ANALYSIS AND PREDICTION OF RELIABILITY OF SOLDER JOINTS

Václav Novotný

Doctoral Degree Programme (1), FEEC BUT

E-mail: xnovot11@stud.feec.vutbr.cz

Supervised by: Josef Šandera

E-mail: sandera@feec.vutbr.cz

Abstract: This article deals with the sphere of solder joints reliability and narrower focus is using simulation software ANSYS and fatigue models for estimate reliabity of solder joints. The text describes the introduction to the issue of the reliability of soldered connections and important factors with their influences. Then are described some options for prediction of reliability. This includes the issue of fatigue models and design of simulation in simulating software ANSYS. In conclusion, there are described output of simulations and options for calculation by fatigue models.

Keywords: EEICT, ANSYS, Reliability, Simulation, Solder joints, fatigue models,

1. INTRODUCTION

The reliability of solder joints is an area of microelectronic manufactory and its importance tends to rise. There is a large number of factors, which have an influence on proper function of electronic devices. The solder joint is part of a heterogenous dynamic system and it's necessary to examine all parts of this system, when there is the requirement for high reliability. The most important component of system is material of solder joint, because its share in the system is about 75% and this is a big part. Material of solder has also other interactions with contact areas of the substrate (20% of the system) and with terminal (5%).

These system components have some influences on each other and they influence together the reliability. It is also possible to divide the factors affecting the soldered joint according to the place of origin. This means that there are external and internal factors. External influences are all environmental conditions around the soldered connections. This may be the ambient temperature and its changes, vibration and humidity. Then there are some internal influences like mechanical deformation, because different materials have different coefficients of thermal expansion.

Mechanical deformation and subsequent stress in joint leads to fatigue of material and it's the cause of worse electrical and mechanical characteristics and then it's start of origin of cracks and rifts. Therefore, it is necessary to have an overview of the actions in the soldered joint. Standard IPC-9701 divides electronic devices to several groups according to several properties. The most important feature of the standard is percentage of errors. Electronic devices for end customer have greater tolerance of errors, but there are other areas like health or aeronautics, where reliability is first unreservedly. Therefore it's important to test or predict reliability. It's possible to test electronic devices, but it is need to do some functional samples and functional samples are very expensive. Here are irreplaceable computer simulations and theoretical models of fatigue for predict reliability. The text shows the basic prediction of reliability using simulation software ANSYS and fatigue models.

2. FATIGUE MODELS AND ANALYSIS OF RELIABILITY

Fatigue models are the key components in the estimation of reliability and their aim is to estimate or calculate the number of cycles to fracture of solder joint. These fatigue models always linked to a specific physical mechanism that operates in the solder joint under the effect of external conditions. Therefore, it is possible fatigue models divide into several groups by majority of mechanism which operates at the moment. Fatigue models can be divided into three basic groups for clarity. The first group is fatigue models based on plastic and elastic deformation and typical model is very famous Coffin-Manson fatigue model, which can describe low-cycle fatigue. Low-cycle fatigue is defined as fatigue, which is caused by low number of cycles (100-1000) and high amplitude of conditions, for example temperature or vibrations. It is also very well known Engelmaier-Wild model and is an improvement of Coffin-Manson.

The second group is based on creep deformation. Conditions for creep deformation are high temperature and constant load and typical model is Knecht-Fox model, where the main parameters are microstructure of solder and diffusion of grain. The biggest group is fatigue models based on energy and changes of energy. The main parameter is the increase or decrease of energy absorbed during one cycle in solder joint and it is assumed that the energy absorbed in solder joint has some deformation effects and causes damages. Absorbed energy may be expressed in pressure of each cycle or in units of work. Typical model based on energy is Darveaux fatigue model and has three parts. The first part is to calculate the time until fracture of the Solder joint and then calculate the rate of the growth rate of fracture. The number of cycles to fracture is the sum of the first two parts. The principle is experimentally obtain the maximum value of the deformation pressure for the solder and then put into the equation, wherein is included the maximum pressure for our tested geometry or model of solder joint. It's possible to achieve some values of deformation pressure for our geometry by simulation in ANSYS.

3. ANALYSIS IN ANSYS

ANSYS is simulation software based on finite element method and finite element method is the application of a network of nodes on the proposed model, which is the subject of analysis. ANSYS software is distributed in two versions, namely ANSYS and ANSYS Workbench Classic, with the ANSYS Workbench offers a graphical extension to the standard ANSYS. ANSYS Workbench includes several modules and complete simulation is comprised of the following modules. At the beginning, it is important to define material data for each part of complete geometry. Furthermore, it is necessary to draw a model. When we have a model, we can define the type of analysis and relevant conditions, which we expect in simulations. We have several ways to create the geometry and we can use them.

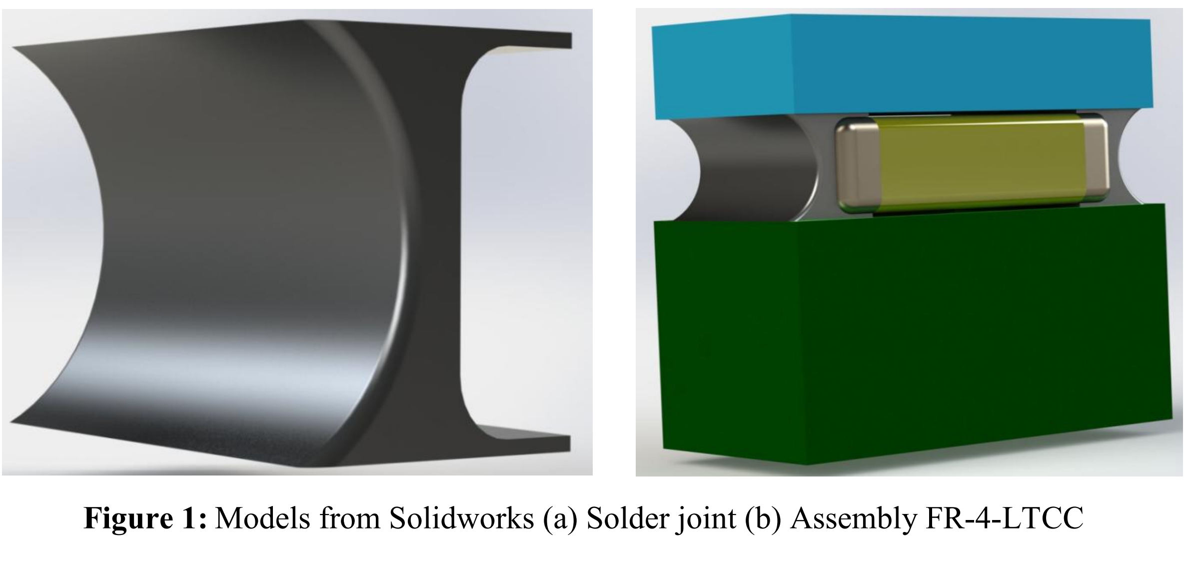

Figure 1: Models from Solidworks (a) Solder joint (b) Assembly FR-4-LTCC

First option is use the DesignModeler, which is integral component of ANSYS. This module behaves as classic and typical 3D CAD system, where it is possible to draw a 2D sketch and the sketch then pull into the third dimension. The main representative of these systems is a program Solidworks and great advantage of ANSYS and DesignModeler is the ability to import geometry from other CAD systems like as SolidWorks. This is useful, especially when its need to create some complex and difficult geometry, then it is better to use drawing software, which is adapted for 3D drawing. The complete geometry for simulation consists five parts. These parts are basic substrate FR-4, Solder joints, SMD Capacitor 0805, LTCC substrate and copper solder pads. Geometry is imported in DesignModeler from Solidworks.

Table 1: Mechanical and physical properties of materials

Material	density [g/cm^3]	Young's modulus [GPa]		Poisson's ratio [-]	
FR-4	2,1	x	25,8	xy	0,177
		y	21,9	xz	0,171
		z	25,8	yz	0,177
96% Al_2O_3	3,67	300		0,21	
Heralock 2000	2,9±0,5	240		0,32	
SAC305	7,5	51		0,36	
Cooper	8,93	76		0,35	
Solder mask	-	6,9		0,35	
C0805 BaTiO3	6,02	67		0,32	

You can see important parameters of material in table (1), where are three basic values. These values are density, Young's modulus and Poisson's ratio and they define physical and mechanical properties of materials. Furthermore, the simulation contains thermal properties, which includes coefficient of thermal expansion, thermal conductivity and specific heat. Its described in table (2).

Table 2: Thermal properties of materials

Material	CTE [ppm/°C]		thermal conductivity [W/mK]	specific heat [J/gK]
FR-4	x	14	0,4	0,12
	y	14		
	z	73		
LTCC	6,1		3	-
SAC305	21,6		64	0,23

Another part of simulation continues in Mechanical, which is modul intented for adjustment of conditions of simulations. There is a large number of options like set of types of contacts between different parts, applying the finite element mesh and applying conditions for simulations. The first important step is set correctly mesh. Mesh contains nodes, where each node is the point for calculation. ANSYS defines basic mesh and then it's possible to adjust this mesh as required. This step is called refinement and it's important in case of difficult geometries. There are two ways for application of refinement. The first option is to apply refinement to the whole geometry and second option is to apply refinement only on involved part, for example some camber. Analysis of the solder joint is composed of two parts. First it is necessary to perform transient thermal analysis, which defines transmittance of heat through assembly and it's depending on time and temperature conditions. For this analysis are important coefficient of thermal expansion, thermal conductivity and specific heat.

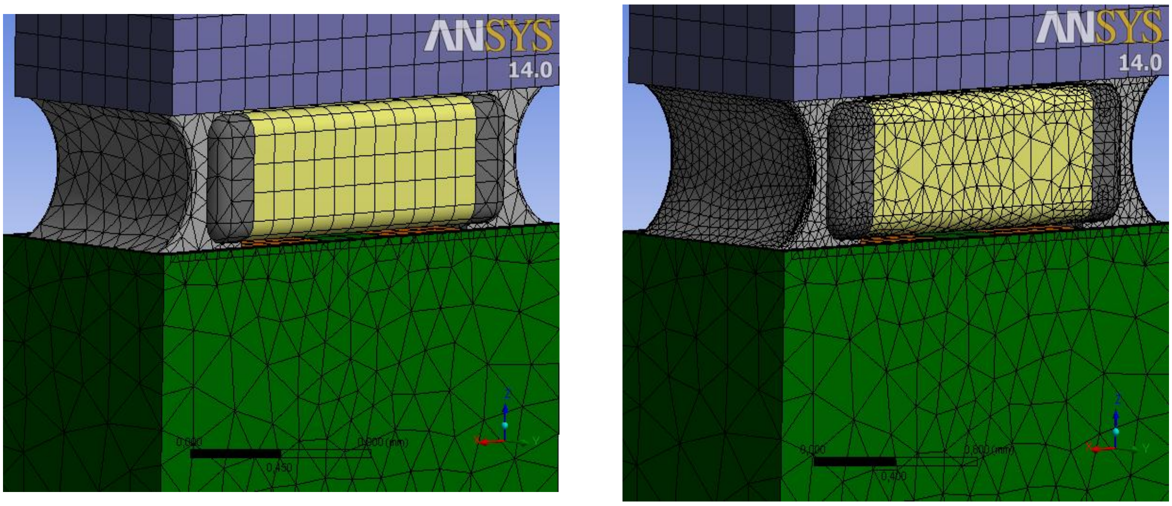

Figure 2: Mesh of nodes in ANSYS (a) Basic degree of mesh (b) Improved mesh

Further conditions are set for simulation of environment in temperature chamber. These are cyclic temperature changes from -40 °C to 125 °C. Results from thermal simulations are then imported to static structure analysis and are one of conditions for this type of analysis. Further static structure analysis use mechanical and physical properties from table (1). It's density, young's modul and Poisson coefficient. The output of the analysis is strain energy and equivalent stress in solder joint. For estimation of reliability is difficult increase of equivalent stress during one cycle and it's used in Darveaux equation or in some other equation. Results of simulations show that highest equivalent stress is in area around joint of LTCC and Solder SAC305. It's therefore, that there is the biggest diference in coefficients of thermal expansion.

This is shown in the corresponding table (2). Further also has a big influence thermal conductivity of LTCC, which that value is bigger than FR-4. ANSYS allows to measure a large number of variables and equivalent stress is selected because it's very complex variable. Equivalent stress is also known as Von-Mises stress and may be characterized as three-dimensional stress, because all materials in assembly are anisotropic and deformation is different in each direction. Equation of Von-Mises stress is combination of six segments of stress and summarized into one. Von-Mises stress can be displayed for the whole assembly and it's possible to get a general overview, but for calculation of estimate of reliability is difficult to display detail of solder joint only.

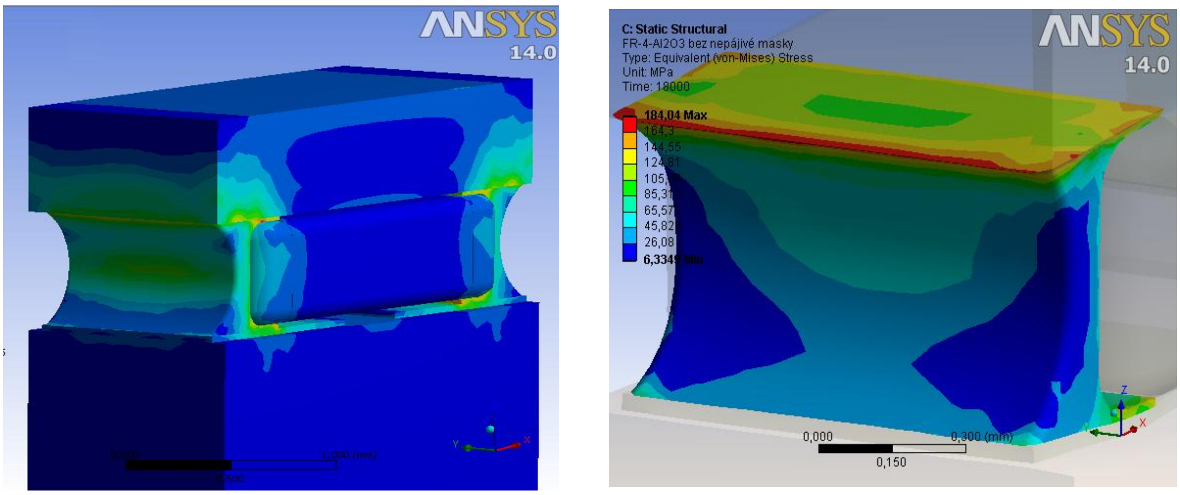

Figure 3: Von-Mises stress in ANSYS (a) Complete assembly (b) Detail of solder joint

Calculation of reliability is conjunction of analysis in ANSYS and use of some fatigue model for estimate of reliability. Two types of fatigue models are included in calculation. The first model is a part of ANSYS and he defines the viscoplastic behavior of solder. The model is composed from several equations and these equations contain parameters, which are obtained experimentally for a certain type of solder. The second model is designed to calculate the number of cycles to failure of solder joint. For this purpose is selected Darveaux model, which is composed from four parts. First component of Darveaux equation is value of increase equivalent stress during one thermal cycle in chamber and it's in MPa. Another part is to calculate the number of cycles to the start of crack in solder joint:

$$N_o = C_1 \Delta W^{C_2} \qquad [-, MPa, \frac{cykl}{MPa}, MPa, -] \qquad (1)$$

Equation (1) consists three parts. The left side represents number of cycles to start of crack and right side of equation is multiple of material constant of solder and increase of equivalent stress within one cycle, which is aquired from ANSYS. Furthermore, it is necessary to calculate the rate of growth of crack and finite number of cycles to crack is combination of speed to start of crack and rate of growth of crack. It is shown in equation:

$$N_f = N_0 + \frac{a}{\frac{da}{dN}} \qquad [-, -, mm, -] \qquad (2)$$

4. CONCLUSION

Estimating the reliability of soldered joints is very complex and it is possible to say that it is a multidisciplinary, because there intersect electronics, mechanical engineering and physics. Prediction of life service of solder joints has a great options and it increases with the development of computer technology and simulation programs like ANSYS. Also is available a large amount of fatigue models. At this moment it is very advantageous combination to use ANSYS in cooperation with some fatigue model. This text shows the use of simulation of solder joint and calculation reliability using Darveaux fatigue model and procedures that lead to successful simulation.

ACKNOWLEDGEMENT

Funding for this research was obtained through grant project BD 18409001 FEKT-S-11-5 „Research for excellent technologies for 3D packaging and interconnection" supported by Ministry of Education, Youth and Sports of the Czech Republic.

REFERENCES

[1] LI, S., HUANG, Z., WANG, J., GAO, S. Temperature Cycling Analysis of Lead-Free Solder Joints in Electronic Packaging. [cit. 2015-02-03]. Dostupný z www: http://www.simulia.com/forms/world/pdf2007/Li.pdf

[2] ENGELMAIER, W. *Surface Mount Solder Joint Long-term Reliability: Design, Testing, Prediction. Soldering.* 1989, vol. 1, issue 1. DOI: http://dx.doi.org/10.1108/eb037660.

[3] LEE, W.W, L.T NGUYEN a G.S SELVADURAY. *Solder joint fatigue models: review and applicability to chip scale packages: review and applicability to chip scale packages*.

[4] BATH, Jasbir. *Lead-free soldering*. New York: Springer, xiv, 299 p. ISBN 9780387684222

[5] ŠVECOVÁ, O, NOVOTNÝ, V. *The Use Of Creep-Fatigue Models For Solder Joints Reliability Prediction.* SECOND FORUM OF YOUNG RESEARCHERS In the framework of International Forum „Education Quality 2010". Izhevsk, Russia: Publishing House of IS-TU, 2010. s. 406-411. ISBN: 978-5-7526-0442-3.