Fatigue damage in solder joints

Piet J.G. Schreurs
Materials Technology Section
Eindhoven University of Technology, Postbox 513,
5600 MB Eindhoven, The Netherlands
e-mail: p.j.g.schreurs@tue.nl

Key words: fatigue, solder joints, electronic joints

Abstract
Miniaturisation of electronic products is increasingly important for reasons of functionality enhancement and freedom of design. This is realised by integration into silicon and developing miniature packages with high I/O and power dissipation density. Generally this leads to elevated temperatures, not only in the components but also in the solder joints. Elevated temperature cycling leads to fatigue damage propagation eventually resulting in failure of the joint. Numerical tools for the prediction of the fatigue lifetime of the solder joints are needed, not readily available and now under development. These tools will be integrated into simulation software which allows virtual prototyping in the design process of (consumer) electronics.

In this paper we look at the behaviour of one single solder bump, which connects an electronic chip (package) to a mother board. Temperature cycling provokes cyclic stresses due to different coefficients of thermal expansion of the combined materials. The cyclic loading results in fatigue damage initiation and propagation in the solder bump.

The fatigue damage propagation is simulated with the finite element method by using a smeared crack approach [1]. In every integration point of the mesh the consumed lifetime (cycles) of the material is calculated using the stress state and a damage evolution law. When the damage (consumed life) in a point reaches a certain limit value, a microcrack is introduced. In an iterative procedure more and more new microcracks are initiated while existing cracks are growing in length. The crack growth is associated with a reduction in certain material properties.

The simulation procedure will be outlined and effects of different parameter values will be discussed. Simulation results are shown to be mesh independent and compare qualitatively well with experimental results.
1 Introduction

In electronic devices chips are often connected to a chip package, which is connected to the Printed Circuit Board. Traditionally through-hole solder joints are used where pins of the package are soldered into holes of the board. Miniaturisation and increasing number of connections have led to the development of Ball Grid Arrays, where pads of package and board are joined with a solder bump. Nowadays chips are directly connected to the board without using a package (Chips On Board technique). Solder bumps are also used in this case, mostly in combination with an epoxy adhesive.

Component and joint temperatures are increasing due to the fact that miniaturisation leads to higher energy densities and decreased cooling rates. Repeated switching will lead to temperature cycling. Due to the mismatch of the Coefficients of Thermal Expansion (CTE) of various components, the cyclic temperatures provoke high cyclic stresses, which result in fatigue damage.

Defects in electronic devices are often caused by the failure of one single solder joint. Optimal longterm behaviour is therefore of utmost importance and can be achieved by selection of proper materials and optimization of design. Virtual prototyping, essential to shorten design cycles, is heavily based on numerical simulation and analysis of fatigue damage caused by cyclic loading is essential in this process. A numerical procedure to describe the initiation and propagation of fatigue damage is described in the following sections and applied to the analysis of the damage evolution in a solder bump, which is shown in Figure 1. The height \( H \) and the width \( W \) of the bump are (typically) 0.1 mm. Table 1 shows the CTEs of the materials involved.

![Figure 1: Solder bump.](image)

<table>
<thead>
<tr>
<th>material</th>
<th>CTE ( \times 10^{-6} ) ( ^{\circ} \text{C}^{-1} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Si</td>
<td>3 - 4</td>
</tr>
<tr>
<td>Cu</td>
<td>17</td>
</tr>
<tr>
<td>Sn63Pb37</td>
<td>25 - 27</td>
</tr>
<tr>
<td>( \text{C}_a\text{H}_b\text{O}_c )</td>
<td>16 - 26</td>
</tr>
</tbody>
</table>

2 Fatigue damage

A damage variable \( D \) is introduced to quantify the damage in the material. Its value varies between zero (0) for undamaged material and one (1) when the material is completely failed and has lost its coherence. In practice, there will be some damage in every material, due to the fabrication process, so \( D = 0 \) will probably not occur. Also, in real life, a material will be considered to show complete failure before \( D = 1 \). Complete failure during cyclic stress loading with given amplitude \( \sigma_{\text{max}} \) would occur after \( N_f \) cycles, as is indicated in Figure 2. The damage increment during one cycle is \( \Delta D = 1/N_f \). For different stress amplitudes the number of cycles to failure will be different. The value of \( N_f \) is extracted from an experimental \( (S - N) \)-curve which is shown in Figure 3 for a typical solder material. The total damage after a number of load cycles with different amplitudes can be calculated by simple summation of \( \Delta D \).
where it is assumed that Miner’s rule holds, meaning that the sequence of the loading cycles is irrelevant for the accumulated damage.

The procedure which we use to predict fatigue damage initiation and propagation is based on the smeared crack approach. We assume that damage occurs as distributed microcracks in the material. In the framework of the finite element method, these cracks are introduced in the integration points of the elements. In the next section we will describe how the microcracks are initiated. Subsequently the crack growth will be discussed.

The temperature cycling is assumed to result in a cyclic deformation of the bump, where its lower side is fixed and the upper side is displaced in horizontal direction \(0 \leq u \leq U\) while its axial displacement is zero \(v = 0\). The damage propagation is not analysed by prescribing the harmonic displacement in real time. As a matter of fact we only prescribe the maximum displacement amplitude \(U\) of the upper boundary. The elastic behaviour of the undamaged solder material is linear and isotropic with Young’s modulus \(E\) and Poisson’s ratio \(\nu\). Geometric and material parameter values are indicated and given in Figure 4 and Table 2. As we use a two-dimensional plane strain model, the bump is in fact a bar, but the inplane geometry and dimensions are realistic. Although axial stresses are present, only inplane damage evolution is studied. The bump is analysed with linear quadrilateral elements with \(2 \times 2\) Gauss integration points.

### Table 2: Parameter values for geometry and material.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>width (W)</td>
<td>0.1 [mm]</td>
</tr>
<tr>
<td>height (H)</td>
<td>0.1 [mm]</td>
</tr>
<tr>
<td>thickness (T)</td>
<td>0.1 [mm]</td>
</tr>
<tr>
<td>displacement (U)</td>
<td>0.001 [mm]</td>
</tr>
<tr>
<td>Young’s m. (E)</td>
<td>32000 [MPa]</td>
</tr>
<tr>
<td>Poisson’s r. (\nu)</td>
<td>0.37 [-]</td>
</tr>
<tr>
<td>shear m. (G)</td>
<td>16000 [MPa]</td>
</tr>
</tbody>
</table>
3 Crack initiation

The maximum displacement $U$ is prescribed and a finite element analysis is carried out where stresses are calculated in every integration point (first loop). In Figure 5 the inplane stress components are represented as stresses on a material volume (stress matrix $\sigma$). With the stresses known, it is easy to calculate the inplane principal stresses, $\sigma_1$ and $\sigma_2$, and principal stress directions, $\vec{n}_1$ and $\vec{n}_2$, in every point.

The principal stresses on a material volume in an integration point are used together with the $(S - N)$-curve to predict the fatigue life of the material. Note that this seems to be very odd, as $(S - N)$-curves describe complete failure of a tensile bar after a number of cycles at a certain stress level. Dimensions of the tensile bar are mostly normalised and are certainly not infinitely small, as is our material volume. Besides that, we consider two inplane principal stresses. However, having said this, the $(S - N)$-curve gives us the number of cycles to failure for each inplane principal direction: $N_{f1}$ and $N_{f2}$.

Because of the considered biaxial stress state in the material volume the resulting damage may be anisotropic. The incremental cyclic damage must be represented by a damage matrix $\Delta D^*$ and calculated for the principal stress directions. As long as no real crack is introduced in the material point, the material remains homogeneous and principal stress directions may vary. This means that the incremental damage matrix must be transformed to the fixed global coordinate system. This global incremental damage matrix, $\Delta D$, is saved for later use.

Initially we assume that there is no damage in the material. Before a crack is actually initiated, however, damage may be already generated. The resulting global total damage matrix, $D$, is stored for every integration point and is available.

It is now possible to calculate how many times the incremental damage $\Delta D$ must be added to the existing damage $D$ to reach total failure of the material point. To do this, equivalent values of both the existing damage and the incremental damage are calculated according to:

$$D_{eq} = \sqrt{D_1^2 + D_2^2 - D_1 D_2} \quad \Delta D_{eq} = \sqrt{\Delta D_1^2 + \Delta D_2^2 - \Delta D_1 \Delta D_2}$$

(1)

where $D_1$, $D_2$, $\Delta D_1$ and $\Delta D_2$ are the eigenvalues of $D$ and $\Delta D$ respectively. Using these values the calculation of the number of cycles to failure, $n$, for the material point at hand is straightforward:

$$D_{eq} + n \Delta D_{eq} = 1 \quad \rightarrow \quad n = \frac{1 - D_{eq}}{\Delta D_{eq}}$$

(2)

When all integration points have been visited, the most severely stressed point is selected. The number of cycles to failure of this point is $n_{min}$.

In a second loop over all integration points (see Figure 6) the global total damage matrix $D$ in each point is updated by adding to it $n_{min}$ times the global incremental damage matrix $\Delta D$, which was saved for each integration point. Subsequently its principal values, $D_1$ and $D_2$, and principal directions, $\vec{m}_1$ and $\vec{m}_2$, are calculated. Note that these directions are not the same as the principal stress directions which we have found before in such a point.

If a principal value of the total damage exceeds a given limit value $D_{min}$, a crack is introduced perpendicular to the corresponding principal direction. It may be possible that two cracks are introduced, which are perpendicular to each other. A crack in an integration point is modelled by introducing orthotropic material behaviour, where the orthotropic directions coincide with the principal damage directions. At this moment we use the linear elastic material model, the two-dimensional orthotropic version of which has four independent material parameters: two Young’s moduli ($E_1$ and $E_2$), one Poisson ratio ($\nu$) and one shear modulus ($G$). The Young’s moduli are calculated as a function of the principal damage values according to a power-like evolution law and also the shear modulus is updated.
\[
E_1 = (1 - D_1)^{\alpha_1} \cdot E \quad ; \quad E_2 = (1 - D_2)^{\alpha_2} \cdot E \quad ; \quad G = \frac{1}{2} (E_1 + E_2) \frac{1}{2(1 + \nu)} \quad (3)
\]

The reduction of the Young’s modulus can be seen as the result of the reduction of the load-carrying cross-sectional area of the material.

\[\Delta D = \frac{1}{N_f_1} \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix} \Delta D^*\]

\[D = D + n_{\text{min}} \Delta D\]

Figure 5: Crack initiation : first loop.

Figure 6: Crack initiation : second loop.

4 Crack propagation

Although damage can already have accumulated in an integration point, a crack is introduced when this damage reaches a given limit value \(D_{\text{min}}\). After a crack is introduced, we assume that its direction does not change any more. Such a crack however, can be elongated, which means that the Young’s modulus perpendicular to the crack face is reduced further.

Just like in the crack initiation phase the stress state is calculated and subsequently the normal stress perpendicular to the crack face is determined. This value is used to get the number of cycles to failure from the \((S - N)\)-curve and thus the incremental damage during one cycle. Transformation of the incremental damage matrix to the global coordinate system is not necessary anymore, because crack directions do not change. Just as during the crack initiation procedure, the number of cycles to failure is calculated for the integration point. In this case no equivalent value of the total damage is used, but both orthotropic directions are evaluated individually and stored.

After the loop over all integration points is completed, the minimum number of cycles leading to total failure of one (or more) of the integration points, \(n_{\text{min}}\), is determined. With this value all integration
points are visited again to increase the damage and reduce the stiffness – increase the crack length – according to equation (3) or, for uncracked points, to generate a crack.

When a damage variable in one of the two orthotropic directions in an integration point has reached a specified limit value $D_{\text{max}}$, total failure has occurred in that direction and the crack will not grow any further. We take $D_{\text{max}} = 0.999$ for all our calculations.

5 Results

It is known from literature that the analysis of damage propagation with the finite element method may lead to numerical problems and/or non-unique solutions because [2]

- the damage growth leads to negative stiffness: strains increase with decreasing stress;
- the deformation and damage growth is inhomogeneous.

All this leads to localisation of deformation in a zone which is completely determined by the element mesh, which is of course unacceptable.

There are a few solutions to this problem, the so-called regularisation techniques, which are mentioned here without further explanation (see e.g. [2] for more details):

- calculate stresses in an integration point but average them with stresses in the environment (non-local approach);
- use a constitutive equation where stresses are related to strains and to strain gradients (gradient models);
- use time dependent material behaviour.

To test the damage evolution procedure for mesh dependency a square plate is analysed with various meshes of linear quadrilateral elements. Geometry, dimensions and material parameters are indicated in Figure 7 and given in Table 3.

![Plate](image)

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>length</td>
<td>$L$ 1 [m]</td>
</tr>
<tr>
<td>height</td>
<td>$H$ 1 [m]</td>
</tr>
<tr>
<td>thickness</td>
<td>$T$ 0.01 [m]</td>
</tr>
<tr>
<td>displacement</td>
<td>$U$ 0.01 [m]</td>
</tr>
<tr>
<td>Young’s m.</td>
<td>$E$ $10^9$ [Pa]</td>
</tr>
<tr>
<td>Poisson’s r.</td>
<td>$\nu$ 0.25 [-]</td>
</tr>
</tbody>
</table>

Figure 7: Plate. Table 3: Parameter values.

Figure 8 shows the cracks in the plate at the same stage in the damage propagation process for three different meshes ($10 \times 10$, $15 \times 15$, $20 \times 20$ elements). The total damage as a function of the number of load cycles is also plotted. It can be concluded that almost no mesh dependency is apparent. The reason for this may be that the cyclic character of the analysis introduces a time dependent behaviour, which is,
as indicated above, one of the regularisation techniques to prevent problems in damage analysis.

Figure 8: Damage distribution and evolution in a plate, calculated with regular meshes of 10 x 10, 15 x 15 and 20 x 20 quadrilateral elements.

The damage evolution in a bump of solder material is simulated where a subdivision of 10 x 10 linear quadrilateral elements is used, which leads to results with sufficient accuracy and resolution for our purpose. Parameters in the damage evolution relation are varied. In all simulations only one crack is allowed in an integration point ($D_{2,\text{min}} = 1$), which renders $D_{2,\text{max}}$ and $\alpha_2$ irrelevant.

Figure 9 shows the total damage in the solder bump as a function of the total number of cycles. The legend in the figure indicates the values ($D_{1,\text{min}}, \alpha_1$) which are used in the four simulations. The exponential parameter $\alpha_1$ influences the rate of the damage accumulation as is an obvious result of the used evolution law for the Young’s modulus.
Figure 9: Total damage in the solder bump versus number of cycles for different values of parameters $(D_{1,\text{min}}, \alpha_1)$ indicated in the legend and for $U = 0.001$ [mm].

Figure 10 shows damage in a bump which has been experimentally loaded to failure under shear. The damage distribution compares well with the numerical simulation results. The color plot shows the value of $D = D_1$, increasing from blue ($D = 0$) towards red ($D \rightarrow 1$).

Figure 10: Simulated and experimental observed damage in a solder bump.

The influence of $D_{1,\text{min}}$, being the minimum damage value which has to be reached before a crack is generated, is clearly seen in the development of damage in the bump, which is shown in Figure 11 as integration point cracks. For the four simulations mentioned above, three crack distributions are shown, where the total damage $D$ is approximately the same. The number of cycles $N$ is also indicated in the figure.
Figure 11: Damage distribution (microcracks) in a solder bump for different values of parameters $(D_{1,min}, \alpha_1)$ according to the legend in Figure 9.

For higher $D_{1,min}$ the material seems to react in a more brittle way and the crack follows the boundary longer before interior damage appears.

6 Conclusions and future work

The smeared crack procedure to model fatigue damage leads to results which compare qualitatively well with experimental observations. The numerical results are independent of the element mesh used for the spatial discretisation.

In the absence of damage the material behaviour is assumed to be linear elastic. In the future this has to be replaced with a viscoplastic model, suitable to describe creep, because solder material in electronic joints is above its homologous temperature. Implementation of such a model is not a major problem. However, the use of the number of cycles as a time parameter has to be reconsidered, because the viscoplastic behaviour depends on the real time. A multi-level procedure in the time domain is under development at the moment, where the material behaviour during one cycle is averaged.

The solder material is considered to be a continuum. In real solder materials however, the microstructural entities – phases, crystals – are large compared to the dimensions of the joint (= bump). It is also known that the microstructure changes considerably under static and cyclic thermal loading. This change in microstructure can be simulated by solving a diffusion equation and these results will be combined with the damage evolution analysis presented here. Because it is observed that cracks develop at and
grow along phase boundaries, the damage initiation and evolution procedure will need enhancement. Because local stress concentrations can be expected then, a regularisation method will be needed, e.g. calculation of a non-local stress to control the damage evolution. The use of an $(S - N)$-curve to determine the number of cycles to failure in a material point has to be replaced by a local crack initiation criterion and crack growth law. Although Paris-like laws may be used at a local level, we can also use a multi-level procedure in the spatial domain to analyse damage growth (see for such procedures [5]).

Acknowledgements

The research presented here is done in cooperation with and supported by Philips N.V.

References


