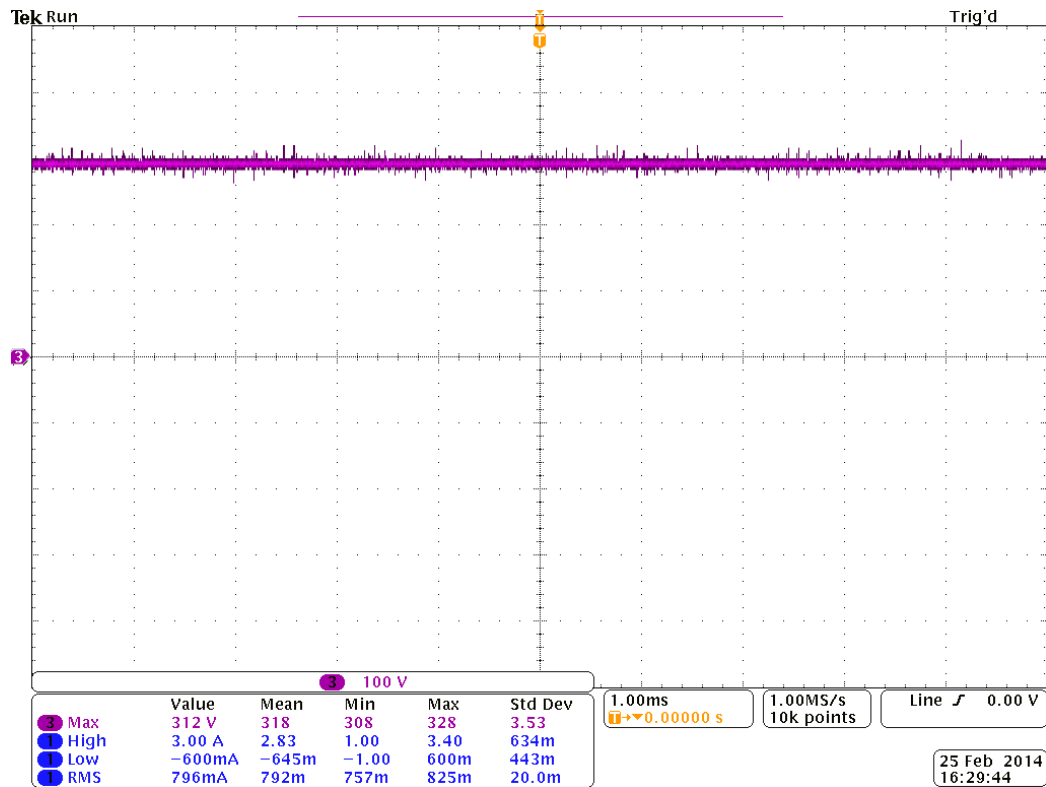**Figure 5.5:** First test realised without the heat-sink and the safety box

Figure 5.6: Vout of the inverter with an input voltage of 300V and no load
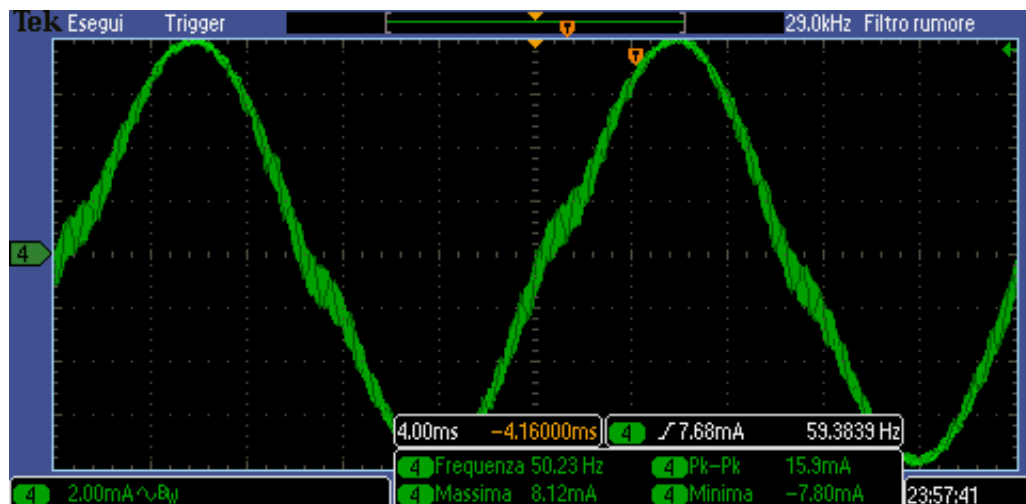

Figure 5.7: Test of the board with no load and no heat-sink with a low value of the current

Figure 5.8: Final test configuration

5.5   Conclusion

The final purpose of our project was to test completely the inverter and to verify if the model matches well the real trend of the losses. To check this it is necessary to know the thermal parameters of our heat-sink, but these were not know. In fact, the heat-sink that was installed in this inverter has not a known value of thermal resistance. Anyway, to resolve this problem, is possible to measure this parameter measuring at the same time the losses and the temperature of the heat-sink for different values of losses. Once this parameter is found it is possible to simulate the real working condition in the model and to compare correctly the results. Anyway we had not the time to make this step, and we limit our work to test the inverter. Even if the inverter was designed to work with an input voltage of 300V and a current of 30A, the power supply we had in the laboratory was not sufficient to test it at the nominal condition. The main reason is that there was not a load capable to tolerate a power of 9kW for more than few seconds: this means that we tested our inverter with a current of 30A and with a voltage of 300V for a long lapse of time, but not at the same time. Anyway in both the conditions, and even with a current of 25A and an input voltage of 200V the inverter worked properly with a modest increase of the heat-sink's temperature. Reasonably also at the nominal conditions the increase of the temperature will be still limited, this may suggest that the heat-sink is a bit oversized and if necessary, it can be replaced with a smaller one.

Bibliography

[1] NICOLAI U., WINTRICH A., TURSKY W., REINMANN T., 2011, *Application Manual Power Semiconductor*, SEMIKRON International GmbH, Germany.
`http://www.semikron.com/download/assets/pdf/application_handbook/`
`application_manual_complete.pdf`

[2] SPIAZZI G., CORRADINI L., 2013, *Appunti dalle Lezioni di elettronica per l'energia*, Padova.

[3] Datasheet INFINEON F4-30R06W1E3
`www.infineon.com`