VHDL-AMS THERMO-MECHANICAL MODEL FOR COUPLED ANALYSIS OF POWER MODULE DEGRADATION IN CIRCUIT SIMULATION ENVIRONMENTS

Olufisayo Olanrewaju
Power Electronics, Machines and Control Group
University of Nottingham
Nottingham, UK
olufisayo.olanrewaju@nottingham.ac.uk

Alberto Castellazzi
Power Electronics, Machines and Control Group
University of Nottingham
Nottingham, UK
alberto.castellazzi@nottingham.ac.uk

Abstract— **This work proposes the development of a simplified thermo-mechanical model suitable for coupling with device physics and a 3D electrothermal model in line with the creation of a comprehensive framework for circuit simulation of multidomain problems. Commercially available numerical analysis software are capable of showing thermo-mechanical effects but lack real-time feedback between domains and require sophisticated CAD/meshing. Here, we show a 1D mechanical model coupled to a thermal model which is capable of generating accurate mechanical Strain and stress values of a power assembly while optimizing the tradeoff with computational efficiency. The thermo-mechanical model was created in VHDL-AMS language because of the multi-domain capability of VHDL-AMS.**

Keywords— thermo-mechanical; thermal stress; multi-domain modelling; Wide band gap, Silicon carbide; FEA, VHDL-AMS.

TABLE I: NOMENCLATURE

SYMBOL	*QUANTITY*	*S.I UNIT*
E	Young's modulus	Pa
v	Poison's ratio	-
α	Coefficient of thermal expansion	K^{-1}
T	Temperature	K
Tref	Reference Temperature	K
A	Surface Area	m^2
σ	Stress	Pa
ε	Strain	-
C	Elasticity tensor	Pa
L	Length	m
F	Force	N
λTh	Thermal Conductivity	W/(m K)
ρ	Density	kg/m^3
Cs	Specific Heat capacity	J/(kg K)
$H_{i,j,k}$	Heat (power loss)	W/ m^2

I. INTRODUCTION

The operation of a power assembly (shown in in Fig 1) produces multidomain interactions (e.g. electrical, thermal, and mechanical) which are of paramount importance to power electronic designers [1] especially for Wide Band Gap (WBG) devices where higher power densities require new packaging techniques [2] and reliability monitoring. To observe these interactions in a simulation with accuracy and little compromise on computational efficiency, a methodology was proposed in which different domains (electrical, thermal, electromagnetic and mechanical) interact with each other concurrently in a simulation [1]. The relations between the electrical and thermal (electro-thermal) as well as the electric and electromagnetic domain have been detailed in [1]. To analyze the mechanical aspect in a power assembly, the temperature from the thermal aspect is sent to the mechanical model and based on the temperature difference from a reference temperature (i.e. zero stress reference temperature which is usually the ambient temperature unless stated otherwise), mechanical properties and the equation model generated in this work, stress and strain results are generated.

Two major issues cause skepticism about thermo-mechanical simulations in the power electronics field- first, the effect of mechanical deformation on the temperature (i.e. feedback to the thermal domain from the mechanical domain) and secondly, the effect of temperature/ temperature changes on the mechanical properties of the materials that make up the different layers of the power assembly. We ignore the first issue (feedback from mechanical domain to thermal domain) as quantifying the changes in thermal impedance as a result of mechanical deformation and stress is extremely challenging for a power assembly because of the characteristics of the material used in the assembly [5]. On the second issue, we see from our review of [4] that mechanical properties in power modules can be assumed constant between -55ºC degrees and 125ºC. The temperature results presented in this work have a peak of 141 ºC (414K). Thus, we can assume that up to 141 ºC, the material mechanical properties are constant. [4], [5].

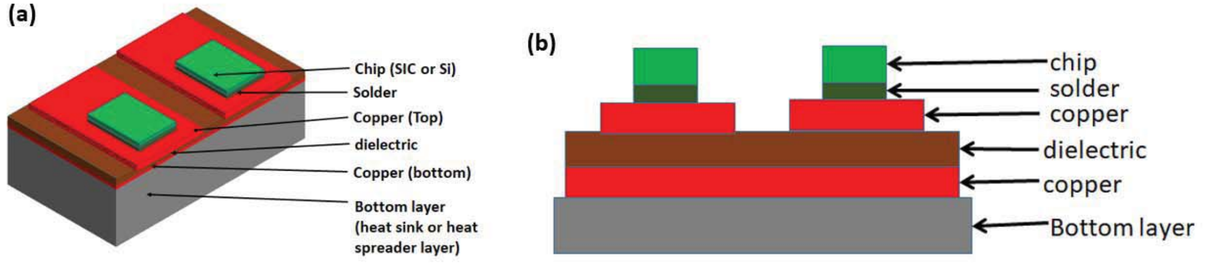

Fig. 1: Power Assembly in 3D (a) and in side view (b) showing the various layers in the power assembly. A power assembly can such as this could be the structure of a power module or the building block of larger power modules structures [2].

With this background information, provided, detail can now be given on the electrical, thermal and mechanical models built in this work, the validation of the models and the use of the models in coupled thermo-mechanical simulations.

II. MODEL DEVELOPMENT

A. Electrical Model

The electrical model is based on the power device model discussed in [1]. The model is a physics-based Silicon carbide (SiC) MOSFET including all main and secondary effects (e.g. breakdown). A number of equations included in the model are shown in (1), (2) and (3).

$$r_{d1} = (Rd0\ L_r + V_{rd1}/Isat)\ A_r \tag{1}$$

$$I_{ds} = B_{eff} (Vg_{eff} - 0.5\ K_T\ Vd_{eff})\ clm \tag{2}$$

$$G_{heat} = V_{rd1} I_{rd1} + V_{ds} I_{ds} + V_{rd2} I_{rd2} + V_{rs} I_{rs} \tag{3}$$

We use the convention I for current and V for voltages; r_{d1} is the resistance of the JFET region of the drain; r_{d2} is the resistance of the remaining drain region; r_s is the resistance of the source region; I_{ds} is the main drain-source current. B_{eff}, Vg_{eff}, Vd_{eff} represent the effective Transconductance, gate voltage and drain voltage parameters respectively. The parameter clm is used to indicate channel length effects while L_r and A_r are the length and areas ratios of the JFET region to the rest of the drain of the Sic MOSFET. For more details on the electrical aspect model see [6] and for validation of the electrical model as part of the proposed methodology, we refer the reader to [1].

B. Thermal Model

A 3D numerical model capable of multilayer analysis was built to generate the temperature results which were then passed to the mechanical model. We discretized the power assembly in Fig. 1 using the mesh network shown in **Fig. 1**Fig. 2. Based on the axis direction of Fig. 2(a) and the convention in Fig. 2(b), the finite difference equation for each node in the 3D thermal model was derived and is shown in (4) [1]. Power loss from the electrical model was applied to at the top of the mesh structure at nodes in the region where the chips are located (at these nodes $H_{i,j,k}$ had a non-zero value) The boundary conditions were adiabatic (negligible heat flow) for the top and side nodes while convection boundary condition was used for the nodes at the bottom to represent the heatsink.

$$\rho . Cs \frac{\partial T\ i,j,k}{\partial t} = \lambda \tau h\ [-2T_P \left(\frac{1}{h1h2} + \frac{1}{h3h4} + \frac{1}{h^2}\right) + \left(\frac{2T_N}{h3(h3+h4)}\right) + \left(\frac{2T_W}{h2(h1+h2)}\right) + \left(\frac{2T_S}{h4(h3+h4)}\right) + \left(\frac{2T_E}{h1(h1+h2)}\right) + \left(\frac{T_A}{h^2}\right) + \left(\frac{T_B}{h^2}\right) + H_{i,j,k}] \tag{4}$$

For the analysis of the power assembly in Fig. 1, we used a constant step size in the X and Z axis (i.e. equal stepsize in nodes for any axis by making h1=h2; and h3= h4 ;) while the step size in the Y axis is varied across nodes in the Y axis based on the thickness of each layer in the power assembly.

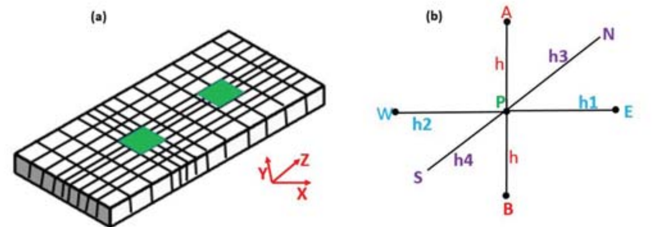

Fig. 2: Mesh used for the power assembly in Fig. [1]. The chip region is colored green in (a) and the convention used in (4) is shown in (b)

The discretization in the Y-axis highlights the benefit of the non-constant mesh step approach for analyzing power assembly structures as proposed in [1] where multilayer systems with layers which have different thickness can be simulated with minimal effect on computational speed.

C. Mechanical Model

As with other domains (electrical, thermal), the mechanical domain is also guided by a fundamental set of equations (Hooke's Law) which relates the stress and strain of a material [3]. The generalized Hooke's law is shown in (5) [3].

$$\sigma - \sigma 0 = C : (\varepsilon - \varepsilon 0 - \alpha\theta) \qquad (5)$$

Where $\theta = T - Tref$; $\varepsilon = 0.5 * (\nabla u + (\nabla u)^T)$; $\sigma 0$ *and* $\varepsilon 0$ *are initial stress and strain respectively.*

Work has been done to solve (5) for a power assembly using a Finite Element Analysis (FEA) model [3]. To solve the issue of computational efficiency that comes with using FEA approaches, closed form solutions were used in [2] to solve (5) treating Fig. 1 as a multilayer system with the heatsink being the substrate while other layers were treated as films.

In line with our application need for the thermo-mechanical model (i.e. for analyzing a power assembly structure), we discovered a number of assumptions can be made in respect to our application need.

a) We would consider a 1D heat flow from top of chip to the bottom of module.
b) The only source of strain comes from thermal (temperature changes). Plastic and elastic creep was assumed to be zero.
c) α is constant. It does not change with temperature.
d) The mechanical model is NOT changing any variable in the thermal model. (The reasons for assumptions c and d have been discussed in the introduction section above).

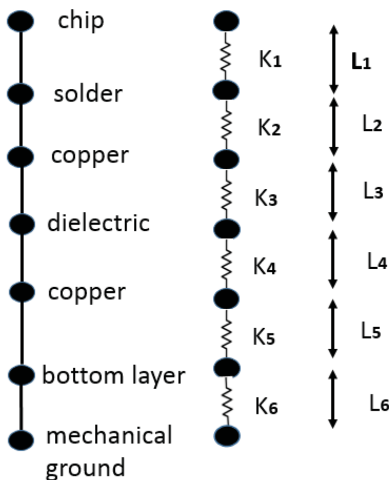

Fig. 3: 1D Force–stiffness matrix node network for the power assembly in Fig. 1.

Based on these assumptions, we can treat Fig. 1 as a 1D beam [7] and each layer as a node on the 1D beam (see Fig. 3). Each node is given a corresponding thermal force derived from the highest temperature value of that layer in the thermal model using (6). Each layer is also given individual stiffness values which are calculated using (7). From the individual force and stiffness values, we generate individual linear Force –Stiffness– Displacement matrix [7] for each layer which includes the interface between adjacent layers. The individual matrices are then added together, coded in VHDL-AMS and solved to generate the displacement (u) values for each layer. For mechanical boundary conditions, we keep the bottom layer fixed [3] (equate the u value for the bottom layer to 0).

$$F_thermal = E\ A\ \alpha\theta \qquad (6)$$

$$K = (A\ E)/L \qquad (7)$$

$$F = K\,u - F_thermal \qquad (8)$$

$$\sigma = F/A; \qquad \varepsilon = u/u0 \qquad (8)$$

Where u0 *is the initial length (thickness) of each layer.*

Once the displacement values have been generated, (8) is used to generate the mechanical force in each layer while stress and strain values for each layer are generated using (9).

D. Modelling Language

The multi-*domain* capability of VHDL-AMS is the major reason we choose it as the modelling language compared to other alternatives for this work. [1]. Due to this benefit, in the mechanical model; we do not need to worry about writing extra equations to ensure the energy conservation laws are kept. Provided, the *ACROSS* (displacement) and *THROUGH* (Force) quantities at each node in Fig. 3 are defined, VHDL-AMS automatically ensures energy conservation thus saving model design and model compile time.

III. RESULTS

To validate this approach, a thermal load of 1kW was applied to the chip in Fig. 1 with an ON time of 1 second and a total period of 5 seconds. Convection coefficient for the bottom surface was 10,000 W/m^2K with an ambient and reference temperature of 27^oC. For validation of the thermal and mechanical model, the power assembly in Fig. 1 was simulated in Abaqus [8] (Abaqus is a FEA software).

A. Thermal Validation

The mesh plot of the FEA thermal simulation at time 1 sec is shown in Fig. 4. We show for the hottest node in each layer (because as explained in the model section, we pass the temperature value at that node to the mechanical model), the result of the FEA model and VHDL-AMS model over the test

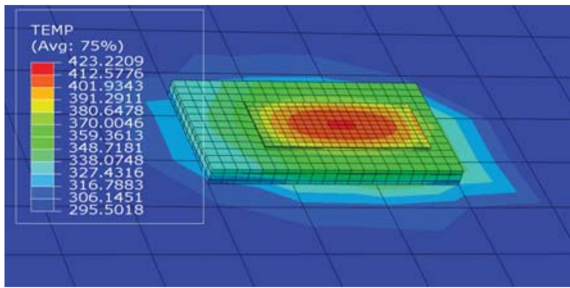

Fig. 4: Temp(K) for the FEA model at time 1 second during the test cycle.

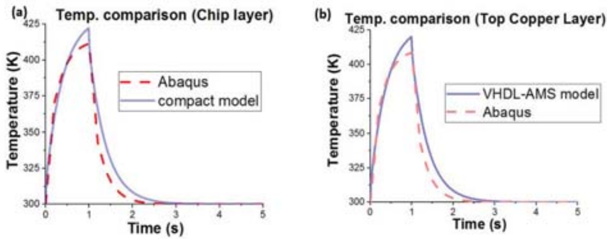

Fig. 5: Temp (K) comparison between the FEA model and the VHDL-AMS model for the chip layer (Fig. 6a) and the top copper layer (Fig. 7b)

cycle in Fig. 5. Due to symmetry, we show just the results of half of Fig. 1 b. The close results for the chip layer (Fig. 5a) and the top copper layer (Fig. 5b) show that the VHDL-AMS model is able to produce accurate temperature results.

B. Mechanical Validation

The FEA simulation of Fig. 1 at time 1 second is shown in Fig. 8. As done with the thermal validation, we compare the stress and strain values at the hottest (as explained under the mechanical model section) in Fig. 9 . The stress comparison

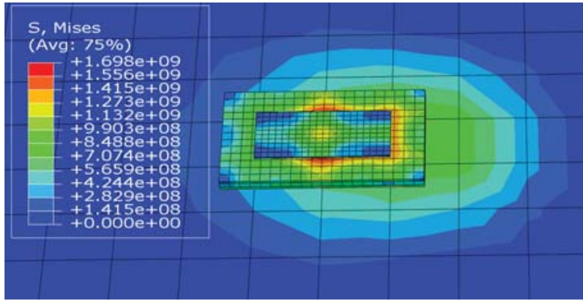

Fig. 8: Stress (Pa) for the FEA model at time 1 second during the test cycle.

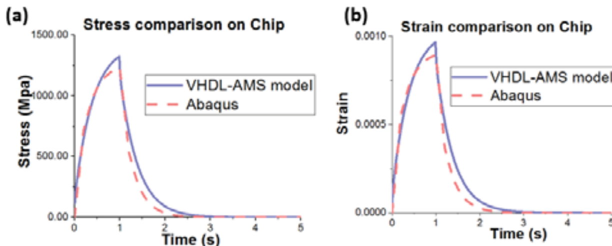

Fig. 9: Stress (Pa) (Fig. 10a) and Strain (Fig. 11b) comparison between the FEA and VHDL-AMS model

TABLE II. SPEED ACCURACY COMPARISON

MODEL	***NODES (mechanical model)***	***Max Temp. (K)***	***Max Stress (MPa) on chip layer***	***Total Simulation Time (secs)***
FEA	3024	412	1230	43.59
VHDL-AMS	7	414	1242	9.49

(Fig. 9a) and strain comparison (Fig. 9b) results show good matching between the VHDL-AMS model and the FEA model. This proves that the VHDL-AMS model is capable of producing accurate results in the mechanical domain.

C. Speed and Accuracy Comparison

A comparison of speed and accuracy of the two models based on the test conditions described above is done in TABLE III. The FEA mechanical model mesh of Fig. 1 has 3024 nodes meaning 3024 equations to be solved numerically. The corresponding number for the VHDL-AMS model is 7. The difference in temperature is 2K which is within our absolute error tolerance parameter of 10 K. The difference in stress values between both models is 0.97% which is far less than our tolerance parameter of 10% [2]. The VHDL-AMS model was implemented in a popular circuit simulator (SIMPLORER [9]) and for the same test condition has a smaller simulation time. Thus, our thermo-mechanical model provides a good trade-off between accuracy and computational efficiency.

IV. CONCLUSION

A methodology for conducting thermo-mechanical analysis of power assemblies such as power modules was described in this work. The goal of this work was to produce a coupled thermo-mechanical model that is accurate, eliminates sophisticated CAD/ meshing and is fast. Thus, it can be used to generate initial temperature, stress and strain values of new power module designs for WBG devices. The equations and assumptions behind the models in this work were discussed in detail. The reason for the choice of VHDL-AMS as the modelling language in this work was also discussed. The VHDL-AMS model was used in a popular circuit simulator without any numerical or convergence issues proving that the model can be used in circuit simulation environments.

We validated the results of our model with a similar model created in FEA because FEA simulations are an acceptable benchmark in industry. The speed advantage of the model created in this work was also highlighted. The 1D nature of the model means that only one temperature value per layer can be fed into the mechanical aspect of the model. The user of the model is free to decide what temperature is transferred to the mechanical aspect but we have chosen to transfer the values at the hottest node and discussed the reason for doing so. There exists a possibility of extending the mechanical aspect of the model to 3D. Based on the validation done in this work, the model can be used for parametric analysis where the thickness of layers, layer material and other parameters can be varied and the effect on temperature, stress and strain is observed.

ACKNOWLEDGMENT

The authors will like to acknowledge the Petroleum and Technology Development Fund whose contribution has been vital to the progress of this work.

REFERENCES

[1] O.Olanrewaju, B. Mouawad, A. Castellazzi, R. Kraus "VHDL-AMS Modelling of Multi-chip SiC Power Modules" in 4th IEEE workshop on Wide Bandgap Power Devices and Applications (WIPDA) 2016.

[2] L. Boteler, S. Miner "Power Packaging Thermal and Stress Model for Quick parametric Analyses" Proceedings of the ASME 2017 International Technical Conference and Exhibition on Packaging and Integration of Electronic and Photonic Microsystems Inter-PaCK, pp. 1-8, 2017.

[3] C. Barbagallo, G. Cammarata, G. Malgioglio "Thermo-Mechanical Analysis of a Multi-chip Power Module" Proceedings of the ASME-ATI-UIT Conference on Thermal Energy Systems: Production, Storage, Utilization and the Environment, 17-20 May 2015, Napoli, Italy (2015)

[4] A. Zeanh *et al.*, "Thermomechanical modelling and reliability study of an IGBT module for an aeronautical application," *EuroSimE 2008 - International Conference on Thermal, Mechanical and Multi-Physics Simulation and Experiments in Microelectronics and Micro-Systems*, Freiburg im Breisgau, 2008, pp. 1-7

[5] A. Zéanh et al., "Lifetime and Reliability Assessment of AlN Substrates Used in Harsh Aeronautic Environments Power Switch Modules", Advanced Materials Research, Vol. 112, pp. 113-127, 2010

[6] R. Kraus, A. Castellazzi, "A Physics Based Compact Model of SiC Power MOSFETs". IEEE Transactions on Power Electronics, VOL 31, No.8, August 2016

[7] University of Colorado "Introduction to Finite element Analysis Chapter 30: Thermo Mechanical Effects" Department of Aerospace Engineering, University of Colorado.

[8] https://www.3ds.com/products-services/simulia/products/abaqus/

[9] https://www.ansys.com/products/systems/ansys-twin-builder