Submodeling technique for assessment and numerical prediction of solder jointsfailuresin mechatronic devices

H. HAMDANI[12], B. RADI[1], A. ELHAMI[2]

1. Laboratoire Ingénierie Mécanique, Management Industriel Et Innovation (LIMMII) - *Faculté desSciences et Techniques Settat- Maroc.*

2. Laboratoire d'Optimisation et Fiabilité en Mécanique des Structures (LOFIMS)*INSA Rouen* ; *Avenue de l'Université, 76800 Saint-Étienne-du-Rouvray - France*

Abstract

In mechatronic devices, the fatigue of the solder joints is a mechanism leading to rupture and which is generated during the severe stresses encountered in operational environments. These thermal or mechanical stresses are variable over time (temperature variation, vibrations, and repeated shocks); the solder joints are thus subjected to fatigue.However, it is necessary to understand the damage of these solder joints under cyclic loading, through the determination of the stress and deformation levels.

Finite element methods are broadly used tothis objective. To realize this modeling, it is necessaryto choose anadequate geometric modeling,and to know sufficiently the mechanical materials behavior.

Many of the modeling techniques were used based on the global model of the studied mechatronic device, which requires a very high calculation time

In order to reduce the computational time, a submodel of the worst case solder joint is developed. This submodelis based on the geometry of the full model.

This submodeling technique allows us to model precisely the inelastic strain of the solder joints. It consists in describing the worst case solder joint with fine mesh elements and in taking into account material nonlinearities. The solder material is assumed to have a rate dependent plasticity.

Keywords: Mechatronic, Submodeling technique, Submodel, FEM, Plastic deformations.

1. Introduction

In embedded electronic components, one of the most commonlyobserved failures occurs when the cycles of thermal load lead toinelastic irreversible deformations causing fatigue cracks in solder joints.

Finite element methods (FEM) are broadly used topredict numerically the thermo-mechanical behavior ofsolder joint. Several authors [1] havedeveloped a sophisticated model based on the FEM methodand on the failure mechanisms of a solder joint subjectedto thermal loading.

The main stage of this work consists in developing anumerical model to simulate the thermo-mechanical behaviorof the solder joint. The ANSYS Finite Element Analysis software [2] is used to solve the heat transferproblem and to compute the strain response due tothe thermal variation. The material nonlinearity due to theplastic behavior of the solder joint is taken into account.

Each numerical model represents in a simplified way a defined problem. For thestructures computation, three elements are essential:

- The geometry of the model
- Modeling of materials behavior
- Solicitation and boundary conditions,

These elements must be represented in a way that is adapted to the needs, throughcompromise between the most faithful representation of the reality and the computation time of a simulation...

2. Finite element analysis

2.1. finite element model

The mechatronic package under investigation is a 256-pin plastic quad flat package with 0.5 mm pitch gull-wingleads. Figure 1 shows a perspective view of the meshed global model [1]. The model is based on thegeometry of thePrinted Circuit Board (PCB), the component-molding compound, its solder interconnects, and gull-wing leads.

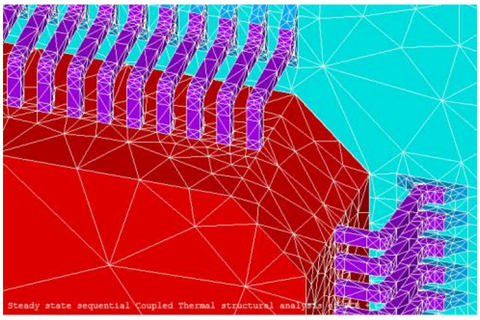

Figure 1: Global finite element model of the 256-PQFP

To avoid tedious calculation and expensive computing resourcesa submodel of the worst-case solder joint is developed. This submodeling technique is based on the geometry of the full model as shown in Figure 1.

This submodeling technique [2] allows us to model precisely the inelastic strain of the solder joints. It consists in describing the worst case solder joint with fine mesh elements and in taking into account material nonlinearities. The solder material is assumed to have a rate dependent plasticity.The symmetrical boundary conditions were applied along the left edge of themodel.This finite element submodelis solved for one thermal cycle as shown in Figure 3.

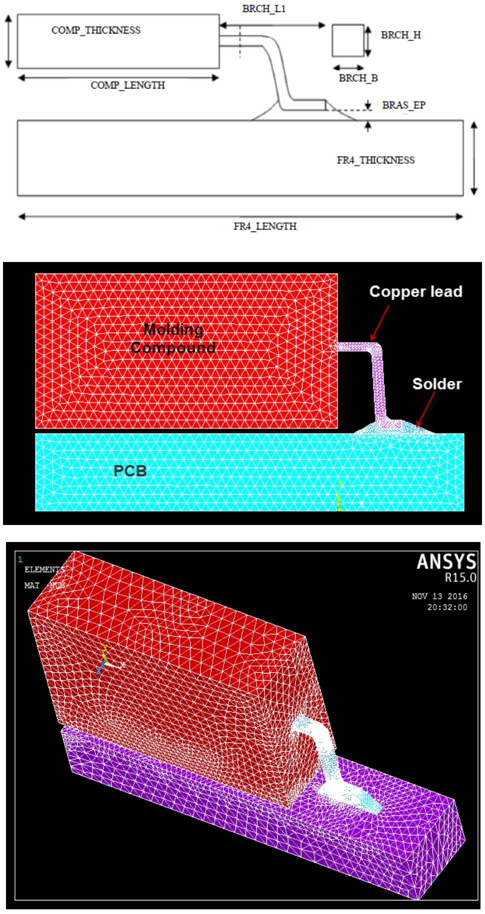

Figure 2: Finite element meshes, and dimensions of the numerical model

2.2. Materials behavior Modeling

For a given structure, there are several models for the materials behavior in terms ofthe solicitation encountered or the level of detail sought. In the case of use in small deformations, the Young's modulus of the material may suffice. For the use with large deformations, the same experimental behavior can be represented by multilinear behavior, or by a non-linear behavior according to the needs.

In the case of mechatronic devices, Failures are localized in solder joint. In addition, deformations may be macroscopically plastic.

For the thermomechanical analysis, using a submodel, the molding compound, FR-4, and the PCB materials are assumedisotropic and linear (see Table 2). The development of plastic deformations in the material of the solder joint(SnAgCu),Depends on the loading velocity. Servalauthors [3] have studied the response of solder joints (SnAgCu) and eutectics (SnPb) and have proposed equations modeling this response.One of the equations developed is Anand's model [Anand 1982], which The has been developed to describe the deformation of metals at high temperature, including, the dependence of deformation velocity. This is one reason why many studies have used the Anand model in describing SAC alloys. Moreover, this model is available directly under the ANSYS finite element modeling software, which may explain its success.

In addition to differentiating elastic and inelastic deformations, Anand's model combines creep and instantaneous plastic deformation into a single term, expressed through a state variable.

In brief, Anand’s model consists of a flow equation governing the inelastic strain rate$\dot{\varepsilon}_p$:

$$\dot{\varepsilon}_p = A \exp\left(\frac{-Q}{RT}\right)\left[\sinh\left(\xi\frac{\sigma}{s}\right)\right]^{1/m} \qquad (1)$$

This equation is similar to that of Garofalo (Sinh), commonly used to describe stationary creep [4], but in Anand’s model, it is added a state variable, defined as resistance to deformation: **s**. This state variable depends on the temperature and takes into account the inelastic deformation rate. Two evolution equations governing the internal statevariable s and its saturation value s^*

$$\dot{s} = \left\{h_0\left|1-\frac{s}{s^*}\right|^a \times sign\left(1-\frac{s}{s^*}\right)\right\} \times \dot{\varepsilon}_p\ ;\ a > 1 \qquad (2)$$

$$s^* = \hat{s}\left[\frac{\dot{\varepsilon}_p}{A} exp\left(\frac{Q}{RT}\right)\right]^n \qquad (3)$$

The nine parameters related to this equations system are presented in the following table:

Paramètre	*Signification*
$S_0(MPa)$	*Initial value of deformation resistance*
Q /R(K)	*Q : activation energy, R : universalgas constant*
$A(s^{-1})$	*pre-exponential factor.*
ξ	*Stress multiplier.*
m	*strain rate sensitivity of stress.*
$h_0(MPa)$	*hardening/softening constant.*
$\hat{s}(MPa)$	*coefficient for deformation resistance saturation value.*
n	*Strain rate sensitivity of saturation (deformation resistance) value.*
a	*strain rate sensitivity of hardening or softening.*

Tableau 1: Anand Model Parameters

Wang and al. 2001 [5] proposed a unified framework for the viscoplasticbehavior of SnAgCu solder materials, which are referred to the Anand constitutive equations. Thematerial parameters of the Anand model for SnAgCu solders are obtainedfrom experimental results and by theseparated elasto-plasto-creep constitutive relations. These material parameters are shown in Table 3 as given by Wanget al. 2001.

Propriétés des matériaux	SAC305	FR4	Résine EPOXY	Cu
Module de Young (GPa)	51.3	17	17	115
Coefficient de poisson	0.3	0.3	0.2	0.31
Densité (Kg/m^3)	740	180	180	8890
CTE (μm/K)	20	18	22	17
Module de cisaillement	19	2.4	7.4	44

Table 2: Material Properties

Paramètre du matériau	SAC305
$A \quad (s^{-1})$	2.23 10^4
Q /R (K)	8900
ξ (sans)	6
m (sans)	0.181
$\hat{s}$ (sans)	73.81
n(sans)	0.018
h_0(MPa)	3321.15
a (sans)	1.82
S_0(MPa)	39.09

Table 3: Parameters of the Anand model

2.3. Thermal loading

To predict numerically the thermo-mechanical behavior ofsolder jointmechatronics devices, aging testsare systematically carried out on test cards. The stresses are simulated by temperature cycles (see Figure 3). These cycles are applied to accelerate the temperature variations experienced by the cards in their operating environment [3].

Figure 3 shows the temperature time history used in the analysis, thus we took the loading situation effectively one thermal cycle of the temperature range from -55 to 125 °C during 60 min

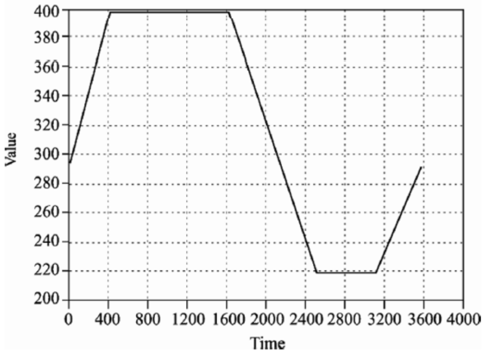

Figure 3: Thermal load cycle

3. Numerical results

The distribution the normal plastic strains of solder joint at last stepis shown in Fig 4. It is observed that the plastic strain is occurred at the solder joint and the extreme plastic strain values are located at the corners of the bond pad. Finite element analysis results indicate that the failure would start from the corner of the interface between solder joint and copper lead.

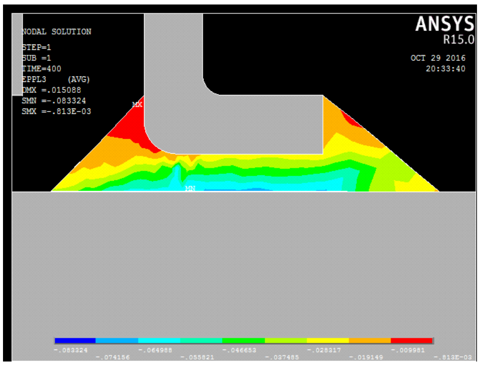

Figure 4: normal Plastic strain distribution in the solder joint

4. Conclusion

In this study, a nonlinear thermomechanical finite element analysis based in submodeling technique is used to predict numerically the thermo-mechanical behavior ofsolder jointmechatronics devices.This approachallows us to model precisely the inelastic strain of the solder joints, to take into account materialnonlinearities, to describe the worst case solder joint with fine mesh elements and avoid tedious calculation and expensive computing resources.

A possible extension of this work is to investigate the effect ofthe uncertainties arising from the random nature of the temperature fluctuations caused by power transients andthermal environment changes, the thermal expansion mismatch of the different materials of the assembly, the material properties and the fabrication process.

Références

[1] ANSYS Guide (2015)*ANSYS Structural Analysis Guide.*

[2] F. Feustel, S. Wiese, and E. Meusel, *"Time-dependent material modeling for finite element analyses of flip chips,"* in Proceedings of the 50th Electronic Components and Technology Conference, 2000., pp. 1548-1553, 2000.

[3] G. Z. WANG, Z. N. CHENG, K. BECKER, J. WILDE. "*Applying Anand Model to Represent the Viscoplastic Deformation Behavior of Solder Alloys"*.Journal of Electronic Packaging, vol.123, Issue 3, pp. 247-253.Septembre 2001,

[4] M. GRIEU. "*Étude de la fatigue des joints brasés de composants électroniques soumis à des sollicitations thermomécaniques vibratoires et combinées."* École nationale supérieure des mines de Paris, Thèse. France. décembre 2010.

[5] Y.Aoues, A.Makhloufi, A.El-Hami "*Optimizing reliability of electronic systems In Embedded Mechatronic Systems 2"*. Elsevier, pp 181–215.2015