A Global Finite Element Model for Improving the Thermo-Mechanical Reliability of IBGTs Modules

Y. Tronel* and W. Fichtner**

*Integrated Systems, Swiss Federal Institute of Technology, Zürich
Technoparkstrasse 1, Zurich, CH-8005,tronel@iis.ee.ethz.ch
** Integrated Systems, Swiss Federal Institute of Technology, Zürich
Gloriastrasse 35, ETH Zentrum CH-8092,fw@iis.ee.ethz.ch

ABSTRACT

Insulated Gate Bipolar Transistors (IGBT's) are widely used in the automotive industry as high power current switches. They are currently being introduced into traction applications (locomotives, trams, metros, etc.) where high reliability is extremely important. Modern locomotives have a lifetime of about 30 years during which they should not fail. Due to the complexity of the modules as mechanical systems and as electric circuits, it is of prime importance to select and set up the most suitable models which allow to extract valuable data. Accordingly, this work describes the models and associated equations which appear to be necessary to carry out failure analysis. It includes a general stationary fully coupled electro-thermo-mechanical analysis which is actually part of the commercial release of the software SOLIDIS$_{ISE}$. More specific features tailored to IGBTs failure analysis are currently implemented. It comprises thermal transient effects and plasticity/damage based models for the stress analysis. Finally, a numerical example showing the damage distribution at a wire bond connection is presented.

Keywords: IBGTs, damage, pure aluminium, finite element model

INTRODUCTION

There are several physical phenomenon at the origin of IGBTs modules failure [1]:

- overheating
- dielectric breakdown
- thermomechanical stress between different materials
- dry and aqueous corrosion
- oxidation in dry air

However, the two dominant failure mechanisms of today's IGBT-packages are bond wire lift-off and solder fatigue. In this work, we will focus on the wire lift-off. The possible candidates at the origin of this lift-off are:

- Stress due to mismatch of thermal expansion coefficients at the interface between bond wire and die.
- Stress due to Joule's heating of the bond wires.
- Stress from the Lorentz-forces between neighbouring current conducting wires.

A priori, the model should include current transport and thermo-mechanical effects. Numerical tests show however that in most cases, the expansion coefficient mismatch contributes the most to the lift-off. Finally, the lift-off can be investigated by means of thermo-mechanical analysis. Simple analytical models [2] assuming steady state elastic behaviour of the material can relate the stress level at the interface as a function of the temperature excursion and the difference of expansion coefficients. An equivalent finite element simulation would give the same global behaviour with additional information concerning the localisation of the stress level along the interface between the two materials.
The examination of the numerical values of the stress shows that the elastic limit is reached for temperature excursion met in practice. It is then necessary to introduce plasticity into the model and accordingly switch from a thermal steady state analysis to a transient one.
From the practical point of view, the introduction of plasticity into a model involves a large amount of development and extend considerably the computation time for two similar configurations:

- the nature of the local plastic constitutive equations leads to a global non-linear system which cannot be solved directly by a standard non-linear solver. In addition, a set of local conditions resulting in local

non-linear systems must be solved to insure both the structural equilibrium and the consistency.

- due to highly non linear behaviour of stress-strain curves for plasticity, the simulation have to be split into several increments assumed to generate sufficient small incremental loads compatible with the practical resolution of the non-linear system described just above.

Plasticity is essentially an irreversible process. After the load is removed, a part of the stress (residual stress) and of the strain (plastic strain) have non zero values. However, at this point we have no information concerning the state of deterioration of the material. Accordingly, a damage variable may be added to the model in order to account for further failure or specific mode of fatigue or more simply to have an idea about the degradation of the material. A common framework called the Continuum Damage Mechanics [3] is ideally suited to described both the irreversibility of deformation (plasticity) and the evolution of the integrity of the material (damage). It should be noted that our model does not account for fracture analysis: cracks or voids are always supposed to be described by a continuous state variable called damage and never have geometric meaning.

STRESS ANALYSIS

The local form of mechanical equilibrium when dynamic effects are neglected is expressed by:

$$\frac{\partial \sigma_{ij}}{\partial x_j} + F_i = 0 \qquad (1)$$

where σ denotes the stress tensor and **F** is a local density force vector (which is the most often taken as 0).
The hypothesis of small displacements and strains in the simulations of IGBTs allows to write the displacement field **u** as the symmetric part of the strain tensor ε :

$$\varepsilon_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \qquad (2)$$

The boundary condition may be either of prescribed displacement or prescribed normal stress over suitable parts of the boundary.

The generic form of the corresponding variational principle is therefore:

$$\int_\Omega \sigma : \varepsilon^* \, d\omega + G = 0 \;\forall\, u^* \in V \qquad (3)$$

u^* is an admissible displacement field belonging to a suitable functional space V. The term G gathers all information concerning exterior load. It includes body force density as well as prescribed traction on parts of the boundary.

THERMODYNAMICS OF IRREVERSIBLE PROCESSES [3]

Within the frame of thermodynamics of irreversible processes, the derivation of constitutive equations and their evolution in time rely on the existence of two potentials known as the state potential and the potential of dissipation.

State potential

The state potential is usually taken as the free Helmoltz energy state function and it constitutes a rather general approach to a basic description of the problem. In the considered case the state potential Ψ is taken as:

$$\psi = \psi([\varepsilon - \varepsilon^p], T, D, r) \qquad (4)$$

which means a dependency on the elastic part of the strain tensor σ, the temperature T, the scalar damage variable D and a generic scalar hardening parameter r. At this level, the formulation assumes implicitly the isotropic behaviour of the material. The associated (dual) variables also called thermodynamic forces may be derived from eqn. 4. if ρ is the density of the material, we have:

$$\sigma = \rho \frac{\partial \psi}{\partial \varepsilon^e} \qquad (5)$$

which gives the expression of the stress tensor and

$$Y = \rho \frac{\partial \psi}{\partial D} \qquad (6)$$

where Y is the opposite of the strain energy density release rate.

Potential of dissipation

This second potential will give the kinetic constitutive relations which will describe the evolution of the system. It is assumed that the dissipation function depends on the dual variables and possibly on the state variables considered as parameters:

$$F(\sigma, R, Y; \varepsilon^e, r, D) \text{ (isothermal case)} \qquad (7)$$

One can derive from this potential both the plasticity constitutive equations and the kinetic damage evolution:

$$\begin{cases} \dot{\varepsilon}^p = \dot{\lambda}\dfrac{\partial F}{\partial \sigma} \\ \dot{r} = -\dot{\lambda}\dfrac{\partial F}{\partial R} \\ \dot{D} = \dot{\lambda}\dfrac{\partial F}{\partial Y} \end{cases} \tag{8}$$

The quantity $\dot{\lambda}$ is a scalar multiplier and its value depends on the specific model. It is important to note that it is the same scalar multiplier for each of the state variable. The relations 8 define the so-called generalised normality rule.

CONSTITUTIVE EQUATIONS

Elasto-plasticity

The total strain tensor is split into its elastic and plastic part as:

$$\varepsilon_{ij} = \varepsilon^e_{ij} + \varepsilon^p_{ij} \tag{9}$$

In the case of thermal coupling, a reversible thermal strain have to be added to the right hand side of eqn 9.

$$\varepsilon^{th} = \alpha \Delta T \tag{10}$$

The elastic part is given by the Hooke's law (in rate formulation):

$$\dot{\varepsilon}^e = K^{-1}\dot{\sigma} \tag{11}$$

where K is the 4th order elasticity tensor which reduces to the Young's modulus E and the Poisson's ration ν in the case of isotropy.
Plasticity is characterised by constraints on the stress state. A yield function f is defined which value is negative for elastic state and zero for plastic state. A positive value of f is forbidden for it has no physical meaning.
If the plastic yield function is assumed to obey the isotropic Von-Mises criterion with strain hardening and if S is the deviatoric part of σ, then the yield surface can be written as:

$$f = (S:S)^{1/2} - \sqrt{\frac{2}{3}}k(\bar{\varepsilon}^p)\bar{\sigma} = 0 \tag{12}$$

$k(\bar{\varepsilon}^p)$ is the hardening function and it depends only on the accumulated equivalent plastic strain. It is the non-linear part of the stress-strain curve obtained by unidimensional experimental tests. $\bar{\sigma}$ is the yield stress also measured by unidimensional tests.
Therefore the normality rule apply and the plastic strain rate is given by:

$$\dot{\varepsilon}^p = \dot{\lambda}\frac{\partial f}{\partial \sigma} \tag{13}$$

Finally the plastic consistency condition $\dot{f} = 0$ allows to calculate $\dot{\lambda}$ as a function of f and its partial derivatives.

Damage coupling

Full Coupling

According to the principle of equivalence first introduced by Lemaitre [3] the Hooke's law of elasticity is modified by damage as follows:

$$\dot{\varepsilon}^e = \frac{K^{-1}\dot{\sigma}}{1-D} \tag{14}$$

while the yield criterion is extended by replacing the stress by the effective stress defined as:

$$\tilde{\sigma} = \frac{\sigma}{1-D} \tag{15}$$

which gives a total yield function:

$$f = (S:S)^{1/2} - \sqrt{\frac{2}{3}}(1-D)k(\bar{\varepsilon}^p)\bar{\sigma} = 0 \tag{16}$$

Clearly, an increase of damage acts as softening while the function k is always increasing and harden the material.

The consistency condition is still written formally as $\dot{f} = 0$ but now includes the term corresponding to the partial derivative with respect to D.

Weak coupling

However a more simple implementation can relax this strong coupling by considering D in eqn. 16 as a fixed parameter when processing the numerical integration. The consistency condition is then similar as in the pure elasto-plastic case.

Kinetic damage law

The expression given in eqn. 8 concerning damage is rather general and we have restricted our kinetic laws to the one of the form described by eqn. 17 which is found to represent a wide class of damage mode (brittle damage and ductile damage).

$$\dot{D} = d(\bar{\varepsilon}, R_v) \tag{17}$$

where $\dot{\bar{\varepsilon}}$ is the equivalent strain rate and R_v is the triaxiality factor defined as:

$$R_v = \frac{2}{3}(1+\nu)+3(1-2\nu)\left(\frac{\sigma_H}{\sigma_{eq}}\right)^2 \quad (18)$$

σ_h and σ_{eq} are respectively the hydrostatic stress and the Von-Mises equivalent stress. An example will be shown in the numerical example section.

Numerical discretization [4,5,6]

Global equilibrium

The spatial discretization of eqn. 3 leads to a non-linear system for a given time station t_{n+1}

$$G(U_{n+1}) = 0 \quad (19)$$

where U_{n+1} is the nodal vector of the unknown displacement.

The problem is solved using a Newton method, which gives for iteration i:

$$G(U^i_{n+1}) + K^i_{n+1}(U^{i+1}_{n+1} - U^i_{n+1}) = 0 \quad (20)$$

The tangent stiffness matrix K^i_{n+1} is consistently computed with the local integration of the constitutive law described in the next section. It must use the consistent values of stress, strain and damage (i.e. f≤0).

Integration of constitutive equations

The algorithm used to integrate numerically the constitutive equation is an extension of the elastic predictor/return mapping algorithm widely used to integrate elasto-plastic equations. This process takes place at the local level and is in practice applied to each integration point.
The first step is called elastic-damage predictor and the following trail stress is computed:

$$\sigma^t = \sigma + (1-D)K\Delta\varepsilon \quad (21)$$

$\Delta\varepsilon$ is the strain derived from the resolution of eqn. 20.
If this trial stress does not satisfy the condition f≤0 then the trial stress is corrected in order to project it back onto the yield surface. By using the backward Euler scheme, we have to solve a non-linear system involving the increment of damage and the increment of plastic strain. The total stress and damage are therefore incrementally updated by means of eqn. 8. and 9.

As mentioned in the damage coupling section, the weak coupling of damage mat be used and in this case the local non-linear system has only the increment of plastic multiplier as unknown.

HEAT TRANSFER ANALYSIS

The transient heat transfer for solids or static fluids is described by the energy balance equation:

$$\rho c\frac{\partial T}{\partial t} - \nabla(\kappa\nabla T) - H = 0 \quad (22)$$

where ρ,c and κ denotes respectively the density, the specific heat and the thermal conductivity of the considered material. The associated boundary conditions are of two basic kinds:

- Dirichlet: prescribed temperature, contact with an infinite reservoir
- Neumann: constant flux and linear flux

It is possible to mix these two basic boundary conditions in a linear way.
The semi-discretized version of this equation, obtained by applying the Galerkin weighted residual method is the following:

$$C(T,t)\frac{dT(t)}{dt} + K(T,t)T = g(T,t) \quad (23)$$

Here the matrixes C and K are respectively the mass and conductivity matrix. The right hand side is a generic thermal load term including generic heat generation sources and the constant part of Neumann boundary conditions.
Finally, a time stepping scheme must be introduced to derive an algebraic finite system of equations. The selected method belongs to the Generalised Midpoint Scheme family and finally leads to the fully discretized system:

$$(C_\theta + \theta h K_\theta)T_{n+1} = [C_\theta - (1-\theta)hK_\theta]T_n + hg_\theta \quad (24)$$

The parameter θ lies in the interval [0,1] and means that the corresponding quantity have to be evaluated at time

$$t_\theta = (1-\theta)t_n + \theta t_{n+1} \quad (25)$$

when θ appears in subscript. For θ > 1/2, eqn. defines a consistent and unconditionally stable implicit scheme.
The explicit dependence of C and K on the time is seldom met in practice and finally, the only possible non-linearities are material property dependence on temperature. In this case eqn. 24 is linearized by approximating the temperature dependant quantities at the beginning of the time step.

It should be noted that the mass matrix is used in its lumped form to avoid numerical spurious oscillations in contradiction with the physical nature of the heat diffusion process. Typically, one can observe oscillations when applying a strong heat flux or a given temperature different of the initial temperature on some part of the boundary: cooling near the interface is seen when heating is expected. It should be also noted that numerical integration in time tends to hide this oscillatory behaviour when the time step is taken large enough.

NUMERICAL EXAMPLE

This example shows the localisation of damage at the interface between the wire and the substrate.
This wire is part of and IBGTs package. The geometry of the wire-bond connection if of wedge-wedge type and for the calculation, only a small part of the substrate is represented; the left part and the right part of the system are connected in the reality. The wire is made of commercially pure aluminium while the substrate is a molybdenum plate. The thermomechanical properties are summarised in table 1.

	Aluminium	Molybdenum
E [Gpa]	330	70
ν [Unity]	0.30	0.35
α [10^{-6} K^{-1}]	25	5
c [J/kg.K]	944	255
ρ [10^3 kg/m^3]	2.7	10.3
κ [W/mK]	238	140

Table1: material properties of Aluminium and Molybdenum

The yield stress for the aluminium is taken as 28 Mpa and the hardening coefficient is 1.5. Preliminary study in the elastic case shows that for the conditions of simulation, the Molybdenum never enters the plastic state.
The evolution of damage law is extracted from [7] and has the following form:

$$\dot{D} = \begin{cases} \dfrac{Dc R_v \dot{\bar{\varepsilon}}}{\varepsilon_R - \varepsilon_0} & \text{if } \bar{\varepsilon} \geq \varepsilon_0 / R_v \\ 0 \text{ if } \bar{\varepsilon} < \varepsilon_0 / R_v \end{cases} \qquad (26)$$

R_v is the triaxiality function introduced above while Dc and ε_r indicate that rupture is reached when $D=D_c$ and $\varepsilon=\varepsilon_R$. ε_0 is a threshold value. Numerical values are valid for a 99.5 pure aluminium which a little lower than the real aluminium used in IGBTs wires. Numerical values for the damage parameters are:

D_c=0.23, ε_R=0.25 and ε_0=0.03.

A temperature excursions of 50°C is applied to the molybdenum. It is an estimation of the overall heating generated by the underlying chip.

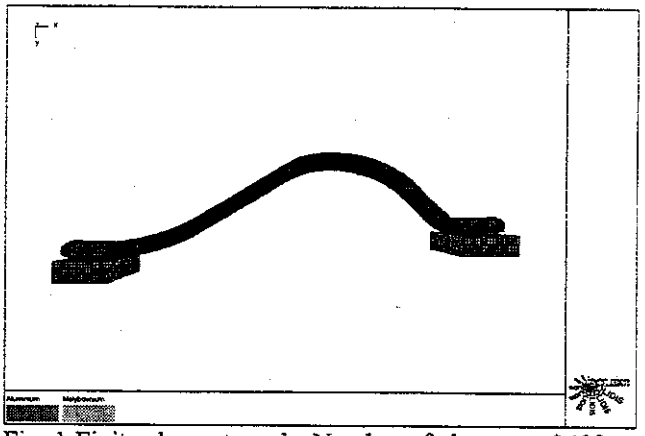

Fig. 1 Finite element mesh: Number of elements: 2498. Number of nodes: 3705.

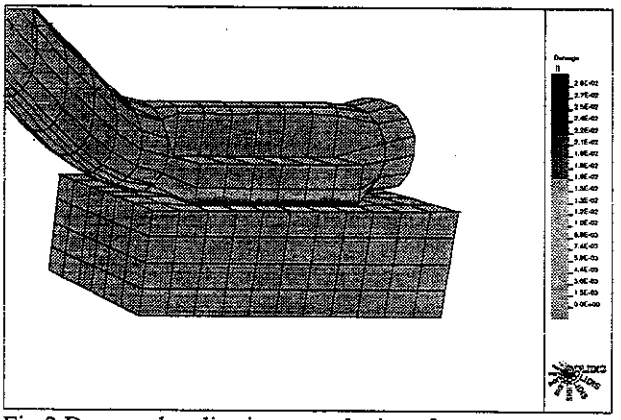

Fig.2 Damage localisation near the interface.

The finite element mesh used for the simulations is shown in Fig. 1. Fig. 2 shows a contour map of the damage parameter for the right connection. It can be seen that the damage is mainly concentrated in the plane of the interface between the wire and the substrate. Its numerical value is however much less than the critical damage D_c. Further investigations including cycling simulations are necessary to estimate the lift-off of the bond. It should be noted also that from the numerical point of view, damage localisation is often a mesh dependant problem. Therefore, simulations with finer meshes near the interface have to be carried out to validate the reliability aspects of these results.

CONCLUSION

In this work, thermo-mechanical models have been investigated to improve the reliability of IGBTs modules and specifically through their main mode of failure: wire lift-off. Plasticity and damage based models together with transient heat transfer have therefore been implemented into the finite element code SOLIDIS.ISE.

The first simulations showed and confirmed the stress and damage localisation observed qualitatively.

ACKNOWLEDGEMENTS

We would like to thank Mr. Alexander Stuck from ABB Corporate Research, Baden-Dätwill,Switzerland for the information provided concerning material data properties and for the operating conditions in IBGTs modules.
This work is part of the MINAST program funded by the Swiss National Science Foundation.

REFERENCES

1. Jeannote, D.A., Goldmann, L.W. and Howard, R.T. (1989). In: *Microelectronics Packaging Handbook*, R.R. Tummala and E.J. Rymaszewski (Eds.), Van Nostrand Reinhold, New-York, pp 225-359.
2. Dittmer, K.J., Poech, M.H., Wulff, F.W. and Krumm, M. (1995). *Failure Analysis of Aluminum Wire Bond in High Power IGBT Modules*. Mat. Res. Soc. Symp. proc, Vol **390**
3. Lemaitre, J. (1992) *A Course on Damage Mechanics*. Springer-Verlad Eds.
4. Benallal, A., Billardon, R. and Doghri, I. (1988). *An Integration Algorithm and the Coresponding Tangent Operator for Fully Coupled Elastoplastic and Damage Equations*. Comm. Appl. Num. Meth. **4**,pp 731-740.
5. Simo, J.C. and Ju, J.W. (1987). *Strain and Stress Based Continuum Damage Models*. Int, J, Solids Structures, **23**, pp 841-869.
6. Doghri, (1995) I. *Numerical Implementation and Analysis of a class of Metal Plasticity Models Coupled with Ductile Damage*. In. J. Numer. Meth. Eng **39**, p 3408-3431
7. Bontcheva, N. and Inakiv, R. (1991). *Numerical Investigation of the Damage Process in Metal Forming*. Eng Fracture Mech. **40**, pp 387-393.