JETIR.ORG ISSN: 2349-5162 | ESTD Year : 2014 | Monthly Issue
JOURNAL OF EMERGING TECHNOLOGIES AND INNOVATIVE RESEARCH (JETIR)
An International Scholarly Open Access, Peer-reviewed, Refereed Journal

Power MOSFET Reliability: LTspice-Based Junction Temperature Analysis

Santosh Arcot

IEEE Member

Abstract : One of the important feature of power switching system is the use of fast switching power semiconductor devices. MOSFET is used in fast switching applications including wireless power transfer switching systems. A thermal model based on thermal time constants is necessary to accurately predict MOSFET device power dissipation and characteristics. Many thermal models discussed in the literature are based on linear approximation and not designed to incorporate dynamic MOSFET characteristics and heatsink model. In this paper, we present literature survey of existing thermal models. A model was developed for MOSFET $\mathbf{R_{DS(ON)}}$ was analyzed along with average power calculation, heatsink temperature and junction to case temperature. Transient thermal model for $\mathbf{R_{DS(ON)}}$ was incorporated into full bridge resonant model using LT spice simulation tool. MOSFET power dissipation & junction temperature within the Semiconductor device is calculated. The proposed model has a dynamic feature, adjusting the device resistance according to simulation time. As a result, the model is well-suited for predicting MOSFET junction temperature based on the resonant current flowing through the device. Through simulation results, we provide estimations for junction temperature rise and average power dissipation, validating the effectiveness of proposed approach.

***Index Terms* - MOSFET, reliability, LT spice, power, temperature, high voltage.**

I. INTRODUCTION

Semiconductor devices used in high voltage power supply applications include MOSFET, IGBT and SiC MOSFET devices (Kimoto, 2022). The determination of a semiconductor device operating temperature is a critical aspect of any high voltage design. As the circuit operating time exceeds the MOSFET's thermal time constant and with device current being a repetitive waveform, the MOSFET is expected to reach a thermal equilibrium and the associated steady state temperature can be modeled like an electrical resistive network. Temperature is derived by multiplying power dissipation (wattage) with MOSFET thermal impedance (Celsius/wattage). However, if the operating time less than the thermal time constant of the MOSFET and if the device current is non-repetitive in nature, the estimation of device temperature may involve lengthy calculations (Muthupandian C, 2012). It is therefore desirable to generate a temperature-time curve for a MOSFET using a circuit simulation process. The objective of this document is to present a LTspice thermal model, that calculates the operating temperature of a MOSFET, operating with a transient waveform.

The problem of calculating a MOSFET junction temperature using SPICE has been dealt with in the past (UM1575 user manual, 2013). However, the model assumes constant heating during the time of applied device power dissipation. Thus, it does not consider the effect of transient waveform and effect of heat sink cooling during a transient. Furthermore, the effect of variation of $R_{DS(ON)}$ of MOSFET due to junction temperature is not considered. For better estimate of MOSFET junction temperature rise, both parameters need to be considered. In this paper we present a method to determine MOSFET (Metal Oxide Semiconductor Field Effect Transistor) junction temperature rise during short term overload. This model is based on LTspice simulator. The method can be used to evaluate power MOSFET reliability and help assist device selection suitable for a given application. This paper also provides a theoretical analysis for steady-state and transient thermal impedance. A method for simulating full bridge resonant converter based on MOSFET devices is presented and a method to estimate power dissipation & junction temperature within the semiconductor device is presented.

II. SIMULATION MODEL

The model presented uses the analog modeling approach using LT spice. The behavioral models can be found in the LTspice library. The proposed model will measure MOSFET drain to source voltage and drain current. Instantaneous power dissipation of a MOSFET is calculated using Equation 1.

$$P_D(t)=V_{DS}(t)*I_D(t) \qquad (1)$$

Instantaneous power $P_D(t)$ is integrated, which yields the value of average power dissipation P_{AVG}. Average power $P_{AVG}(t)$ is a moving average and indicates the average value of power dissipated within MOSFET. Figure 1 shows the heat transfer mechanism between a metal heatsink and power MOSFET. A gap pad is used to improve the heat transfer efficiency.

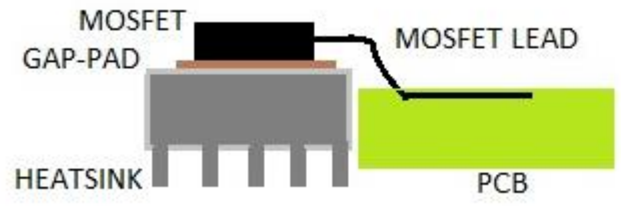

Figure 1: MOSFET Thermal model

The thermal impedance of gap pad is generally observed to be much lower than MOSFET junction-to-case thermal impedance. For ease of calculation, we assume a value 50% of MOSFET junction to case thermal impedance. Figure 2 below shows the simulation model used to determine heatsink temperature. The power dissipated, P_D is converted to current in a 1:1 ratio, which in turn drives an R-C circuit representing heatsink thermal capacitance and resistance respectively. The value of C and R is determined by a heatsink dimensions and material type. T_A represents the ambient temperature and heatsink temperature is seen higher than ambient temperature.

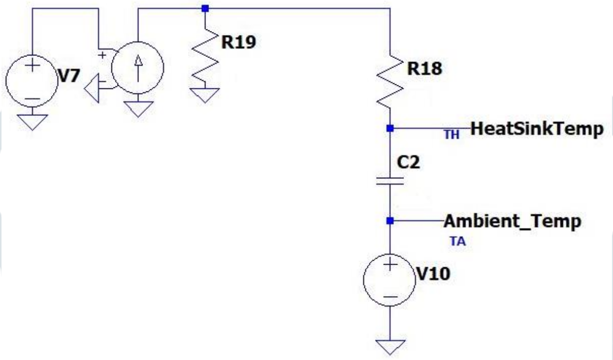

Figure 2: Heatsink Portion of Thermal Model

The MOSFET drain-to-source on resistance, $R_{DS(ON)}$, varies with respect to the change in junction temperature. This variation in device ON resistance was analyzed by (David Divins, 2007) using a voltage source model. Since the resistance is dependent on the junction temperature (Figure 3), the variation in $R_{DS(ON)}$ will alter the drain-to-source voltage drop V_{DS}. Thus, the average power dissipation in a MOSFET would be different if this variation is not considered. Referring to Figure 4, the variation in $R_{DS(ON)}$ is incorporated in the simulation model by injecting a voltage source, V_{DS}_IJ, in series with the MOSFET. This injected voltage is equal to the multiplication of drain current and the new $R_{DS(ON)}$ which is calculated while the simulation progresses based on the junction temperature T_J. The formula for this injected voltage is given by (David Divins, 2007). The device model for the MOSFET used in the simulation contains an $R_{DS(ON)}$ term in it. This default value is the nominal resistance value at room temperature and does not vary with respect to simulation time. Thus, the injected voltage will not consider MOSFET resistance. Only the incremental voltage estimated based on $R_{DS(ON)}$ increase above its nominal value is injected in series with the MOSFET.
Note that the drain voltage V_D considered for the average power dissipation calculation is total of injected voltage and the drop across MOSFET. If two or more MOSFETs are used in parallel, this voltage should be injected in each of the MOSFETs with drain current divided equally between them.

In the following analysis, we consider MOSFET part number STB120NF10 from ST Microelectronics. STB120NF10 Datasheet shows $R_{DS(ON)}$ variation with temperature. Using the datasheet values, curve fitting is done to extract polynomial equation and plotted in Figure 3 .

$$R_{DS(ON)} = *0.0005T_j^2 + 0.1269T_J + 22.537)*R_{DS(ON)\,nominal} \qquad (2)$$

$$V_{DS}\_IJ = (0.0005T_j^2 + 0.1269T_J + 22.537)\, R_{DS(ON)\,nominal} * I_D \quad (3)$$

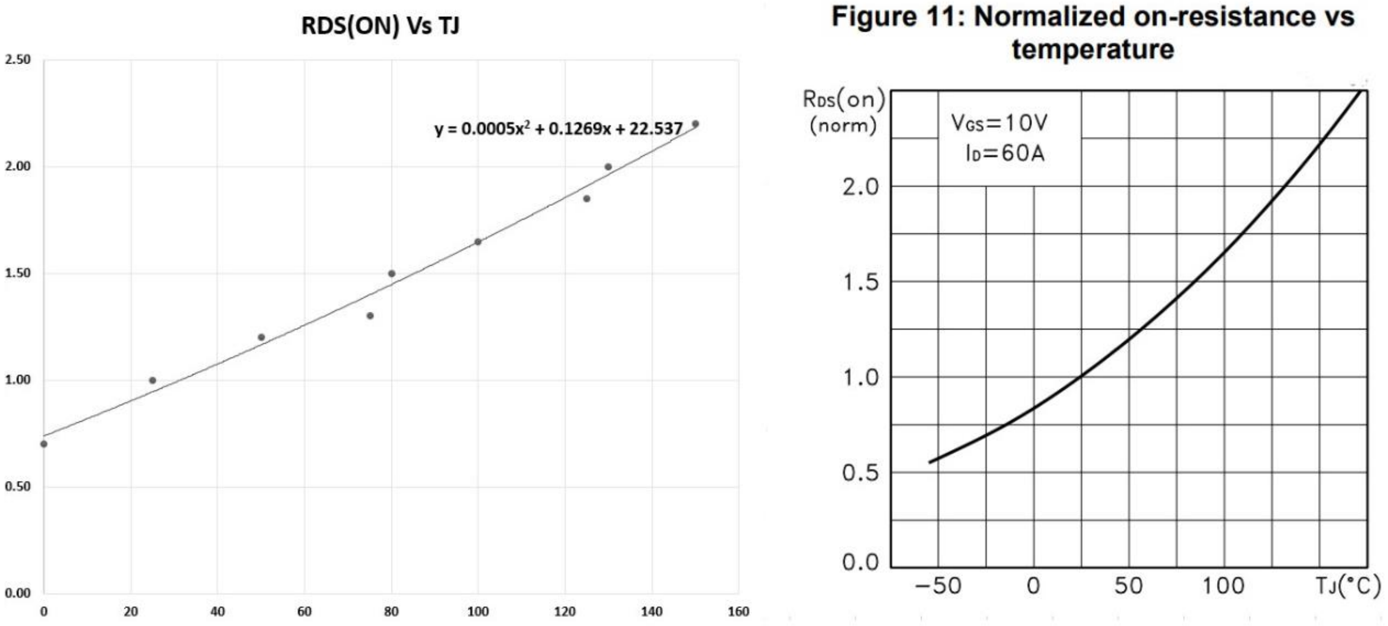

Figure 3: $R_{DS(ON)}$ MOSFET

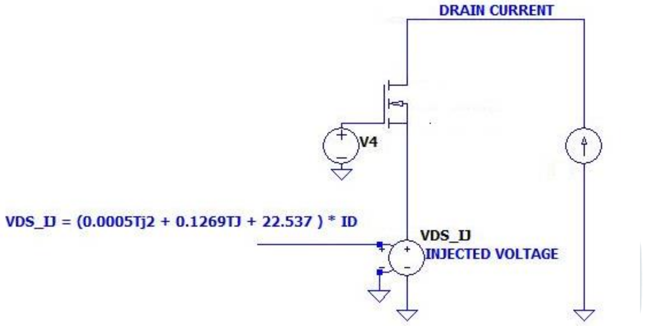

Figure 4: MOSFET Series Voltage Injection

Simulation Model and Implementation

The LT spice simulation model consists of plotting average power dissipation, plotting junction temperature and computation of new $R_{DS(ON)}$ and Injected voltage in series with MOSFET. This completes the closed loop implementation of MOSFET thermal model.

Average power dissipation

As shown in Figure 5, average power is calculated using drain current I_D and drain-source voltage V_{DS}. The values of $V_D(t)$ and $I_D(t)$ are used to calculate instantaneous power dissipation. Average power is derived from the integral of instantaneous power.

Figure 5: Average Power Calculation

Plotting MOSFET junction temperature

Referring to Figure 6, thermal impedance of MOSFET varies with pulse width and time constant for junction to case temperature rise is derived from datasheet. An excel calculation was performed to derive an exponential Equation 4, for thermal impedance.

$$Z_{JC} = 0.48x^{0.0888} \qquad (4)$$

.

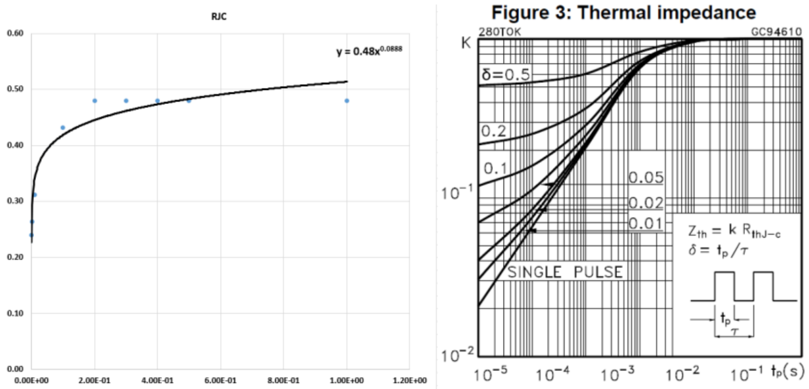

Figure 6: MOSFET Junction Temperature Rise Computation

Computation of $R_{DS(ON)}$ Variation and Voltage Injected

As described previously, the variation in MOSFET $R_{DS(ON)}$ with respect to junction temperature T_J is simulated by injecting a voltage in series with the MOSFET, in proportion to T_J. The value of the new $R_{DS(ON)}$ used is based on the empirical temperature dependence given as,

$R_{DS(ON)} = *0.0005T_j^2 + 0.1269T_J + 22.537)*R_{DS(ON)\ nominal}$

$V_{DS}\_IJ = (0.0005T_j^2 + 0.1269T_J + 22.537)\ R_{DS(ON)\ nominal} * I_D$.

Voltage across R4 in figure 7 is used to calculate polynomial equation 1. The calculated $R_{DS(ON)}$ given by Equation 1 is subtracted by nominal resistance and multiplied by the MOSFET drain current I_D. Analog multiplication techniques using LT spice model was adopted to develop injected voltage and shown in Figure 7.

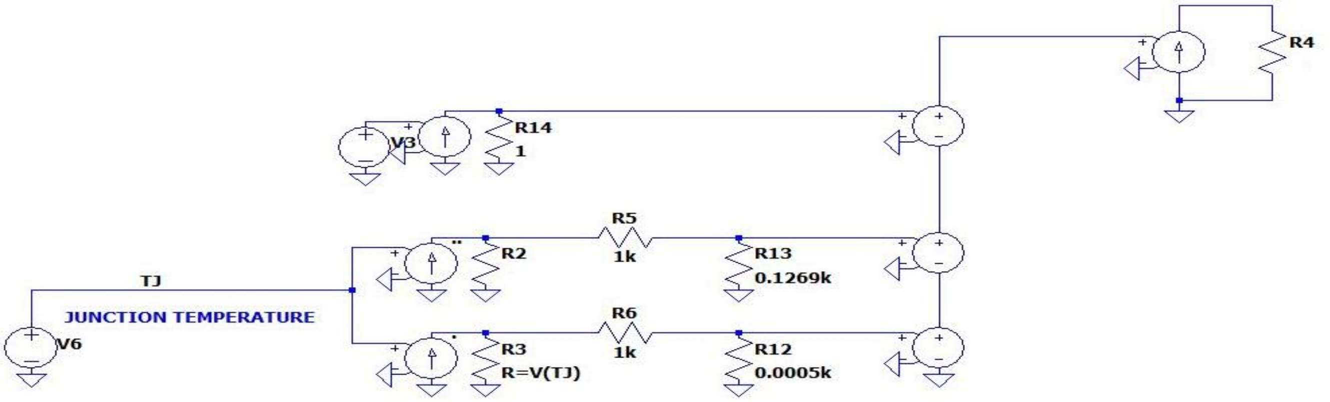

Figure 7: Equation of Injected Voltage in Series with the MOSFET.

Full Bridge resonant converter Simulation

A full bridge resonant converter operating in zero voltage switching is implemented in LT spice (Figure 8). Proposed MOSFET thermal model is then used to calculate device junction temperature. Transient load conditions are simulated which exceed the average current rating of device, although the device current is still within the short-term ratings of power MOSFET. Since the current carrying through MOSFETs last only for few milliseconds, the proposed model helps estimate junction temperature more accurately than a first order approximation. This proposed methodology is helpful in estimating reliability of power MOSFET and to enable selection of the suitable power devices. This model also helps perform component stress analysis based on the tolerances of resonant circuit components and to confirm if power devices can withstand increase stress due to circuit component tolerances.

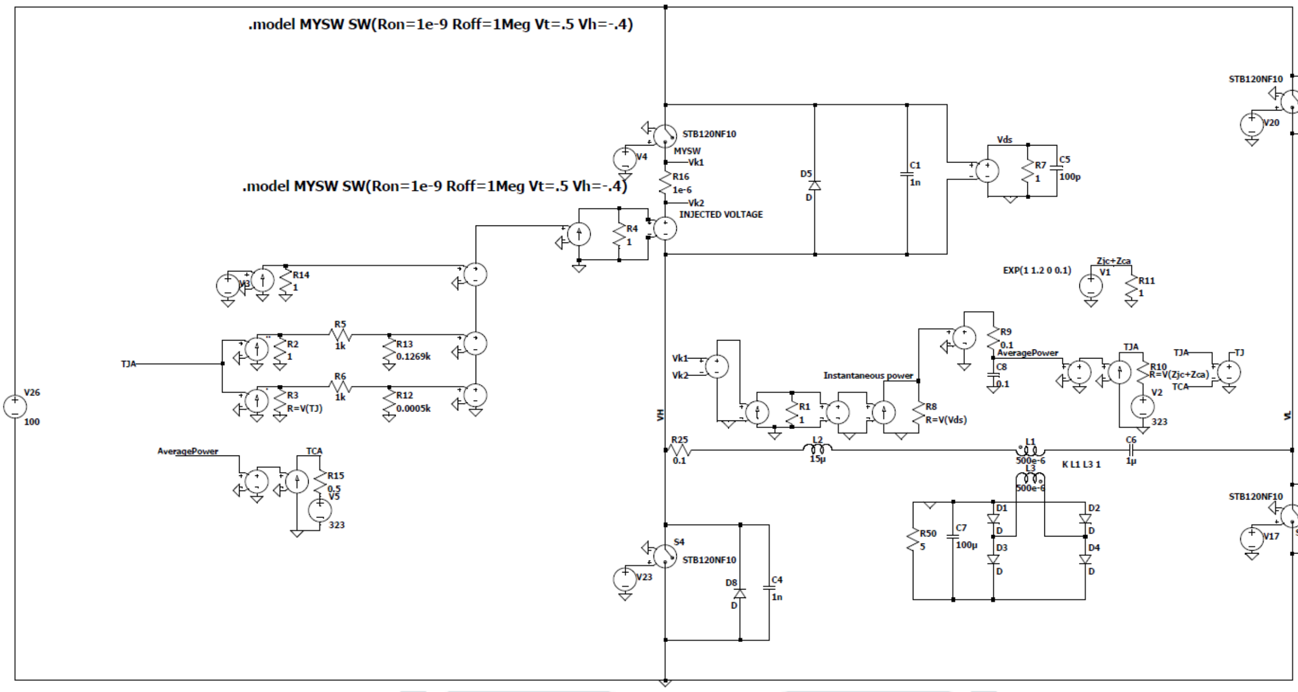

Figure 8: Full Bridge resonant converter.

IV. RESULTS AND DISCUSSION

4.1 Results of Descriptive Statics of Study Variables

A Resonant converter is used to convert AC or DC input voltage to a fixed regulated output voltage. A typical resonant converter is well suited for wireless power transfer systems and is used to improve system efficiency. As described in (S.Arcot, 2023), the leakage inductances effect power switching operation, which in turn varies power dissipation of power devices. Power dissipation of semiconductor devices like MOSFET have positive temperature coefficient due to the nature of $R_{DS(ON)}$. Hence this circuit was simulation to predict MOSFET device temperature. Proposed thermal mode was implemented in LT spice (Figure 8) and results for junction temperature is plotted in Figure9. With an ambient temperature of 323^{0}Kelvin, MOSFET junction temperature reached a peak of 327^{0}Kelvin in 50msec. With an average power dissipation of 4watts (Figure 10), overall junction temperature rise observed per simulation is 4^{0}Celsius. Therefore, the proposed thermal model can predict reliability metrics namely power dissipation and junction temperature. This prediction helps designers in understanding the tradeoffs in selection MOSFET and ensure system goals are met in power switching applications.

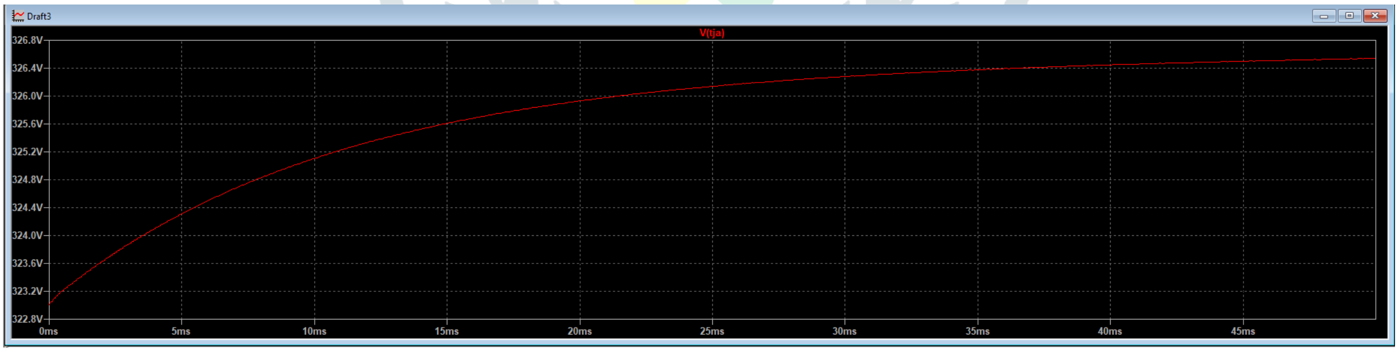

Figure 9: Junction Temperature prediction waveforms

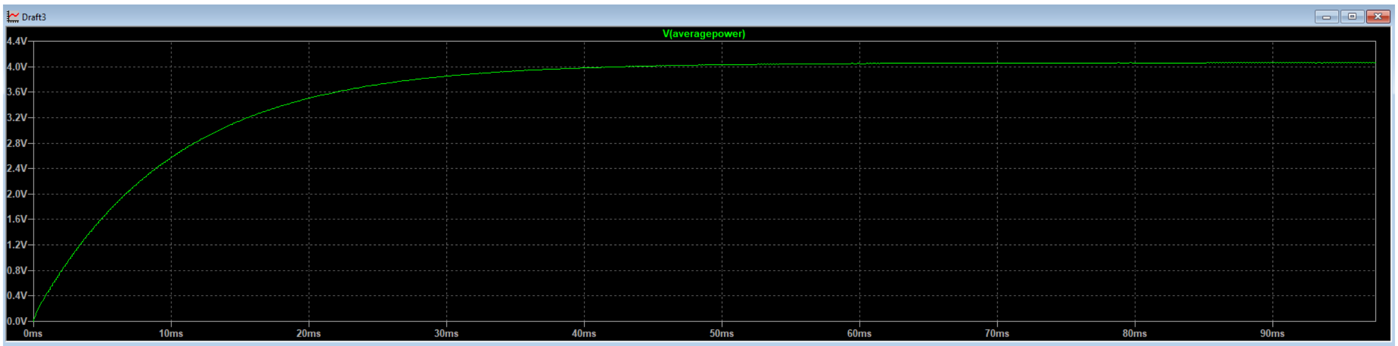

Figure 10: Average power

III. ACKNOWLEDGMENT

The author acknowledges that the research is conducted independently and there is no conflict of interest regarding the publication of this paper.

REFERENCES

[1] Kimoto T. High-voltage SiC power devices for improved energy efficiency. Proceeding of the Japan Academy Series B Physical and Biological Sciences, 2022, 161-189, doi: 10.2183/pjab.98.011.

[2] David Divins, Using Simulation to Estimate MOSFET Junction Temperature in a Circuit Application, International Rectifier, 2007.

[3] SPICE model tutorial for power MOSFET, ST Microelectronics, UM1575, User Manual, 2013.

[4] S. Arcot "Analysis of Induction Based Wireless Power Transfer Scheme", IJRASET, Volume 11, Issue VII, July 2023, pp. 475-479.

[5] Muthupandian C. COMPACT MODELING FOR MULTI-GATE MOSFETS USING ADVANCED TRANSPORT MODELS, Department of Electronic, Electric and Automatic Engineering (DEEEA),Thesis, 2012, Universitat Roveira virgil.