第 30 卷 第 11 期 焊 接 学 报 Vol. 30 No. 11
2009 年 11 月 TRANSACTIONS OF THE CHINA WELDING INSTITUTION November 2009

球栅阵列封装焊点寿命预测的综合方法

陈 颖, 康 锐
(北京航空航天大学 工程系统工程系, 北京 100191)

陈 颖

摘 要: 为了实现球栅阵列(BGA)封装焊点寿命的快速工程估算, 建立了焊点的简化应力分布解析模型, 利用蠕变寿命预测模型求解得到其热疲劳寿命. 在需要精确预计焊点寿命时, 建立了 BGA 封装焊点的三维有限元模型, 利用 ANSYS 中的 Anand 方程得到焊点应力分布, 利用 Darveaux 寿命预测模型预测焊点热疲劳寿命. 采用 ANSYS 的二次开发功能将解析和有限元方法综合在主程序中. 不论是快速估算还是精确预计, 用户只需输入封装及焊点的尺寸、材料参数以及温度循环参数即可得到焊点寿命. 结果表明, 解析模型对参数变化更敏感, 有限元的结果则较平稳.

关键词: 球栅阵列封装; 焊点; 解析法; 有限元法; 寿命预测
中图分类号: TN305.94 文献标识码: A 文章编号: 0253-360X(2009)11-0105-04

0 序 言

在电子封装结构中, 焊点起着电连接、芯片与基板间的刚性机械连接及散热的三重作用, 其可靠性关系到芯片乃至整个电路的正常运行. 集成电路封装行业长期关注的焦点就在于寻找一种能够精确预计焊点疲劳失效的方法, 这种方法能够避免耗时、高成本的试验.

目前, 通过失效物理建模来预测焊点寿命已成为研究的焦点. 为了得到焊点的寿命, 首先需建立材料本构模型并求解焊点应力应变场, 这个过程可以通过解析方法或有限元方法来实现. 为了更加精确地描述焊点力学行为, 通常在求解焊点应力应变场时要使用粘塑性本构关系. 这类本构模型是高度非线性, 且与时间相关的, 求解起来非常复杂. 有限元法在求解复杂方程方面有更多的优势. 目前有两种商用有限元软件提供了统一的粘塑性本构关系模型, ABAQUS 的 UMAT(用户材料子程序)模型和 ANSYS 的 Anand 模型[1], 其中后者的应用更加广泛. 利用 Anand 本构模型, 能够较精确地计算出焊点内部三维应力应变场. 有限元方法也存在一些缺陷, 例如每次建模只能针对一种几何尺寸和载荷状态, 要改变尺寸和材料参数需要重新建模, 计算灵活性受到了限制. 另外对不熟悉该软件的人来说, 难度较大.

收稿日期: 2008-03-20

解析方法在工程中主要用于产品设计阶段的估算和快速优化. 与有限元法相比较, 解析法简单、快捷、容易操作. 但由于对材料本购关系描述的简化, 解析法的结果往往精度较差. 例如 Yamada[2,3] 研究了表面贴装焊点应力应变场的解析分布, 焊点材料为弹性. Heinrich 等人[4,5] 研究了焊点材料为弹性、粘弹性的球栅阵列(BGA)封装焊点的最大位移, 其结果可以作为焊点应力应变场求解的边界条件. Chen 等人[6] 给出了多层封装焊点应力应变计算结果, 但是焊点也假设为弹性材料. Sundararaja 等人[7] 给出了焊点材料为粘塑性时焊点裂纹扩展模型, 采用的本购方程为弹性、塑性、蠕变分离的本购方程, 得出的结果比较合理.

综上所述, 在选择了适当的本构模型和寿命预测模型后, 使用解析法和有限元法进行焊点寿命的求解各有长短, 如果能结合二者的长处, 避其短处, 形成解析法和有限元法相结合的焊点寿命预测手段, 在工程中将会有更广阔的应用前景.

文中介绍了一种综合的 BGA(球栅阵列)焊点寿命预测方法, 介绍其中解析模块的原理以及有限元模块的集成和数据传输, 最后以一个实例说明软件的使用.

1 解析模块原理

1.1 焊点应力场求解

为了使解析模型能够预计焊点与时间相关的力

学行为,对模型进行了如下的假设:(1)封装中所有焊点的材料和形状都是相同的.(2)芯片层、焊点层、与 PCB(印刷线路板)板(底板层)可简化为图1 所示的结构.(3)焊点中的剪切应力只沿着一个方向有变化,即半径方向,而在焊点层高度方向是均一的.(4)芯片和基板为各向同性线弹性体,忽略芯片端部的弯矩,假设正应力为零.(5)焊点的材料为粘塑性各向同性.

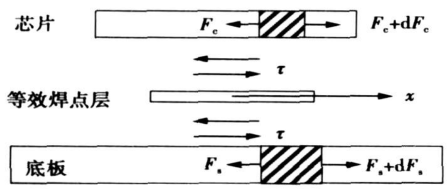

图 1 BGA 焊点简化受力模型

Fig. 1 Simplified force model of BGA solder joint

图1 为焊点的简化受力模型,包括芯片层、等效焊点层、底板层三个部分.其中 F_c, F_s 分别为芯片层、底板层受力;τ 为焊点层所受的剪切应力.根据文献[2,3,7],列出焊点求解的力学方程如下.

x 方向力的平衡方程为

$$\left.\begin{aligned}\frac{dF_c}{dx}-\tau=0\\ \frac{dF_s}{dx}+\tau=0\end{aligned}\right\} \tag{1}$$

芯片变形方程为

$$\left.\begin{aligned}\frac{du_c}{dx}=\frac{(1-\mu_c{}^2)F_c}{E_c t_c}\\ \frac{du_s}{dx}=\frac{(1-\mu_s{}^2)F_s}{E_s t_s}\end{aligned}\right\} \tag{2}$$

对于在剪力作用下的焊点,应力与位移的关系为

$$\gamma=\frac{u_c-u_s}{t_j} \tag{3}$$

式中:u_c, u_s 分别为芯片层、底板层位移;E_c, E_s 分别为芯片层、底板层弹性模量;μ_c, μ_s 分别为芯片层、底板层泊松比;t_c, t_s, t_j 为芯片层、底板层、焊点层的厚度;γ 为焊点层的应变.

焊点层的粘塑性应力应变关系满足 Anand 本构方程为

$$\frac{d\gamma}{dt}=A[\sinh(\xi\frac{\tau}{s})]^{\frac{1}{m}}\cdot e^{-\frac{Q}{RT}} \tag{4}$$

$$\dot{s}=\left\{h_0\left|1-\frac{s}{s^*}\right|^{\alpha}\cdot\mathrm{sign}(1-\frac{s}{s^*})\right\}\cdot\frac{d\gamma}{dt} \tag{5}$$

$$s^*=s\left[\frac{1}{A}\frac{d\gamma}{dt}e^{-\frac{Q}{RT}}\right]^n \tag{6}$$

式中:A 为指数前因子;ξ 为应力乘子;s 为变形抗力;m 为应力对应变速率敏感系数;Q 为激活能;R 为气体常数;T 为温度;s^* 为设定的中间量;h_0 为硬化或软化常数;α 为应变速率对硬化或软化的敏感系数;s 为变形抗力饱和值系数;n 为应变速率对饱和值的敏感系数.

总体应变为弹性与粘塑性应变之和,表示为

$$\gamma=\frac{\tau}{G}+\int_0^t A[\sinh(\xi\frac{\tau}{s})]^{\frac{1}{m}}\cdot e^{-\frac{Q}{RT}}dt \tag{7}$$

式中:G 为焊点材料的剪切模量.

边界条件:(1)芯片中心的剪应力为零,即 $\tau(0)=0$.(2)温度循环下为

$$\frac{(1-\mu_c^2)F_c l}{E_c t_c}-\frac{(1-\mu_s^2)F_s l}{E_s t_s}=(\alpha_s-\alpha_c)\Delta Tl+(u_c-u_s) \tag{8}$$

式中:l 为焊点的半径;α_c, α_s 分别为芯片层、底板层的线膨胀系数.

联立式(1)~式(4),式(7)得

$$\frac{\partial^2}{\partial x^2}\left(\frac{\tau}{G}+\int_0^t A[\sinh(\xi\frac{\tau}{s})]^{\frac{1}{m}}\cdot e^{-\frac{Q}{RT}}dt\right)=\left(\frac{1}{E_s t_s}+\frac{1}{E_c t_c}\right)\frac{\tau}{t_j} \tag{9}$$

最终要求解的是由式(5),式(6),式(9)联立的方程组,且满足边界条件.该方程组通过线性化的方法整理为线性常微分方程组,并通过 Runge-kutta 数值方法求得 $\tau(x,t)$和 $s(x,t)$.最后代入式(4)得到焊点层的应变数值分布.

1.2 焊点寿命预测模型

选用了文献[7]提到的基于蠕变的一种寿命预测方法,即

$$N_{fc}=0.159\times(\Delta\gamma_c)^{-1.98} \tag{10}$$

式中:N_{fc} 为焊点蠕变失效前所经历的循环数;$\Delta\gamma_c$ 为剪切蠕变应变范围.

2 有限元模块原理

2.1 有限元参数化模型与求解

BGA 焊点的排列方式、焊点个数和间距各不相同,必须对结构进行参数化建模,以达到根据外界修改的参数,自动生成有限元模型的目的.为缩短分析时间,增加分析效率,将 BGA 模型作适当的简化处理,建立有限元模型.假设:(1)忽略印刷铜线等部件对结果的影响;(2)不考虑制造过程中所造成的残余应力及应变;(3)假设材料间为理想连接;

(4) 环境温度循环情况下，假定温度变化时，结构整体温度皆相同.

由于对角线上离结构中心最远的焊球在热循环过程中所受到的应力应变最大，同时考虑到结构具有对称性，可采用对称的简化模型进行计算. 焊点 Anand 模型是高度材料非线性的，计算过程将会经过多步迭代，时间可能长达十几个小时. 为了缩短计算时间，提高效率，构造了焊点的条形模型. 研究表明，这种条形结构模型虽然不如四分之一或八分之一模型准确，但误差在能够接受的范围内，时间却大大缩短.

加载方式分为三种：一种为环境温度循环，即只有外界环境的变化. 第二种为功率循环，即给定芯片的工作温度循环，其它部分的温度需要计算而得. 最后一种为环境温度循环与功率循环相结合，这种方式更加接近实际情况.

2.2 Darveaux 寿命预测模型

在现有的焊点疲劳寿命预测模型中，基于能量的 Darveaux 模型成为应用最为广泛的方法. 这种方法已被应用到各种类型的封装中，并与试验结果有很好的一致性[1]. Darveaux 模型中关于裂纹产生和裂纹扩展的方程式如下所示

$$N_0 = K_1 (\Delta W)^{K_2} \tag{11}$$

$$\frac{d\alpha}{dN} = K_3 (\Delta W)^{K_4} \tag{12}$$

$$N_f = N_0 + \alpha \Big/ \frac{d\alpha}{dN} \tag{13}$$

式中：N_0 为焊点起裂时经历的循环数；N_f 为焊点失效前所经历的循环数；α 为焊球直径；$d\alpha/dN$ 为裂纹扩展速率；ΔW 为两个循环的塑性功平均值之差；K_1，K_2，K_3，K_4 为材料常数，由有限元预计和试验数据综合拟合而成. 由于非弹性应变能与焊点的有限元网格密度直接相关，根据网格单元尺寸，给出了不同的参数值组合.

Darveaux 寿命模型在使用之前进行有限元应力应变场计算，然后取焊点危险部位的一个薄片状的 3D 模型，塑性功 ΔW 在焊点单元内部进行平均.

3 BGA 焊点寿命预测的综合方法

利用 VB 编程实现 BGA 焊点寿命预测的综合方法. 主程序中包括了两个模块，解析模块和有限元模块. 解析模块中，利用 Runge-kutta 法求解式(9)，得到应力应变分布，而后利用式(10)求得焊点寿命. 有限元模块部分要在主程序中调用 ANSYS，并实现主程序与 ANSYS 之间的数据传输，这些功能是利用 ANSYS 的二次开发来实现的. 图 2 所示为 VB 对 ANSYS 调用的过程.

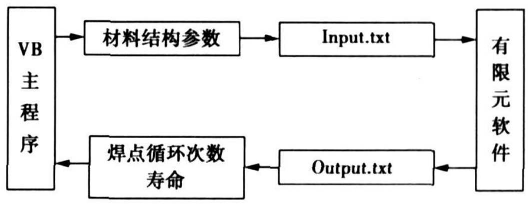

图 2　VB 主程序与 ANSYS 之间的数据传输

Fig. 2　Data communication between VB program and ANSYS

材料参数、结构参数、环境温度参数由 VB 输出到文本文件 Input 中，由 ANSYS 批处理文件将其读入并转化为其中的相应变量. 批处理文件采用 APDL(参数化设计语言)进行参数化设计，其中的变量由外部以相应的格式输入，方便用户进行优化设计. ANSYS 调用 APDL 批处理文件进行焊点应力应变场计算. 计算完毕后，进行后台后处理，即仍然通过 APDL 软件提取危险焊点的应变能密度，计算得到焊点寿命，输出到文本文件 Output 中，由主程序读取结果，显示到主程序界面. ANSYS 能自动提取用户关心的焊点部位的应力—应变、应力—时间、应变—时间、塑性能—时间数据，使得用户直观地看到在焊点整个寿命期间，其内部力学性能的演变，帮助找到整个设计中最薄弱的环节，以便进行结构改进.

用户通过焊点寿命预测综合方法的软件，在封装设计的初期，只需输入基本的结构、材料和温度循环参数，即可以通过解析法快速地实现焊点寿命的预测，计算时间约为几秒钟. 有限元模块能够实现三维建模，对材料描述也更为接近实际，特别适合在进行封装精确设计和优化的阶段使用. 由于涉及了材料的非线性和结构非线性，有限元计算的时间较长，求解速度视网格密度与计算机速度而定.

4 计算实例

实例中，芯片与基板相连后进行塑封，通过 BGA 焊点与 PCB 板相连接. 焊点材料为 63Sn37Pb，温度循环为＋125 ~－40 ℃，升降温时间为 15 min，高低温保持时间为 15 min. 初期用解析模块对焊点寿命进行快速预测时，得到寿命为 1 607 h. 精确有限元计算后，根据 Darveaux 寿命模型计算焊点寿命为 1 375 h. 两个结果相比较，用解析模块得到的结果通常会高估焊点的寿命. 同时，在改变结构或材料参数计算焊点寿命时，解析模块对参数变化更为敏感，有限元模块的结果则较平稳.

有限元计算模型如图3所示. 根据求解得到的焊点的塑性功可以确定最外侧焊点为最危险焊点. 图4为从有限元中提取数据, 在主程序中绘制的危险焊点的等效应力应变随时间变化曲线.

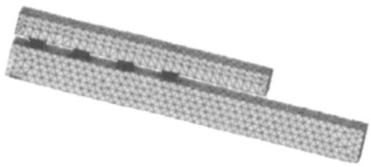

图3 实例的有限元模型

Fig 3 FEA model for example

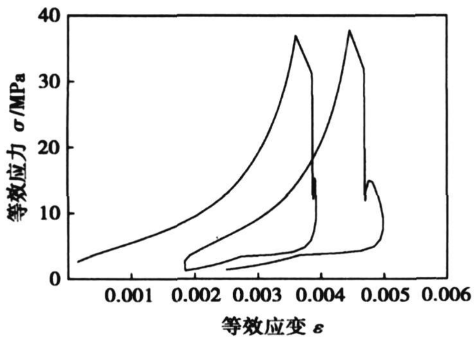

图4 危险焊点等效应力应变曲线

Fig. 4 Equivalent stress-strain curve for dangerous solder joint

5 结 论

(1) 建立了BGA封装焊点的解析模型与有限元模型, 利用ANSYS的二次开发功能将解析和有限元方法集成在BGA焊点寿命预测综合方法中, 实现了工程估算和精确预计的结合.

(2) 实例结果表明, 解析方法计算的焊点寿命要高于有限元的结果, 解析法对结构或材料参数变化更为敏感, 有限元法的结果较平稳.

参考文献:

[1] Wang G Z, Cheng Z N, Becker K, *et al*. Applying Anand model to represent the viscoplastic deformation behavior of solder alloys[J]. ASME Journal of Electronic Packaging, 2000, 123(4): 247—253.

[2] Yamada S A. Mechanism for board warpage by thermal expansion of surface mounted connector[J]. IEEE Transactions on Components, Hybrids and Manufacturing Technology, 1986, 9(4): 508—512.

[3] Yamada S A. A bonded joint analysis for surface mount components [J]. ASME Journal of Electronic Packaging, 1992, 114(1): 27—31.

[4] Heinrich S M, Shakya S, Lee P S. An analytical model developed for predicting the time-dependent shearing[J]. Journal of Electronic Packaging, 2000, 122(4): 328—334.

[5] Heinrich S M, Shakya S, Lee P S. Improved analytical estimate of global CTE mismatch displacement in areal-array solder joints[J]. ASME Journal of Electronic Packaging, 1997, 119(4): 218—227.

[6] Chen W T, Nelson C W. Thermal stresses in bonded Joints[J]. IBM Journal of Research and Development, 1979, 23(2): 178—188.

[7] Sundararajan R, McCluskey P, Azarm S. Semi analytic model for thermal fatigue failure of die attach in power electronic building blocks [C]//Fourth International High Temperature Electronics Conference, Albuquerque NM, USA, 1998: 94—102.

作者简介: 陈 颖, 女, 1977年出生, 博士, 讲师. 主要从事电子产品可靠性物理、电子产品PHM等研究. 发表论文20余篇.

Email: cheny@buaa.edu.cn

would seriously impair the properties of the soldered joint is found.

Key words: Zn based alloy; interfacial zone; reaction mechanism; microstructure

Finite element analysis on reliability of lead-free soldered joints for CSP device

YE Huan, XUE Songbai, ZHANG Liang, WANG Hui (College of Materials Science and Technology, Nanjing University of Aeronautics and Astronautics, Nanjing 210016, China). p 93—96

Abstract: Finite element method was employed to analyze the reliability of soldered joint in a CSP device. Anand model was used to establish the constitutive equation of Sn3.0Ag0.5Cu solder, the stress behavior of soldered joint was studied. The results indicate that the maximal stress is located at the upper surface of the soldered joint which is under the outermost of chip. The phenomenon of stress relaxation and accumulated enhancement could be observed obviously from the curves of stress and temperature with time cycle. Reliability of soldered joints with three usually-used heights are compared, and the results shows that the 0.35 mm × 0.18 mm one has the best reliability. Moreover, the influence of chip thickness on the reliability of soldered joints are investigated in the last part, the simulation result indicates that the influence is little.

Key words: chip scale package; lead-free soldered joint; reliability; finite element analysis

Numerical simulation on deposition process of duplex thermal barrier coating by plasma spraying

HOU Pingjun, WANG Hangong, WANG Liuying, YUAN Xiaojing (501 staff, The Second Artillery Engineering College, Xi'an 710025, China). p 97—100, 104

Abstract: Numerical simulation was performed by finite element analysis (FEA) to investigate the temperature and stress in a typical duplex thermal barrier coating. During the spraying process, the temperature of the back surface of substrate increases step by step, both the temperature and the stress of the coating fluctuate periodically within a wide range. After the deposition, the specimen was cooled to the room temperature slowly. The stresses become constant values, and the maximum radial tensile stress exists at the interface between the ceramic layer and the bonding layer, and the maximum axial and shear stresses exist at the interface, where is the concentrated stress area. The stresses of the middle interfaces are uniform. The maximum tensile stress on the ceramic layer surface is 423.7 MPa.

Key words: numerical simulation; plasma spraying; thermal barrier coatings; temperature field; residual stress

A disparity map segmentation algorithm for 3D reconstruction of weld workpiece

WANG Jun[1], LIANG Zhimin[1,2], GAO Hongming[2] (1. School of Materials Science and Technology, Hebei University of Science and Technology, Shijiazhuang 050018, China; 2. State Key Laboratory of Advanced Welding Production Technology, Harbin Institute of Technology, Harbin 150001, China). p 101—104

Abstract: In remote welding, the segmentation of disparity map is an important step to create 3D model of weld workpiece by stereo vision sensor. In this paper, the USF plane range image segmentation algorithm was introduced into disparity map segmentation, and by a region combination step, the revised algorithm can deal with the disparity map containing cylinder surface. The combination step could be divided into boundary detecting and curve region relabeling. During boundary detection, the pixel at the boundary of the segmented region after plane segmentation was recorded. In curve region relabeling, the adjacent regions in the same curved surface were assigned with the same label by comparing the boundary pixels' normal direction and distance between them. A segmented result of disparity map of saddle workpiece is shown to prove the feasibility of the algorithm.

Key words: disparity map segmentation; stereo vision; 3D reconstruction; remote welding.

Integrated life prediction method of ball grid array soldered joint

CHEN Ying, KANG Rui (Department of System Engineering of Engineering Technology, Beijing University of Aeronautics & Astronautics, Beijing 100191, China). p 105—108

Abstract: To quickly estimate the soldered joint lifetime of the ball grid array package, the analytical model of the simplified stress distribution was founded and the thermal fatigue lifetime was calculated by the creep lifetime prediction model. When the precise results were needed, the three dimensional finite element model was founded and the stress distribution in soldered joint was calculated by Anand constructive function of ANSYS, then the thermal fatigue lifetime was given by the Darveaux model. The analytical method and the finite element method were integrated to a main program by the secondary development function of ANSYS. With this program, the lifetime of the soldered joint can be quickly estimated or precisely predicted with the packaging and soldered joint dimension, material parameters and the thermal cycle parameters. Results of the case show that the analytical model results are more sensitive to the variation of the parameters, while the analyzed results by finite element model are stable.

Key words: ball grid array package; soldered joint; analytical method; FEA; lifetime predication

Effect of Ti content on microstructure toughness of deposited metal with flux cored wire

WANG Zheng[1], GUI Chibin[1], WANG Yuhua[2] (1. College of Naval Architecture & Power, Naval University of Engineering, Wuhan 430033, China; 2. Naval Military Delegation Room of 407 Factory, Luoyang 471039, Henan, China). p 109—112

Abstract: The effects of Ti content on microstructure characteristic of deposited metal with flux cored wire were investigated with different contents of Ti-Fe powder under the protection of CO_2 and Ar + 20% CO_2. The results show that the inclusions in deposited metal are mainly complex oxides composed of MnO-TiO_x-Al_2O_3-SiO_2 in Si-Mn-Ti deoxidized flux cored wire. With the increasing of Ti content in flux cored wire, the content of Ti in the inclusions increases, and most of inclusions are in the range of 0.3 ~ 2.0 μm in diameter, which promote the formation of acicular ferrite, thus those contribute to the increased toughness of deposited metal. Dissociative Ti can improve the intensity and the rigidity of microstructure, but it does worse to the microstructure toughness when too much dissociative Ti gets into the microstructure of deposited metal.

Key words: Ti; deposited metal; inclusion; toughness