DIPLOMARBEIT

Titel der Diplomarbeit

Accelerated Lifetime Estimation of Copper Ball Bonds Using Experimental Methods and Finite Element Analysis

angestrebter akademischer Grad

Magister der Naturwissenschaften (Mag. rer.nat.)

Verfasserin / Verfasser: Wolfgang Karl Trasischker
Matrikel-Nummer: 0503357
Studienrichtung (lt. Studienblatt): A411
Betreuerin / Betreuer: Dr. Golta Khatibi

Wien, am 11.04.2011
First of all I want to thank Prof. Brigitte Weiss and Prof. Viktor Gröger for giving me the opportunity to do my diploma thesis in the research group for Micromaterials at the University of Vienna. I also want to express my utmost gratitude to Infineon Technologies Austria and FFG for the cooperation and the funding of the Comet-K project.

I appreciated the good teamwork and the motivating spirit within the colleagues and I am especially grateful for the constructive support of scientific head of the Comet-K project Micromat Dr. Golta Khatibi and her ambitious guiding. Furthermore I want to thank Dr. Martin Lederer for his patience and input in numerous discussions. Thanks to Mag. a Alice Lassnig for her contributions to this work.

For the possibility of getting insight into the professional working of Infineon Technologies Austria I am glad to thank Dr. Michael Nelhiebel and DI Rainer Pelzer. Not only the samples for the experiments were provided by them but I also profited by their knowledge, active support and confidence in my competences.

Special thanks to KAI GmbH where Dr. Bala Karunamurty greatly contributed to my intentions and ideas in a very competent and friendly way.

Thanks to Dr. Ingrid Taferner and the Erich Lackner foundation for the financial support and the motivating requirements during my academic studies!

Overall the biggest thanks go to my parents Helga Seeberg and Karl Trasischker and my sister Christiane Tschernitz. I am thankful for the possibilities they provided me and the fact that they kept on supporting me and my ambitions.

Finally I want to thank Freya Klepsch for being that patient and understanding during the work on this thesis.
# Contents

## 1 Introduction and Motivation

## 2 Basic Considerations

<table>
<thead>
<tr>
<th>2.1</th>
<th>Strength and Deformation of Materials</th>
<th>13</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1.1</td>
<td>Tensile Test</td>
<td>14</td>
</tr>
<tr>
<td>2.1.2</td>
<td>Mechanical Properties of Materials</td>
<td>15</td>
</tr>
<tr>
<td>2.1.3</td>
<td>Stress States - Yielding and Fracture Under Combined Stresses</td>
<td>16</td>
</tr>
<tr>
<td>2.1.4</td>
<td>Behaviour of Materials and Material Models</td>
<td>18</td>
</tr>
</tbody>
</table>

| 2.2  | Thermal Stress and Strain             | 19 |

| 2.3  | Fatigue Models                        | 20 |

| 2.3.1| Stress Based Approach to Fatigue       | 20 |
| 2.3.2| Strain Based Approach to Fatigue       | 21 |

| 2.4  | Finite Elements                       | 23 |

| 2.4.1| Basic Ideas of FEM                    | 23 |
| 2.4.2| Linear Analysis                        | 26 |
| 2.4.3| Nonlinear Analysis                     | 26 |
| 2.4.4| FEM Tools                              | 27 |

| 2.5  | Wire Bonding                           | 28 |

| 2.5.1| Power Semiconductor Architecture      | 28 |
| 2.5.2| Welding in Bonding Processes           | 29 |
| 2.5.3| Materials in Wire Bonding              | 32 |
| 2.5.4| Testing of Wire Bonds                  | 34 |
| 2.5.5| Wire Bonding Related Failures in Power Semiconductors | 35 |
| 2.5.6| Advanced Testing and Desired Developments | 35 |

## 3 Experimental

| 3.1  | Ultrasonic Fatigue Testing            | 37 |

| 3.1.1| Frequency and Time Compression         | 38 |
| 3.1.2| Testing Principles                     | 38 |
| 3.1.3| Testing Equipment and Setup            | 39 |
3.2 Accelerated Testing of Micro-Joints .......................... 41
  3.2.1 Experimental Procedure .............................. 41
  3.2.2 Determination of Acceleration ......................... 42
3.3 Finite Element Modelling for Accelerated Testing of Micro-Joints 43
  3.3.1 Material Models .................................. 44
  3.3.2 Geometry ........................................ 45
  3.3.3 Mesh and Model ................................ 47
  3.3.4 Loads ............................................ 48
  3.3.5 Geometry Check ................................ 48
  3.3.6 Stress Determination .............................. 49
3.4 Lifetime Modelling ......................................... 49
  3.4.1 Life Time Modelling → Estimated Stresses ............ 50
  3.4.2 Life Time Modelling → FEM calculated Stresses .... 50
3.5 Experimental - Tech A ..................................... 50
  3.5.1 Sample and Sample Preparation → Tech A .......... 51
  3.5.2 Experimental Procedure → Tech A ................... 51
  3.5.3 Experimental Results → Tech A ..................... 52
  3.5.4 Simulation → Tech A .............................. 54
  3.5.5 Results of the Simulation → Tech A ................. 62
  3.5.6 Lifetime Modelling → Tech A ....................... 63
3.6 Experimental - Tech B ..................................... 67
  3.6.1 Sample and Sample Preparation → Tech B .......... 67
  3.6.2 Experimental Procedure → Tech B ................... 70
  3.6.3 Experimental Results → Tech B ..................... 73
  3.6.4 Simulation → Tech B .............................. 83
  3.6.5 Results of the Simulation → Tech B ................. 92
  3.6.6 Lifetime Modelling → Tech B ....................... 93
3.7 Comparison of Tech A and Tech B Samples ................. 96
  3.7.1 Shear Test ...................................... 97
  3.7.2 Estimated Stresses ............................... 97
  3.7.3 FEM Calculated Stresses .......................... 98

Summary ................................................. 100

Zusammenfassung ......................................... 105
Chapter 1

Introduction and Motivation

In global competition, manufacturers of microelectronic components are urged to produce innovative products with better performance and higher reliability at reduced costs and shorter time to market periods. Electronic industry acts in one of the most dynamic and challenging industry sectors and covers a wide variety of products ranging from telecommunication to automotive, aviation and traction applications.

High reliability is one of the main issues for the electronic industry. Market and the consumer often require zero-failure products, which becomes a very challenging task due to increasing complexity, higher packing density and continuous down scaling of the electronic devices.

A wide spectrum of materials and material combinations (metals, semiconductors, dielectrics) with dissimilar physical and electrical properties are used in microelectronic components. Interaction of the inherent properties of the different parts and materials often affects the performance and requires a fundamental understanding of the material scientific aspects. Material interconnects are used for electrical contact, mechanical support or heat dissipation and play a major role for the reliability of electronic components. Used interconnections show a wide range of size (from mm to nm), material combination and production method what makes them a complex subject and electronic devices susceptible to fatigue and failure during their service life.

Wire bonding is mainly used for providing electrical connection between the silicon chip and the external leads of a semiconductor device. Several different techniques like ball, wedge, stitch or ribbon bonding are used and deal with wire diameters between 20 µm and 700 µm. At present wire
bonding covers most of the interconnections in semiconductor chips because of cost effectiveness and flexibility of this bonding technology. Although being well established no physical model is available for this process, leaving it for empiric descriptions.

In figure 1.1 an example for a fine wire ball bond array, as used in microelectronics, can be seen.

![Bond array with a ball bond diameter of 40 µm.](image)

Figure 1.1: Bond array with a ball bond diameter of 40 µm. [13].

Numerous temperature cycles during on/off periods of the devices induce repeated loads and stresses into the interface layers of the respective components due to the mismatch of thermal expansion. The accumulation of these stresses leads to plastic deformation, crack initiation and propagation, followed by thermo-mechanically induced fatigue, failure of the interconnect and finally the device.

Failure driven by thermal mismatch of adjacent materials has been recognised to be one of the main reliability concerns of electronic devices and especially affects wire bond connections. Losing electrical connection between contacts due to a wire lift off is an important failure mode of microelectronic components. See a picture of a wire lift off for ball and wedge bond in figure 1.2.
In order to test the lifetime of wire bonded interconnects often extremely time intensive procedures, like thermal cycling are applied. Pushed to new developments under short time-to-market periods and reduced costs there is need for an alternative solution in order to replace even highly accelerated thermal cycling.

Using a different approach for rapid qualification of interconnects the Micromaterials Group at the University of Vienna has developed a mechanical setup to induce cyclic stresses in various types of metallic interconnects. The method works at 20 kHz and enables end of life tests and the examination of lifetime curves in extremely reduced testing time. Due to the fact that wire bonding is nowadays one of the most important methods to provide electrical connection between contacts and the promising developments by the research group for Micromaterials the aim of this diploma thesis was to apply the proposed method to thin wire ball bond interconnections as used in microelectronic devices. Because of the importance of thermosonic wire ball bonding of copper wire to aluminium and the lack of fatigue testing procedures and data this interconnection type was chosen as investigative object of this work.

Primary objectives of this thesis were to develop a experimental method to determine the reliability and fatigue lifetime of single copper ball bonds on aluminium and to propose according lifetime models for single and multiple ball bonded microelectronic devices. Commonly used to test the reliability of ball bonds are destructive methods like standard pull and shear test. These procedures evaluate the adhesion
of the wire bond to the pad without including any cyclic loading or response. The test results are also dependent on factors like skills of the operator or failure mode what may lead to faulty output. Thus new and fast experimental methods for reliability assessment of miniaturized interconnects are strongly required.

In this work investigated samples were two different commercial power semiconductors as used in automotive applications. Wire ball bonding was performed with a 50 µm thin copper wire what is comparable with the diameter of a human hair. Due to the sample geometry and the extreme small size of processed components a special sample preparation was required in order to apply the accelerated testing method. Microscope supported work, tailored templates and the use of an especially developed soldering technique allowed a suitable and reproducible sample preparation. In order not to modify the sample properties all applied methods had to support the mechanical and thermal restrictions of the microcomponent. Severe exposure to heat may degrade metallic interfaces and excessive mechanical manipulation is likely to cause damage of the sensitive components before the actual test.

Neither comparable sample preparation nor testing has yet been found in literature for this interconnection type.

Experimental determination of stress-strain behaviour of interconnects during an accelerated fatigue loading is extremely difficult due to their small size and complex geometry.

Thus in this work in addition to an estimated average shear stress approach the Finite Elemente Method (FEM) was used as a numerical tool to detect resonance effects and calculate equivalent von Mises stresses.

Testing for both samples included different sample preparations and geometries leading to incomparable experimental results based on the estimating average shear stress. FEM simulation was performed, using 3 dimensional geometrical models, in order to determine influences of the geometry on the experimental output and to enable a comparison of measured data for the different samples.

Based on the Basquin relation experimentally obtained data was to be further processed in order to propose lifetime predicting models in terms of estimated average shear stresses and equivalent stresses computed by means of FEM. Consideration and compensation of geometry effects by FEM has become especially valuable for the comparability of lifetime predicting models obtained by different experimental procedures.
This work is organized in three chapters.  
A short introduction and motivation for this work is given in the first chapter **1 Introduction and Motivation**.  
From a basic approach it was tried to provide all information required to understand the applied methods and interpretations in chapter **2 Basic Considerations**. Within this chapter brief overviews are given regarding topics used for the FEM simulation and the Lifetime Modelling. Topics related to wire bonding that are useful for the understanding of this work are briefly discussed.  
The used technologies and sample preparations are presented in chapter **3 Experimental**. The basic experimental setup, sample preparation, FEM simulation and lifetime modelling are discussed for both tested samples. Obtained results were compared with each other in the end of this chapter.
Chapter 2

Basic Considerations

2.1 Strength and Deformation of Materials

The following section provides a brief overview about relevant topics regarding the strength and deformation of materials. For detailed information see [10] and [11].

Mainly three types of deformation can be distinguished: elastic, plastic and creep deformation. In contrary to elastic and plastic deformation, creep shows time dependency.

For the elastic region one can state that atomic bonds are stretched but not broken, yielding a return to the initial state after removing the load.

Plastic deformation can be explained with the movement of dislocations resulting in a lasting deformation.

Mainly used in techniques and industry are polycrystalline materials. From a macroscopic point of view and neglecting time dependent effects one can separate deformation of polycrystals into two regions:

1. Elastic region

A polycrystalline body is composed of single crystals, called grains, with different orientations. If an external load acts on the body, those grains oriented in a way that the Schmid-factor, \( m = \cos(\kappa) \cdot \cos(\lambda) \), (according to Schmids law for the critical resolved shear stress \( \tau = \sigma \cdot \cos(\kappa) \cdot \cos(\lambda) \) with \( \sigma \) being the applied tensile stress, \( \kappa \) is the angle between the normal of the slip plane and the direction of the applied force and \( \lambda \) is the angle between the slip direction and the direction of the applied force) is high, will start deforming. Due to continuity reasons, neighbours have to deform as well but in general a neighbouring grain will not have reached the critical shear stress. Thus elastic deformation occurs and leads to higher internal stresses. With this
increase of internal stress more and more grains will reach the critical shear stress and the body can deform plastically. Considering dislocations, the grain boundaries act as obstacles for the moving dislocations and they will form a pile up in front of the next grain. The accumulation of dislocations will increase the internal stress of the grains and will so force further grains to reach the critical shear stress.

2. Plastic region
In polycrystalline materials different resolved shear stresses occur because of the differently oriented grains. Due to continuity reasons a single grain can only deform if five slip systems are activated or neighbouring grains deform as well. With respect to dislocations this may be explained by a pile up of dislocations in front of insuperable grain boundaries and a transfer of stress to the neighbouring grain. Macroscopic plasticity of a polycrystal starts when grains can deform plastically without violation of the assembly.

2.1.1 Tensile Test
A tensile test measures the reaction of a material due to an applied tensile load. Typically engineering stress versus strain curves are recorded, like depicted in figure 2.1. A macroscopic description is given beneath.

![Figure 2.1: Schematic stress strain curve of a tensile test.](image)

Loading starts in the elastic region. A linear region can be recognized and yields the simplified Hook's law (see equation 2.1) that is valid for small deformations.

\[ \sigma = E \epsilon \]  

(2.1)
The proportionality factor or the slope, \( E \), of the curve is called Young’s modulus. The end of the linear region displays the beginning of the plastic region with a continuous transition. In order to distinguish between elastic and plastic region, a specific plastic displacement value was determined separating the two regions. Typically the yield stress or proof stress, \( R_{p\,0.2} \), is determined at a plastic deformation of 0.2%.

After this criterion the plastic deformation starts and the curve deviates from the initial line. The stress continues to increase. With decreasing \( d\sigma/d\epsilon \) the curve reaches its point of maximum stress, called the ultimate tensile strength, \( R_m \) at \( A_g \), before it decays and finally ends in the fracture of the specimen. The point of the highest stress is characteristic because of a local change of the cross section, called necking.

According to a more scientific approach, \( R_{p\,0.2} \) is called yield strength, \( \sigma_{\text{yield}} \), \( R_m \) is called ultimate tensile strength, \( \sigma_{\text{UTS}} \) and \( A_g \) is called uniform strain, \( \epsilon_{\text{uniform}} \) in the upcoming sections.

### 2.1.2 Mechanical Properties of Materials

Loaded solid bodies try to resist deformation by internal forces and stresses. For small deflections the stress is proportional to the strain (see equation 2.2 and 2.3).

\[
\text{tensile deformation: } \frac{F_{\text{normal}}}{q} = E \frac{\delta l_1}{l_0} \rightarrow \sigma = E \epsilon \quad (2.2)
\]

\[
\text{shear deformation: } \frac{F_{\text{parallel}}}{q} = G \frac{\delta l_2}{l_0} \rightarrow \tau = G \gamma \quad (2.3)
\]

Here \( F_{\text{normal}} \) and \( F_{\text{parallel}} \) are the forces normal and parallel to the respective surface, \( q \) is the cross section, and \( \delta l_1, \delta l_2, l_0 \) and \( \epsilon \) are the deformations and initial dimensions.

Going into more detail equations (2.2) and (2.3) change from a one dimensional equation to a tensor equation (2.4).

\[
\sigma_{ij} = \sum_{k,l=1}^{3} C_{ijkl} \epsilon_{kl} \quad (2.4)
\]

The three dimensional stress tensor, \( \sigma_{ij} \), displays the general stress state of a cubic volume. It includes nine components which reduce to 6 independent components due to symmetry reasons. Along the diagonal one can find the normal stresses \( \sigma_{xx}, \sigma_{yy}, \sigma_{zz} \), whereas at the other positions one can find the shear stresses \( \sigma_{xy}, \sigma_{xz}, \sigma_{yz} \). The elements of the stress tensor depend on the
orientation of the coordinate system. 

$C_{ijkl}$ is often called the **stiffness matrix** although it is a tensor of rank 4 holding 81 elements in general. Again due to symmetry reasons the number of independent elastic constants can at least be reduced to 21 making it possible to rewrite it to a 6x6 matrix $C_{ij}$.

For the general strain state one can form the three dimensional **strain tensor**, $\epsilon_{kl}$.

The isotropic elastic constants are further reduced to the Young’s modulus, $E$, and the Shear modulus, $G$. These two constants can be linked to each other by the Poissons’s ratio, $\nu$.

$$-\nu \epsilon_{yy} = \epsilon_{zz} = -\nu \epsilon_{xx}$$ \hspace{1cm} (2.5)

In equation 2.5 $\nu$ links an elongation in $z$, that is parallel to the tensile stress, to a transverse deformation in $x$ and $y$, meaning that elongated bodies change their cross sections.

$$G = \frac{E}{2(1 + \nu)}$$ \hspace{1cm} (2.6)

Equation 2.6 shows the dependency of the shear modulus $G$ on the independent elastic constants $E$ and $\nu$.

### 2.1.3 Stress States - Yielding and Fracture Under Combined Stresses

In general materials can undergo complex stress states as a reaction to their loading. To describe the respective loading, different methods were developed. A brief overview based on [10] is given beneath.

**Maximum Normal Stress**

The largest normal stress is utilised to represent the stress state (see equation 2.7). Since normal stresses depend on the coordinate system, it is reasonable to perform a coordinate transformation to the principal system before computing the stresses, then called principal normal stresses. A three dimensional graphic interpretation of the according failure criteria yields a cube aligned to the principal axes. The safe region lies inside this surface.

$$\sigma_{normal} = \text{MAX}(|\sigma_1|,|\sigma_2|,|\sigma_3|)$$ \hspace{1cm} (2.7)
Maximum Shear Stress Criterion

Maximum shear stresses can be calculated on basis of the principal system. The obtained stresses are 45° inclined to the principal axes (see equation \(2.8\)).

\[
\tau_1 = \frac{|\sigma_2 - \sigma_3|}{2}, \quad \tau_2 = \frac{|\sigma_1 - \sigma_3|}{2}, \quad \tau_3 = \frac{|\sigma_1 - \sigma_2|}{2},
\]

(2.8)

The highest value is used for according failure criteria, see equation \(2.9\).

\[
\tau_{\text{shear}} = \text{MAX}(\tau_1, \tau_2, \tau_3)
\]

(2.9)

Octahedral Shear Stress Criterion

Assuming a coordinate system with x-, y-, z-axes coincident with the principal axes and a plane intersecting the axes at equal values, one obtains an octahedral plane. The according normal and shear stresses on this plane are (see equations \(2.10\) and \(2.11\)).

\[
\sigma_{\text{oct}} = \frac{\sigma_1 + \sigma_2 + \sigma_3}{3}
\]

(2.10)

\[
\tau_{\text{oct}} = \frac{1}{3} \sqrt{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}
\]

(2.11)

With some manipulation one gets expressions including general stress states (see equations \(2.12\) and \(2.13\)).

\[
\sigma_{\text{oct}} = \frac{\sigma_x + \sigma_y + \sigma_z}{3}
\]

(2.12)

\[
\tau_{\text{oct}} = \frac{1}{3} \sqrt{(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2)}
\]

(2.13)

In a failure criteria one assumes that failure occurs, if the shear stress on a octahedral plane reaches the critical value, see equation \(2.14\).

\[
\tau_{\text{oct-crit}} = \tau_{\text{oct}}
\]

(2.14)

With \(\tau_{\text{oct}}\) being the octahedral shear stress and \(\tau_{\text{oct-crit}}\) being the stress necessary to cause failure.

For convenience it is useful to write the failure criteria in terms of uniaxial tensile stresses, yielding equation \(2.15\).

\[
\tau_{\text{oct-crit}} = \frac{\sqrt{2}}{3} \sigma_0
\]

(2.15)
Where $\sigma_0$ is the critical stress for a uniaxial tensile test. Combination of equation (2.13) and (2.15) yields a useful and valid failure criteria. A complex stress state can be evaluated and compared with uniaxial tensile data; a property that is known for most materials (see equation (2.16)).

$$
\sigma_v = \frac{1}{\sqrt{2}} \sqrt{(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2)} = \sigma_0
$$

(2.16)

The explained criterion is widely used and also known as von Mises criterion, with $\sigma_v$ being the von Mises stress. Maximum shear stress criterion and octahedral shear stress criterion show some similarities and in fact they cover almost the same security domain. A further mutual attribute is the fact that the hydrostatical stress vanishes.

2.1.4 Behaviour of Materials and Material Models

Based on [10] and [11] a short outline on the behaviour of materials and material models is given.

Considering monotonic loading, the stress strain curve will follow Hook’s law till the yield strength and will progress in the plastic region. If the direction of loading is then reversed, a hysteresis of the stress-strain curve is observed. This is called Bauschinger effect and includes a strain hardening effect of the material. Proceeding with the described reversing of the strain direction yields cyclic loading, playing an important role in fatigue analysis.

Most common are two hardening rules, called isotropic and kinematic hardening.

Isotropic Hardening

After loading a material to a certain point $(\sigma_1, \epsilon_1)$ beyond the yield stress, the strain direction is reversed. The point of yielding according to the reversed direction will appear at $(-\sigma_1, \epsilon'_1)$. So a range of two times the highest load is necessary, $\Delta \sigma = 2\sigma_1$, to cause yielding for the reversed strain direction, like depicted in figure 2.2 (a).

Kinematic Hardening

Again the material is loaded till $(\sigma_1, \epsilon_1)$, passing the yield strength at $(\sigma_{yield}, \epsilon_{yield})$. Reversed loading happens along a line parallel to the Hook’s law till the total amount of reversed load reaches two times the initial yield stress, $\Delta \sigma = 2\sigma_{yield}$. After this value is reached further deformation is plastic. See a sketch of the described material behaviour in figure 2.2 (b).


2.2 Thermal Stress and Strain

For many materials an increasing temperature is accompanied by an expansion of the body, and the contrary holds for decreasing temperatures. Only a simplified approach is given beneath, for detailed information see [10] and [21].

Materials are in general not isotropic. Hence thermal expansion may not be isotropic and can cause internal stresses and strains.

Assuming isotropy the thermal strain, $\varepsilon_{th}$, is linear proportional to the temperature change, $\Delta T$ (see equation 2.17).

$$\varepsilon_{th} = \alpha \Delta T \quad \text{with} \quad \alpha = \frac{1}{L} \frac{dL}{dT}$$  \hspace{1cm} (2.17)

With $\alpha$ being the coefficient of thermal expansion, L is the initial length of the specimen and T the absolute temperature. This effect plays an important role for composite bodies with a mismatch of the thermal coefficients of expansion. According to equation 2.17 the strain occurring between two bodies with different expansion coefficients is (see equation 2.18),

$$\varepsilon_{th} = (\alpha_1 - \alpha_2) \Delta T$$  \hspace{1cm} (2.18)

with $\alpha_1$ and $\alpha_2$ being the two respective thermal expansion coefficients.
2.3 Fatigue Models

Fatigue is the failure of materials due to repeated loads. A brief outline including basic fatigue models is given according to [10], [28] and [31]. Polycrystalline bodies deform as discussed in section 2.1. In a loaded solid planes with a high Schmid factor, m, will start deforming and develop slip bands. As more and more loading cycles are applied the number of slip bands increases. Certain bands will develop into cracks and proceeding within the grains forms microstructural damage.

2.3.1 Stress Based Approach to Fatigue

In cyclic loading the applied load alternates between a maximum and a minimum value. When these load levels are constant, constant amplitude stressing is defined (see figure 2.3). Important relationships appearing in this type of analysis are the stress range, $\Delta \sigma$, the mean stress, $\sigma_m$ and the stress amplitude, $\sigma_a$, as listed in equation 2.19.

$$\Delta \sigma = \sigma_{\text{max}} - \sigma_{\text{min}}, \quad \sigma_m = \frac{\sigma_{\text{max}} + \sigma_{\text{min}}}{2}, \quad \sigma_a = \frac{\Delta \sigma}{2} \quad (2.19)$$

![Figure 2.3: Cyclic stresses.](image)

The average stress, $S$, is formed by the fraction of the according load and area. Special stress distributions due to geometry or stress raisers like notches are not included.

More sophisticated is the calculation of point stresses, $\sigma$, that are in general not equal to the average stress. Due to its complexity it is not easy to obtain values for the different point stresses and frequently one has to rely on numerical methods like Finite Elements to compute satisfying results.

In order to display and analyse fatigue data Wöhler curves are used. These graphs display the dependence of the applied stress on the ordinate from...
the cycles to failure, $N_f$, on the abscissa. Each point in a Wöhler curve corresponds to the failure of the specimen at a certain number of cycles due to the applied load. A schematic sketch of a Wöhler curve can be seen in figure 2.4.

![Figure 2.4: Systematic sketch of a Wöhler curve.](image)

For many materials Wöhler curves converge to a certain level of stress under which fatigue failure does not occur, called **fatigue limit**. There are two regions of fatigue, called **low-cycle** and **high-cycle fatigue**. The latter describes fatigue due to a high number of cycles, where the applied stresses are sufficient low, so that plasticity effects play a minor role. In this case a stress based approach is reasonable and fatigue data can be described with a power law called **Basquin relation**, like shown in equation (2.20).

$$\sigma_a = C \cdot (N_f)^b$$  \hspace{1cm} (2.20)

With $C$ and $b$ being fitting constants on the one hand and material properties on the other hand.

Low-cycle fatigue is defined up to $10^4$ cycles.

### 2.3.2 Strain Based Approach to Fatigue

In contrast to section 2.3.1 fatigue will be investigated with respect to strain. Recorded data are therefore displayed in a strain versus cycles to failure plot. In general strain is composed of an elastic and a plastic part (see equation (2.21)).

$$\epsilon_{\text{total\,-a}} = \epsilon_{\text{elastic\,-a}} + \epsilon_{\text{plastic\,-a}}$$  \hspace{1cm} (2.21)

The elastic part of equation (2.21) is according to Hook’s law: $\epsilon_{\text{elastic\,-a}} = \sigma_a / E$. Values for the plastic strain can be obtained by measuring the half maximum width of the stress-strain hysteresis.

If the elastic and plastic strain are plotted separately versus cycles to failure,
individual relationships can be derived. For the elastic region, the relationship \( \epsilon_{\text{elastic-a}} = \sigma_a / E \) in combination with equation 2.20 yields equation 2.22.

\[
\epsilon_{\text{elastic-a}} = \frac{\sigma'_f}{E} (N_f)^b
\]  

(2.22)

A similar relationship for the elastic region in equation 2.21 was discovered by Coffin and Manson in 1954. It describes the dependency of plastic deformation, \( \epsilon_{\text{plastic-a}} \), from cycles to failure, \( N_f \), for fatigue data, see equation 2.23.

\[
\epsilon_{\text{plastic-a}} = \epsilon'_f (N_f)^c
\]  

(2.23)

With \( \sigma'_f, \epsilon'_f, c \) and \( b \) being constants. Combination of equations 2.22 and 2.23 yields a Coffin-Manson like relationship (see equation 2.24).

\[
\epsilon_{\text{total-a}} = \frac{\sigma'_f}{E} (2N_f)^b + \epsilon'_f (2N_f)^c
\]  

(2.24)

When total strain is plotted against cycles to failure on a logarithmic scaled diagram a linear relationship for low- and high-cycle fatigue can be observed like depicted in figure 2.5.

![Figure 2.5: Total strain (solid bold) as the sum of elastic (solid) and plastic strain (dashed).](image)

For low-cycle fatigue effects due to plastic deformation and ductility dominate the fatigue performance of the sample.

In a high-cycle fatigue region elastic strain and the yield strength of the probe play the important role and equation 2.24 can be reduced to its first term, leading to the Basquin relation.
2.4 Finite Elements

Being a method to numerically solve mathematical problems, the theory of finite elements represents a powerful method to solve various problems. In the past years the Finite Element Method (FEM) has experienced vast progress and rapid growth, making it to an important tool in engineering. In this section only the basic principles will be mentioned according to references [22] and [25]. It is tried to give a not FEM experienced reader an impression of what are the fundamental ideas.

2.4.1 Basic Ideas of FEM

With the aim of determining the stress and strain distribution over a loaded body in a structural analysis, the body needs to be described mathematically. This can be done by discretization, what means a subdivision of the body into well defined elements. Kirsch in 1868 [20] followed exactly this idea, divided a volume into cubes and replaced them by a framework. In FEM this process is called meshing and the obtained discretization is called mesh.

By adopting the framework according to the properties of the respective cube it is possible to copy the real body by the assembly of the framework. The mathematical description yields a system of equations familiar to those of usually developed frameworks. Due to the lack of computational power this ansatz was neglected until it was revitalized around 1959.

The basic ideas of FEM can be explained by a two dimensional framework, see figure 2.6 (a). One element is defined by a truss, (1), (2), (3), and joints, called nodes, (a), (b), (c). Forces and Displacements can be defined in a local

![Diagram](image_url)
coordinate system with forces and displacements parallel to the respective truss. A local relation between forces, \( f_i \), and displacements, \( u_j \), also called degrees of freedom, is searched. In general degrees of freedom can include displacements as well as Rotations and can be related to the local or global coordinate system. For bar (1), a local relationship between force, \( f \), and displacement, \( u \), can be compiled, like shown in equation 2.25.

\[
\begin{pmatrix}
  f_1 \\
  f_2
\end{pmatrix} =
\begin{pmatrix}
  k_{11} & k_{21} \\
  k_{21} & k_{22}
\end{pmatrix}
\begin{pmatrix}
  u_1 \\
  u_2
\end{pmatrix} = f = k u
\]

(2.25)

The matrix \( k \) is called element stiffness matrix in the local coordinate system. Deformations and forces can be transformed to the global coordinate system, see equation 2.26.

\[
u = T U \quad \& \quad F = T^T f
\]

(2.26)

With \( U \) being the global displacements, and \( T \) the transformation matrix. Inserting equation 2.26 into 2.25 yields a global formulation, see equation 2.27.

\[
F = T^T k u \rightarrow F(e) = K(e) U(e)
\]

(2.27)

Where subscript \( (e) \) indicates element matrices in the global coordinate system. A Formulation for the complete framework can now be assembled, see equation 2.28.

\[
F_k = \sum_l K_{kl} U_l \rightarrow F = K U
\]

(2.28)

With \( k, l=1, 2, \ldots, N \), \( N \) being the number of global degrees of freedom and \( K \) being the total stiffness matrix. Assembling the total stiffness matrix is a complex process and is not further explained in here. Including boundary conditions a system of known and unknown forces, \( F_a, F_b \), and unknown and known displacements, \( U_a, U_b \), is obtained. Finally a system of linear equations for the unknown displacements, \( U_a \) and forces, \( F_b \) can be solved. For a structure including many elements thousands of unknowns can arise what makes it reasonable to use computational power for the solution.

Variational principles like the Virtual Displacement search for the equilibrium state where the variation of the total energy, \( \delta A \), vanishes. This leads to equation 2.29 with \( \sigma \) being inner stresses, \( \delta \epsilon \) being the virtual strain, \( F \) being the external Forces and \( \delta U \) being the virtual displacements.

\[
\int_V \sigma^T \delta \epsilon \, dV = F^T \delta U
\]

(2.29)

With the inner virtual work, \( \int_V \sigma^T \delta \epsilon \, dV = \delta A_{(i)} \), being the first variation of the strain energy \( \delta \Pi_{(i)} \) and the outer virtual energy \( F^T \delta U = \delta A_{(o)} \) being
the first variation of the negative outer potentail $\delta \Pi_{(o)}$ equation \[2.29\] becomes (see equation \[2.30\])

$$\delta \Pi_{(i)} = -\delta \Pi_{(o)} \quad \rightarrow \quad \delta(\Pi_{(i)} + \delta \Pi_{(o)}) = \delta \Pi = 0 \quad (2.30)$$

For a vertical solid bar under gravitational acceleration the total energy is the sum of inner, $\Pi_{(i)}$, and outer energy, $\Pi_{(o)}$, see equation \[2.31\]

$$\Pi = \Pi_{(i)} + \Pi_{(o)} = \int_0^l \left( \frac{1}{2} ES u(x)^2 - qu(x) \right) dx \quad (2.31)$$

with E being the Young’s modulus, S being the cross section area, q being the load per length and u(x) being the displacement.

The Ritz method uses the ansatz $u(x) = \sum_i a_i \phi_i$ as interpolation function in order to define the variation of the respective potential over the total length of the element. $\phi_i$ are known ansatz functions and $a_i$ unknown coefficients. Inserting this ansatz into equation \[2.31\] $\Pi$ is only depending on the coefficients $a_i$. With the claim that the total Energy needs to be a minimum it follows (see equation \[2.32\])

$$\frac{\partial \Pi}{\partial a_i} = 0 \quad (2.32)$$

This leads to a system of equations similar to equation \[2.28\] which can be solved for the unknown $a_i$ and yields a solution for the problem. So with the method of Ritz an approximate functional for the energy needs to be set up. Due to the fact that in general no exact solutions are obtained this methods can be modified in order to obtain satisfying convergence towards the exact solution. For practical use in FEM the described method of Ritz needs to be modified although the basic ideas remain the same.

Application and further processing of the mentioned approaches for general situations, including dynamics, yields equation \[2.33\]

$$F(t) = M \ddot{U} + C \dot{U} + K U \quad (2.33)$$

With $M$, $C$ and $K$ being constant system matrices, describing the dynamic properties of the system. $M$ is the total mass matrix which is multiplied by the acceleration, $U$, forming an inertia term. $C$ the damping matrix and the velocity $\dot{U}$ express a damping term, while $K$ the already mentioned stiffness matrix and the displacement $U$ express a force-displacement term. Finally $F(t)$ displays the external forces.

At this point it is clear that the numbers of equations to be solved increases with the numbers of elements and nodes. This makes FEM a very complex process requiring computational power.

25
Nowadays many excellent commercial computer programs are available for FEM analysis like ANSYS or ABAQUS. In general an analysis is performed following three main steps:

1. **Preprocessing** In the preprocessing phase the model for the FEM simulation is created. All informations chosen to enter the model are part of the preprocessing. A graphical model is assembled and according material properties defined. If possible often symmetry areas are introduced in order to save computational effort. Constraints and loads are defined and the mesh is constructed.

2. **Solution Phase** During the solution phase a solving method and according accuracy settings are chosen and computed.

3. **Postprocessing** For the evaluation in the postprocessing phase often graphic user interfaces can be used and particular results can be read out.

### 2.4.2 Linear Analysis

For linear Finite Element Analysis solving the system of equations like displayed in the simplified case of equation \[2.28\] may include inversion of the stiffness matrix and further computing for the unknown displacements \(U\). It has to be mentioned that mesh refinement increases the number of matrix elements and complicates the solution process. For models with a small number of nodes and degrees of freedom often direct methods like the elimination method of Gauß are used. In case of larger systems iterative numerical procedures are used to solve the problem. The Lanczos method and therefrom derived methods are used for band matrices or matrices that are sparsely populated. The conjugate gradients method works for large symmetric matrices.

### 2.4.3 Nonlinear Analysis

Nonlinear effects in Finite Element Analysis can occur due to geometric reasons, material properties, boundary conditions or nonlinear effects like buckling. Basically Nonlinear Problems are treated like linear ones. Hence setting up the according equation can be applied like already discussed. The obtained system of equations is nonlinear and can in general only be solved with numerical procedures. In the Standard Newton Raphson method the problem is successively solved by linearization. This means that the according load
Figure 2.7: Standard Newton Raphson method for an incremental iterative procedure of a problem with a nonlinear material model, [5].

is applied in steps, or increments, while a suitable solution is searched with an iterative procedure. For the stiffness matrix this means that it is not constant over the whole process. A systematic sketch of an incremental-iterative procedure, called Standard Newton Raphson method, can be seen in figure 2.7 where the slope at the beginning of step, m, can be interpreted as the stiffness matrix.

2.4.4 FEM Tools

In the following sections some very common tools in FEM for structural analyses are introduced.

Modal Analysis A load independent and linear system is considered. This leads to a homogeneous linear differential equation like equation 2.34 where damping is neglected.

\[ 0 = M \ddot{U} + K U \]  \hspace{1cm} (2.34)

A harmonic ansatz leads to the linear eigenwert problem (see equation 2.35).

\[ (K - \omega^2 M) \phi = 0 \]  \hspace{1cm} (2.35)
With \( \omega \) being the eigenfrequency and \( \phi \) being the eigenvector describing the corresponding mode shape. All results, except the eigenfrequency, \( \omega \), obtained by a Modal Analysis are not scaled.

**Static Analysis** Time independent loads are applied and for the solution no time dependent effects are considered. This approach can be used for situations without time dependence or with negligible time depending effects and low damping.

**Transient Analysis** In case that loads act over a certain period of time, the data describing the acting loads need to be a discrete set. For the application of loads in a simulation, loadsteps are applied. The solution for a certain loadstep, \( i \), forms the initial state for the next loadstep, \( i+1 \). The same relationship within the loadsteps is suggested by the theory of time integration, where time integration means that effects or reactions are summed up over time.

### 2.5 Wire Bonding

A Wire bond is a mean to produce an electrical conductive interconnection and is widely used in microelectronic components. In recent years billions of transistors, LEDs or ICs were produced every year and hence wire bonding experiences a broad field of application. Nowadays about \( 10^{13} \) wire bonds are produced per year.

Wire bonding typically connects two metallic contacts. For the connecting process on either side thermosonic or ultrasonic bonding is used. According to Lu et al. \[26\] and Ciappa et al. \[7\] one of the main failure mechanisms of power semiconductors is the bond wire lift off due to thermo-mechanical fatigue of the interface between wire and the bonding site, called metallization pad. Regarding this problem a brief overview, mainly based on \[13\] and \[15\], about the main topics, challenges and testing methods is given in the following.

#### 2.5.1 Power Semiconductor Architecture

Power semiconductors are used to control electric currents over one Ampere and electric voltage over 25 Volt.

In order to define a power semiconductor we consider an electronic control unit (ECU). Such a device consists of a sensor, a microcontroller and a signal processing unit. The sensor records and transforms an input parameter into
an electric signal. This signal is then processed to a suitable control signal and used by the processing unit, which is a power semiconductor, to control an actuator.

The principal assembly of a power semiconductor consists of several main layers or components. There is a back side contact, often referred as leadframe or lead, providing a basis for the rest of the chip. This layer mainly consists of a metal. The substrate, being a semiconductor is attached to the leadframe by a layer called chip adhesive. The active chip area is part of the substrate and comprises the chips logic and processing unit. This unit is very complex and is made up by several layers, elements and materials. Electrical contact between this part and the leadframe, is often provided by wire bonds. To protect the rather sensitive and complex assembly it is coated with a plastic called mould compound. The finally obtained device is called a package. An example for a typical package and a sketch of the described assembly are depicted in figure 2.8 (a) and (b).

![Figure 2.8: Power semiconductor package (a) and schematic cross section (b), [15].](image)

### 2.5.2 Welding in Bonding Processes

According to the American Welding Society (AWS) welding means the inseparable joining of components by the application of heat or pressure [16]. Common welding processes are based on heating the parts over the liquidus. Modern techniques work without high temperature but provide joining by high pressure and friction. This development in combination with a totally automatic process (auto-bonding) yields very short welding times and plays a crucial role for the production of power semiconductor devices.

Three bonding techniques are commonly used for wire bonding, which are Thermocompression (TC), Ultrasonic (US) and Thermosonic (TS) Bonding. For none of them a complete physical model is available, leaving
this field for empiric descriptions. A brief description of the three techniques is given according to [13].

**Thermocompression Bonding**

Developed at Bell Laboratories in 1957 this method was used till it was replaced by US bonding in the 1960s. TC bonding is a type of solid-phase welding that requires high forces and temperatures. The weldments (wire and bond pad) are plastically deformed, sweeping aside surface contaminants. This leads to an intimate contact between cleaned surfaces. Short-range interatomic forces and heat provide the required activation energy and result in a welded bond. The mentioned temperatures are approximately 300°C and can be too high for certain devices. TC bonding is very sensitive with respect to surface contamination and takes a relative long time. [13]

**Ultrasonic and Thermosonic Bonding**

US bonding is performed by application of ultrasonic energy and a clamping force. The wire is pressed onto the bonding pad by the bonding tool while it is displaced with ultrasonic energy. The bonding tool works with a piezoelectric element, converting electrical energy into mechanical movement and displaces the wire holder or capillary at its edge for up to 100 μm. Operating frequencies can be higher than 100 kHz. Advantages of the US bonding process in comparison with TC bonding are shorter bonding times and lower temperatures. If the US bonding process is supported by heat it is called TS bonding, performed for the first time by Couloulas in 1970 [8]. For TS bonding interface temperatures between 125°C and 220°C are required. This can cause problems with the heat resistance of the microelectronic devices. As advantages the bonding time can be shortened and small displacements are possible although clean surfaces are provided. [13]

Both methods, US and TS bonding, affect the metallic structure of the weldments close to the interface even in monometallic welds. This is shown in various investigations done with transmission electron microscopes (TEM) [23] and scanning electron microscopes(SEM) [17].

**Wedge vs. Ball Bonds**

Wire bonding distinguishes between wedge and ball bonds. An explanation of the different bonding processes for ultrasonic bonding is described in the following and depicted in Figure 2.9 (a) for ball bonding and (b) for wedge
bonding.

Figure 2.9: Ball bonding process (a) and wedge bond process (b), [13].

For the wedge bonding process the adjusted tool-wire arrangement is lowered to its first position and pressed to the bonding site with a certain force. Ultrasonic energy is applied, displacing the wire parallel to the axis, producing the first weld. Then the tool is raised and moved along the wires direction to the second site where the bonding process is repeated, producing the loop. Finally a wire clamp pulls the wire in order to brake it at the wedge bond heel.

During ball bonding the wire is held by a capillary tool. A discharge resulting in an electronic flame-off spark (EFO) is produced. This melts the wire and forms a ball-shaped end. The ball is then pressed to the bonding site and ultrasonic energy is applied, forming the weld. In a next step the tool is raised and moved to the second bond site. There a stitch bond is produced. That means that without an EFO the wire is lengthwise pressed against the bonding pad and the wire is finally broken by pulling or squeezing. The EFO melts the wire and leaves the area right above the ball annealed, called heat affected zone (HAZ). This zone shows coarse grains and is the weakest part of the wire.

Auto-wedge bonders have a limitation with respect to the direction of the second bond while auto-ball bonds don’t show such a restriction. So for
wedge bonds the second bond site needs to be situated in the direction of the wire used for the first bond, what yields longer bonding times for wedge bonds. \[13\]

### 2.5.3 Materials in Wire Bonding

A brief overview about possible material combinations and metallurgical problems according to [13] and [35] is given. Information about metallurgical combinations that have been ultrasonically welded (bonded) with fine wire is shown in table 2.1.

Table 2.1: Metals which have been successfully joined together by ultrasonic welding, [13].

|            | Al | Be | Cu | Ge | Au | Fe | Mg | Mo | Ni | Pb | Pt | Re | Si | Ag | Sn | Ti | W | U | Zr | Pb |
|------------|----|----|----|----|----|----|----|----|----|----|----|----|----|----|----|----|----|----|----|
| Aluminium  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  |
| Beryllium  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  |
| Copper     | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  |
| Germanium  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  |
| Gold       | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  |
| Iron       | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  |
| Magnesium  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  |
| Molybdenum | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  |
| Nickel     | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  |
| Columbium  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  |
| Palladium  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  |
| Rehnium    | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  |
| Silicon    | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  |
| Silver     | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  |
| Tantalum   | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  |
| Tin        | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  |
| Titanium   | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  |
| Tungsten   | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  |
| Uranium    | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  |
| Zirconium  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  | O  |

Examples for important wire-bond couples are Au-Al, Cu-Al, Au-Pd or Au-Au.
The formation of intermetallic compounds (IMC) accompanies the bonding process while Kirkendall voids often occur during the service. An IMC is a solid phase composed by two or more metallic elements. Its crystal structure differs from that of the other constituents. During ball bonding on semiconductor devices, a solid state weld is achieved by thermosonically assisted interdiffusion at the interface between the wire and pad material. Depending on the existence of phases in the according phase diagram and the exposed temperatures, IMC can form at the interface of the wire bond.
The Kirkendall effect can be described by the mismatch of diffusion coefficients of the respective components. The inequality of the resulting diffusion fluxes leads to a mass flow accompanying the interdiffusion process. This causes the diffusion couples to shrink on the one side and swell on the other side. The Kirkendall effect can lead to diffusional porosity, called Kirkendall voids. Exposure to high temperature may amplify the mentioned effects. For welded components as used for wire bonding, the formation of IMC can lead to Kirkendall voids. For bonding of Au wire on Al metallization the IMC will occur in the interface between Au wire and Al metallization. Such components begin to form during the bonding process and grow during any time when high temperatures are encountered. Depending on the stability of the additional phases some layers may grow at room temperature. IMC formation is considered to be a necessary part of Au-Al bonding and is always present. According to the Au-Al phase diagram five IMCs exist: \( \text{Au}_8\text{Al}_3 \), \( \text{Au}_4\text{Al}_2 \), \( \text{Au}_2\text{Al} \), \( \text{AuAl}_2 \) and \( \text{AuAl} \). Mechanical properties of the listed IMC vary from Au and Al and differ among themselves. Lattice parameters, hardness and specific resistance increase for the components with respect to their constituents. The increased hardness (and brittleness) can lead to cracking during temperature cycles or other stress. The presence of IMC does not necessarily lead to bond failure but the interface strength may degrade. There are three main bond failures due to the formation of Au-Al intermetallics. Increasing electrical resistance favours the formation of Kirkendall voids. This can lead to failure by driving the circuit out of its electrical specifications. Voids beneath the bond can degrade the mechanical properties. Void-free bonds are stronger than the pure metals but are also more brittle. Systems with IMC are more likely to fail because of brittle fracture due to temperature cycling induced stresses than pure Au or Al wires.

**Al-Cu Wire Bonding**

Due to the high price of gold, better electrical conductivity, higher resistance to wire sweep during plastic encapsulation and slow velocity of IMC growth, ball bonding of Copper on Aluminium became important. The Copper-Aluminium phase diagram shows five intermetallic compounds: \( \text{Cu}_9\text{Al}_4 \), \( \text{Cu}_4\text{Al}_3 \), \( \text{Cu}_3\text{Al}_2 \), \( \text{CuAl}_2 \) and \( \text{CuAl} \), which reveals the possibility of various IMC similar to the Au-Al system. For certain aging conditions at 150°C to 200°C in air only \( \text{CuAl}_2 \) and \( \text{CuAl} \) components were found. The growth rate of IMC for Cu-Al is less than half of that for Au-Al bonding. Since Cu oxidizes, the bonding process needs to be done in an inert atmosphere.
However the bondability of Cu balls to Al pads has proven to be excellent. \[13\]

### 2.5.4 Testing of Wire Bonds

Bond wires are mostly tested by means of the **pull test** and the **shear test**. These tests work mechanically and can be performed destructively or non-destructively. A short explanation of these tests according to \[13\] is shown in the following.

**Pull Test**

The pull test is the widest used wire bonding test method nowadays. It works good for wedge bonds but hardly tests the right parameters for ball bonds. The wire loop is pulled by a hook what leads to a tension in the wire. A sketch of the process can be seen in figure \[2.10\] (a). By measuring the applied load and simple math several bond parameters can be defined and measured. For wedge bonds the applied force results in peeling the bond from its pad, testing the relevant interface of the bond. The problem for ball bonds is that normally the wire breaks in the heat-affected zone, right above the ball bond. This is an area that is not relevant for the actual bond failures. \[13\]

![Figure 2.10: Schematic sketch of the pull test (a) and the shear test (b), \[13\].](image)

**Shear Test**

Applicable for wedge and ball bonds the shear test was introduced in 1967 \[2\]. A shear tool is sidewise moved to the desired wire bond and a force applied. This yields a shear stress within the bond-pad interface. Recorded stress and displacement can later be evaluated. For further explanation see figure \[2.10\] (b). Package geometry, the shear tools vertical arrangement and the bond properties show some limitations to shear testing. Self explanatory sketches of failure modes are shown in figure \[2.11\] \[13\].
2.5.5 Wire Bonding Related Failures in Power Semiconductors

The dominant failure in power semiconductors under field conditions is the wire bond lift off due to thermo-mechanical stresses. Alternating temperatures also lead to the deformation of the bond wire, which causes repetitive flexing and strains. This flexing causes crack initiation just above the bond foot. Such failure modes can be reduced by avoiding small bending radii of the bonding wire. Further failure reasons include solder fatigue, wire burn out as well as device failures. These modes shall not be discussed in detail within this work.

Several studies on wire bond failure (see [13] [12] [7]) were performed using thick wires and mainly wedge bonds up to 700 µm wire thickness. The observed effects can be used as a basis for investigating related effects for thin ball bonded wires.

2.5.6 Advanced Testing and Desired Developments

The requirements of power semiconductor devices go towards higher power density and better performance of power switches. Although showing several disadvantages like price, crosstalk and inductance wire ball bonds are likely to continue playing an important role in packaging industry. Current ball bond testing techniques do not satisfy the industries needs. Power cycling and temperature cycling are time consuming. The pull test is not suitable for ball bond testing, while it works for wedge bonds. Shear testing the bond yields better results but shows several failure modes. A common problem for pull and shear test is the fact that both methods expose the samples to static loading, while bond connections experience cyclic loading under field conditions.
Thus new and fast methods for thin wire ball bond testing are desirable. Regarding this promising investigations were conducted using larger wedge bonds [12] [18].
Chapter 3

Experimental

Experiments were performed on two different types of commercial power semiconductors. The first sample included 34 copper ball bonds on aluminium pads, named Tech A hereafter, while the second sample included one copper ball bond on an aluminium pad, named Tech B hereafter.

This chapter comprises the experimental work, FEM simulations and the lifetime analysis. It is divided into nine main sections. The first four parts, 3.1 Ultrasonic Fatigue Testing, 3.2 Accelerated Testing of Micro-Joints, 3.3 Finite Element Modelling for Accelerated Testing of Micro-Joints and 3.4 Life-time Modelling describe the methods used for the two technologies, Tech-A and B. Sections 3.5 Experimental - Tech A and 3.6 Experimental - Tech B include detailed reports about the experimental procedure, FEM simulations and the obtained results for Tech-A and B, respectively. The lifetime models are compared in section 3.7 Comparison of Tech A and Tech B Samples.

3.1 Ultrasonic Fatigue Testing

According to Roth [30], ultrasonic fatigue testing involves cyclic stressing of material at high frequencies. The advantage of the ultrasonic method is the ability to provide fatigue data within a reasonable length of time. High-frequency testing enables rapid evaluation of high-cycle fatigue data and the determination of fatigue limits.

This section assesses the underlying concepts and basic techniques for conducting ultrasonic fatigue testing. Explanations are given according to Roth [30].
3.1.1 Frequency and Time Compression

Ultrasonic fatigue testing applies loads with a high frequency in order to accumulate high numbers of cycles in a reasonable time period. A conventional fatigue test at 1 Hz would need 32 years to accumulate $10^9$ cycles. At an ultrasonic frequency of 20 kHz this test would take 14 hours. While kHz frequencies are advantageous for high numbers of cycles, the minimum number of reasonable applicable cycles is limited to $10^5$. Accelerated test methods alter testing conditions. The influence of frequency and strain rate on cyclic material behaviour must be comprehended. Depending on the material, fatigue data recorded by ultrasonic methods may only slightly differ from data recorded with the use of more conventional methods [6][9][30].

3.1.2 Testing Principles

Ultrasonic fatigue testing is a resonant test method, in which a displacement wave is generated in a resonant specimen. This wave is produced by a transducer that is vibrating at the same frequency as the natural frequency of the mounted specimen. Displacement and strain propagate in a bar shaped material that is subjected to resonant acoustic loading. A sound wave injected longitudinally into one end of a bar of uniform diameter and length L, travels at a certain velocity through the bar, is reflected at the opposite end of the bar and travels back to its point of origin. The velocity of wave, c, is with some simplification (see equation 3.1)

$$c = \sqrt{\frac{E}{\rho}}$$

with E being the Young’s modulus, and ρ being the material density. If the time necessary for the wave to move to and fro the bar is equal to the period of the injected sound wave, a standing wave develops and the bar will be in resonance. The length of the bar will be equal to the half wavelength of the sound wave. For the displacement, u(x), along the bar we get equation 3.2

$$u(x) = u_0 \cos(kx)$$

with $u_0$ being the displacement at the end of the bar and $k = \frac{2\pi}{\lambda}$ the wave number including the wave length, λ. According to equation 3.2, the strain distribution, $\epsilon(x)$, can be computed like shown in equation 3.3

$$\epsilon(x) = \frac{du(x)}{dx} = k \ u_0 \ sin(kx)$$
The maximum displacement occurs at $|\cos(kx)| = 1$, or $x = 0$ and $x = \frac{\lambda}{2}$, while the maximal strain occurs at $|\sin(kx)| = 1$, or $x = \frac{\lambda}{4}$. Figure 3.1 shows the distribution of longitudinal displacement and strain of a bar with length $\frac{\lambda}{2}$ and demonstrates that maximum displacement (displacement antinode) occurs at zero strain (strain node) and vice versa. Along with equation 3.3 the stress distribution can be calculated, see equation 3.4

$$\sigma(x) = E\epsilon(x) = Eku_0 \sin(kx)$$

(3.4)

Cyclic straining can be produced in a bar at any desired resonance frequency by choosing the length of the bar correctly. For bars with uniform cross-section the desired length for fatigue testing will be $\frac{\lambda}{2}$. A difference between a conventional fatigue testing method and the resonant method is that the distribution of cyclic strain and stress varies from 0 to maximum over the length of the sample, rather than being constant along the length of the sample.

The system described for the uniform resonant bar can be applied to ultrasonic fatigue testing. These principles can be used to design the mechanical part of the converter, the acoustic horn, and the test specimen or sample holder.

3.1.3 Testing Equipment and Setup

The typical components of an ultrasonic fatigue testing system are the energy supply and the wave train. The latter is composed of the sonic energy converter, an acoustic amplifying horn and the test specimen or sample
holder. For running and monitoring the process, components like amplitude, frequency, and cycle measuring tools as well as cooling systems are used. For ultrasonic fatigue testing typically applied electric power ranges from 500 to 4000 Watt.

The electrically excited converter generates vibrations. Due to the used setup of the resonant wavetrain this leads to a standing wave and a cyclic displacement at the end of the sample holder. In figure 3.2, the displacement and strain distribution along the wave train are depicted. For generating the ultrasonic displacement mainly piezoelectric driven devices are used. Displacements achieved by this technique range from 0.01 to 0.02 mm at 20 kHz.

Acoustic horns transmit the developed displacement from the sonic converter to the specimen. These components are placed in the wave train to raise the strain amplitude in the specimen to the required level. Hence acoustic horns are bars of resonant length with continuous or discontinuous cross sections. Due to continuity reasons the vibration amplitude can be increased in case of a reduction of the cross section and vice versa. This yields the possibility of scaling the displacement and the strain. For simple horn geometries the amplification or attenuation follows equation 3.5,

$$u_{output} = \frac{Area_{input}}{Area_{output}} u_{input}$$  \hspace{1cm} (3.5)

where $u_{output}$ and $u_{input}$ are the output and input displacement amplitudes, respectively. [30]
3.2 Accelerated Testing of Micro-Joints

In the following the experimental setup for the accelerated testing of micro-joints is described based on [19]. Related setups were used for the samples called Tech-A and B.

3.2.1 Experimental Procedure

An ultrasonic fatigue testing setup as described in section 3.1 is used to expose metallic micro-joints to fatigue [19]. The resonant length, l, of the sample holder can be calculated with equation 3.1.

The micro-component specimen is attached to the free end of the sample holder (place of maximal displacement). Coupling between the sample holder and an active mass is provided solely by the micro-joint. During an active test, the holder is forced to longitudinal vibrations and the active mass is excited and displaced as well. Due to its inertia the active mass experiences an acceleration relative to the holder. Hence a cyclic shear stress is introduced into the micro-joint. A schematic of a possible used setup and the principles of the testing procedure is depicted in figure 3.3.

![Figure 3.3: Sketch of the micro-component attached to the sample holder, [19].](image)

The shear stress in the micro-joint depends on the active mass, the stiffness of the micro-joint, its geometry and the applied energy. For estimating the average shear stresses in the micro-joint, the relations of equations 3.6 and 3.7 can be used.

\[ F = m_{active} \cdot a \]  \hspace{1cm} (3.6)
\[ \tau = \frac{F}{A} \]  \hspace{1cm} (3.7)

F is the force due to the acceleration, a, acting on the mass, \( m_{active} \). The related average shear stress, \( \tau \), can be calculated with the contact area, A,
of the micro-joint. Accelerations are imposed by the moving sample holder and can be derived via computing the derivative of the displacement, \( \frac{\partial^2 u}{\partial t^2} \bigg|_{x=\lambda} \), or velocity, \( \frac{\partial \nu}{\partial t} \bigg|_{x=\lambda} \), at the free end of the sample holder with respect to time.

According to a harmonic oscillator driven by an external sinusoidal force, the phase shift between the external force and the movement of the oscillator converges to zero for eigenfrequencies considerably higher than the external driving frequency of the oscillator. In case that the relevant eigenfrequency of the prepared micro-component sample is higher than the testing frequency of the resonant fatigue setup, resonance effects between sample holder and active mass can be neglected.

The peak acceleration value is given by equation 3.8.

\[
a_{\text{max}} = 2\pi f \nu_{\text{max}}
\]  

With \( f \) being the frequency and \( \nu_{\text{max}} \) being the peak velocity.

Failure of the samples at the interconnect is called lift off, which is similar to the failure mode under actual service conditions. In a first approach it is necessary to find a suitable range of amplitude where cycles between \(10^5\) and \(10^9\) can be recorded and fatigue data can be obtained.

Lift off mass and the fracture surface area are determined after fatigue fracture and can be used to calculate estimated average forces and shear stresses according to equations 3.6 and 3.7. The test results can be used to plot Wöhler curves.

This method only provides average shear stresses. There are no experimental means in order to directly measure the stress in these extremely small joint. Thus Finite Element Simulation provide an efficient tool for calculation of stresses and strains.

In high cycle fatigue, where plastic strain does not play an important role the Basquin relation (see equation 3.9) can be used in order to describe the lifetime.

\[
\sigma_a = C (N_f)^b
\]

Determination of the coefficients \( C \) and \( b \) determines the fatigue data collected for the test specimen interconnect and enables comparison with tabulated values.

### 3.2.2 Determination of Acceleration

Exact knowledge about the movement of the sample holder and the active mass is absolutely necessary. For this experiment the Laser-Doppler-Vibrometer (LDV) was used to determine the velocity of the sample.
Simultaneously a strain gage was used to record strain values at the mid section of the sample holder. This procedure made it possible to correlate velocities measured by the LDV with strain values of the sample holder measured by a strain gage. This was done because strain values are easier to obtain during an experiment.

The described method shows a reliable and fast method to determine velocities \[19\].

**LDV Application**

Acceleration of the specimen and the holder can be determined by application of a LDV. The velocity of the vibrating micro-component is measured parallel to the direction of motion. This procedure yields the peak velocity, the vibration frequency and enables the calculation of the peak acceleration according to equation \[3.8\].

**Strain Measurement**

Strain gages are used to measure the strain of an object. They consist of a metallic foil pattern attached to an insulating flexible background. The metallic foil pattern can be described by a wire that is folded several times. Attaching the strain gage to the object is done by a suitable adhesive. Characteristic for a strain gage is the k-factor, linking the measured voltage caused by the changed resistivity and the strain. To calculate a strain from a measured voltage a relationship like equation \[3.10\] can be used.

\[
\epsilon = \frac{\Delta U}{k \cdot f}
\]  

(3.10)

Where \(\epsilon\) is the strain, \(\Delta U\) the voltage change, \(k\) the mentioned k-factor and \(f\) the amplification factor. In order to monitor the signal and the calculated values, the strain gage output can be displayed with an oscilloscope.

### 3.3 Finite Element Modelling for Accelerated Testing of Micro-Joints

Analysis of stress and strain distribution with the help of the Finite Element Method (FEM) has become a very useful tool in the last years. Several publications include detailed analysis of bond connections using Finite Elements \[12\][26]. FEM was also used to evaluate accelerated testing of other micro-joint experiments \[15\].

43
For this experiment FEM simulation was performed to clearly determine the validity of the sample geometry, and to determine stress values for the lifetime estimation. 3-dimensional simulations were carried out with the FEM code ANSYS. In the following a general description of methods used for the simulations of both samples, Tech-A and Tech-B, is given.

### 3.3.1 Material Models

In order so simulate the response of a sample to an external load, the material properties must be known.

The reaction of materials due to repeated loading was discussed in section 2.1.4. For computer simulations the obtained models need to be expressed via linearized relationships. Depending on the applied stress level and the simulation purpose adequate material models can be chosen. See a description of the relevant material models in the following.

**Linear Material Model**

For modal analyses and simple structural simulations with stresses well below plasticity a pure linear model is used. For a mathematical description of this elastic model the Young’s Modulus, $E$, and the Poisson’s ratio, $\nu$, is used (see figure 3.4 (a)).

**Bi-Linear Material Model**

Covering plasticity and suitable for many materials is the bi-linear material model. It contains an elastic region, described by the Young’s Modulus, $E$, and a plastic region described by the yield stress, $\sigma_{\text{yield}}$, and the tangent modulus, $T$, as described in figure 3.4 (b). In general one speaks of the tangent modulus as the slope of the stress strain curve. It may be compared to the Young’s modulus in the elastic region.

**Multi-Linear Material Model**

If there is need for a more sophisticated model a multi-linear material model can be used. It is again divided into an elastic part and a plastic part. The stress-strain curve in the plastic range is described by a table, making it possible to match the material model with a set of straight lines, see figure 3.4 (c). Bi-linear and multi-linear material models can be extended with isotropic or kinematic hardening effects.
3.3.2 Geometry

For measuring the dimensions of the sample a light microscope with scaling function is used. Horizontal dimensions are measured at the unmolded sample without further preparation in a top view. Vertical dimensions are obtained with cross-section grindings. The same procedure is conducted for the geometry of the prepared sample, as it is used for the experiment. Collected dimensions are assembled in a graphical 3D modeller and yield a valid three dimensional reconstruction of the used samples.

In order to design a suitable geometry for the FEM analysis, complicated geometries are simplified in a manner that modifications do not affect the result accuracy. From a structural point of view it is possible to split the samples into several main volumes. The observed main volumes and their properties for both samples, Tech-A and Tech-B, are listed below. For more detailed information about volume geometry of the respective sample see the sections including the individual simulations for Tech-A and Tech-B.

1. **Leadframe → Tech-A & Tech-B**

   The leadframe was observed at both samples, Tech-A and Tech-B, and is made of copper. Result accuracy and expected stress intensities are not high and a linear material model was chosen for the leadframe (see table 3.1).

   \[
   \sigma = E \cdot \varepsilon \\
   \sigma = \sigma_{elast} + \sigma_{plast} \\
   \sigma_{plast} = T \cdot \varepsilon_{plast} \\
   \sigma_{elast} = E \cdot \varepsilon_{elast} \\
   \Delta \sigma = \sigma_{plast} + \Sigma \Delta \sigma
   \]

   ![Figure 3.4: Linear (a), bilinear (b) and multilinear (c) material models](image)

Table 3.1: Material model - Leadframe (Cu) → Linear

<table>
<thead>
<tr>
<th>$\rho$ [^{kg/m^3}]</th>
<th>$E$ [^{MPa}]</th>
<th>$\nu$ [^{-}]</th>
</tr>
</thead>
<tbody>
<tr>
<td>8920</td>
<td>97.6</td>
<td>0.33</td>
</tr>
</tbody>
</table>
2. **Chip Adhesive** → Tech-A & Tech-B

The chip adhesive is made of Solder (Sn63 Pb37) and was observed for both samples. Result accuracy and expected stress intensities suggest a linear material model (see table 3.2).

**Table 3.2: Material model - Chip Adhesive (Sn63 Pb37) → Linear**

<table>
<thead>
<tr>
<th>$\rho$ [kg/m³]</th>
<th>$E$ [MPa]</th>
<th>$\nu$ [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>8800</td>
<td>31.4</td>
<td>0.33</td>
</tr>
</tbody>
</table>

3. **Substrate** → Tech-A & Tech-B

The substrate is made of silicon and common for both samples. Lithographically applied substructures are neglected and the principal shape of this volume is a cube. Mainly coincident with the real behaviour of silicon a purely linear material model was chosen (see table 3.3).

**Table 3.3: Material model - Substrate (Si) → Linear**

<table>
<thead>
<tr>
<th>$\rho$ [kg/m³]</th>
<th>$E$ [MPa]</th>
<th>$\nu$ [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>2330</td>
<td>130.0</td>
<td>0.33</td>
</tr>
</tbody>
</table>

4. **Metallization** → Tech-A & Tech-B

The metallization is made of Aluminium and shows dimensions of a thin coating with 6 µm thickness in both samples. It is modelled as a layer with constant thickness all over the substrate. Variations from the constant thickness due to the bonding process were not taken into consideration. A bi-linear material model with kinematic hardening was chosen (see table 3.4).

**Table 3.4: Material model - Metallization (Al) → Bi-linear**

<table>
<thead>
<tr>
<th>$\rho$ [kg/m³]</th>
<th>$E$ [MPa]</th>
<th>$\nu$ [-]</th>
<th>$\sigma_{yield}$ [MPa]</th>
<th>$T$ [MPa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>2680</td>
<td>68.0</td>
<td>0.33</td>
<td>137.0</td>
<td>100.0</td>
</tr>
</tbody>
</table>

5. **Nailheads** → Tech-A & Tech-B

34 nailheads are distributed over the metallization of Tech-A and one

---

1 With permission from Infineon Technologies AG
nailhead was placed at the metallization of Tech-B. Their shape was modelled with the help of a torus with an outer radius of 130 or 140 \( \mu \text{m} \). A multilinear material model with kinematic hardening was chosen (see table 3.5).

Table 3.5: Material model - Nailheads (Cu) → Multi-linear

<table>
<thead>
<tr>
<th>( \rho ) ( [\text{kg} / \text{m}^3] )</th>
<th>( E ) ( [\text{MPa}] )</th>
<th>( \nu ) [-]</th>
<th>( \sigma_{\text{yield}} ) ( [\text{MPa}] )</th>
<th>( \sigma_{\text{UTS}} ) ( [\text{MPa}] )</th>
</tr>
</thead>
<tbody>
<tr>
<td>8920</td>
<td>97.6</td>
<td>0.33</td>
<td>287.9</td>
<td>370.5</td>
</tr>
</tbody>
</table>

### 3.3.3 Mesh and Model

A model in FEM Analysis contains all properties that are chosen to describe the original. Also the mesh, boundary and symmetry conditions are part of a FEM model.

Often used to save computational time are symmetry regions. Inserting a mirror symmetry plane where an according symmetry is observed reduces the number of nodes to be computed.

In order to obtain an effective model with respect to result accuracy and computational time, adjustment of the discretization, called mesh, according to accuracy is important. Subdivision of original volumes makes it possible to obtain fine meshed regions where necessary and rather coarse meshed areas where low accuracy is required. For both samples this method was used to provide a nice transition from very fine meshed areas to rather coarse meshed areas. To reconstruct the properties of the model before subdivision, special attention is payed to the contact properties of adjacent volumes. These contact pairs define the behaviour of surfaces with respect to each other. A glued contact means that the contact pair shows perfectly rigid properties. The volumes are bonded without the possibility of separation due to loads. For special purposes it is further possible to define gliding contact pairs. The surfaces of the adjacent volumes are able to glide over each other depending on a friction coefficient but formation of a gap is prevented. A modification of this contact property is a frictional contact pair with the ability to glide and form a gap.

Graphical examples of the described methods are depicted in the chapter about FEM simulation of the respective sample.
3.3.4 Loads

Depending on the applied analysis type it is tried to simulate the inertia loads which act on the sample during the fatigue experiment. In the experiment the sample is attached to the sample holder and periodically deflected. Due to the inertia of the sample stresses are imposed. In the simulation the same effect can be achieved via fixing the sample spacially and applying a global acceleration, consistent with the experimentally measured displacements.

Static Analysis is a standard tool in ANSYS. Loads or global peak accelerations are applied without any time time dependence.

For Transient analysis a load needs to be defined as a function of time. The peak acceleration of the sample can be determined via a LDV or a strain gage. Displacement, velocity and acceleration follow sinusoidal functions in time. The cycle period is equal to \( \frac{1}{f} \), where \( f \) is the testing frequency. For an FEM simulation it is necessary to express the load by a set of numerical values. This can be done by dividing a sine into a certain number of steps, \( n \), like \( \frac{2\pi}{n} = \Delta \phi \). The acceleration at a certain step \( i \), \( a_i \), can then be obtained by computing the value of the sine and multiplying it with the peak value, \( a_0 \), see equation 3.11.

\[
a_i = a_0 \cdot \sin(i \cdot \frac{2\pi}{n}) = a_0 \cdot \sin(i \cdot \Delta \phi)
\]  

(3.11)

The subdivisions form load-steps for the transient analysis. Load-steps are solved in a manner that the solution of load-step \( i \) forms the initial state of the upcoming loadstep \( i+1 \). Length of time, \( \Delta t \), of one load-step depends on the testing frequency, \( f \), and the number of Load-steps, \( n \), see equation 3.12.

\[
\Delta t = \frac{1}{fn}
\]  

(3.12)

3.3.5 Geometry Check

The correctness of the geometry is checked because resonance effects and preparation irregularities can influence the experimental results. In order to exclude resonance effects a Modal Analysis is performed. This means a load-independent and not scaled analysis, yielding eifenfrequencies, \( f_{\text{eigen}} \), and corresponding mode shapes. If the computed eigenfrequencies are much lower than the testing frequency of 20 kHz, resonance effects can be neglected. To ensure this, Transient Analyses can be performed over several periods of the applied loads. The recorded results are checked for resonance effects and the existence of a stable solution. In case that both

48
simulations, Modal & Transient, do not yield evidence for resonance effects, the experimental setup is confirmed and for later transient analysis only one period of loading is performed. To determine the influence of sample preparation irregularities on the recorded data it is possible to run simulations while certain properties are varied. ANSYS shows the possibility to define parameters that are simulated one after the other in a consecutive analysis.

3.3.6 Stress Determination

The exact knowledge of stress values is crucial for the lifetime prediction. To describe the FEM calculated stress state, experimentally applied loads are simulated. The maximum of the according von Mises stress is computed to be further used for the lifetime prediction.

In general estimated average shear stresses are lower than FEM calculated von Mises stresses at stress concentrations. This may be due to neglecting of complex geometry effects and stress concentrations of the estimated approach. Another reason is the definition of the von Mises stress. Assuming a stress state with only one shear stress, $\tau_{xy}$, acting on a solid body, the relation of shear stress and von Mises stress according to equation (2.16) becomes $\sigma_v = \sqrt{3} \cdot \tau_{xy}$. If stress components of a general stress state are neglected and only one relevant shear stress is chosen in order to describe the stress state (like done by the estimated approach in equation (3.7)), the relation of shear stress and von Mises stress reads as $\sigma_v \geq \sqrt{3} \cdot \tau_{xy}$.

3.4 Lifetime Modelling

High cycle fatigue data, displayed in a Wöhler curve, are phenomenologically well described by the Basquin relation, see section 2.3.1. The stress can be expressed with estimating calculations or by FEM simulation. The Basquin relation can successfully be applied to a loading regime up to $1E+9$ cycles to failure.

For the lifetime estimation of accelerated testing of micro-joints two methods were used in this work. A brief description of lifetime modelling using estimated stresses and FEM is given below. Related methods were performed by Khatibi et al. [18].
3.4.1 Life Time Modelling → Estimated Stresses

Wöhler curves for high cycle fatigue can be fitted with a power law, 
\[ \sigma_{\text{estimated}} = C \cdot (N_f)^b \], with \( \sigma_{\text{estimated}} \) being the average stress. The obtained coefficients, \( C \) and \( b \), fully describe the lifetime of the investigated sample and enable the calculation of any lifetime, \( N_f \), at any estimated stress, \( \sigma_{\text{estimated}} \).

3.4.2 Life Time Modelling → FEM calculated Stresses

For lifetime modelling on wire ball bonds the precise evaluation of stress is important. Estimated stresses do not include geometry effects and stress concentrations. On basis of a purely elastic material model, a stress intensity factor would overestimate the calculated stress in the bond interconnection [18]. Thus a fully transient FEM analysis including plasticity of the accelerated micro-joint test can be performed. The maximum von Mises stress calculated by means of FEM within the bond interconnection may be interpreted to cause crack initiation. This can worsen the mechanical bond properties and so be responsible for an accelerated fracture.

Respective von Mises stress values need to be correlated with the experiment in order to express the Basquin relation in terms of FEM calculated stresses. For the correlation estimated shear stress values, \( \tau_{\text{est-model}} \), are calculated with model properties of the simulation, \( m_{\text{frac-model}} \), \( a_{\text{model}} \) and \( A_{\text{frac-model}} \). The computed shear stress values from the simulation can be linked to the cycles of failure, \( N_{f-model} \), with the experimentally obtained Basquin relation. This procedure yields a stress/cycles to failure couple, \( (\tau_{\text{est-model}} / N_{f-model}) \) where the \( \tau_{\text{est-model}} \) describes the FEM simulation and can thus be exchanged with the von Mises stress according to the simulation, \( \sigma_{\text{FEM}} \). Executing this method for two different loads of the simulation, yields two equations with two unknown coefficients, see equation 3.13.

\[
\sigma_{\text{FEM}-1} = C_{\text{FEM}} \cdot (N_{f-1})^{b_{\text{FEM}}} \quad \text{&} \quad \sigma_{\text{FEM}-2} = C_{\text{FEM}} \cdot (N_{f-2})^{b_{\text{FEM}}} \quad (3.13)
\]

Solving yields the coefficients of the new Basquin relation. The by this procedure obtained coefficients, \( C_{\text{FEM}} \) and \( b_{\text{FEM}} \), fully describe the lifetime of the investigated sample in terms of FEM calculated maximal von Mises stresses.

3.5 Experimental - Tech A

Samples named as Tech A are devices with 34 copper nailheads on aluminium pads. The experiments regarding this sample were performed by Alice.
Lassnig and published in [24] and [33]. Within this chapter an overview about the sample, sample preparation method, the experimental procedure and the obtained results is given. Finite Element Simulation and Lifetime Modeling is discussed in more detail since they were performed by the author. An ultrasonic fatigue testing setup as described in section 3.1 was used to subject metallic micro-joints to fatigue loading. Testing was done according to section 3.2. For more details regarding the experimental procedure see [24].

3.5.1 Sample and Sample Preparation → Tech A

Tech-A samples were provided in an unmolded condition. That means that the samples were removed from production before encapsulation with the mold compound. Copper wire thickness was 50 µm and Aluminium pad thickness was 6 µm. Wire bonding produced a ball bond with a diameter of 130 µm and a height of 50 µm.

In order to obtain sufficient high stresses required for fatigue damage in the bond interface it was necessary to prepare the samples. This was achieved by application of a special soldering technique as described in the following.

Preparation

The copper wires of the unmolded sample were cut in a manner that only the nailheads remained at the chip.

Using special templates a thin copper sheet was soldered to the nailheads. Due to the good reactivity of copper and solder the nailheads were strongly bonded to the solder layer while the aluminium metallization showed no bonding to the solder due to lack of wetting. A sketch of the sample preparation can be seen in figure 3.5 (a).

3.5.2 Experimental Procedure → Tech A

For the fatigue testing the prepared Tech-A sample was attached to a sample holder and mounted to an ultrasonic fatigue system. During the loading the sample holder and the attached micro-component were excited to longitudinal vibrations. Due to its inertia the chip experienced an acceleration relative to the moving sample holder. Hence cyclic shear stress was induced in the interconnect area. Coupling between the copper sheet, which was glued to the sample holder, and the chip was provided by 34 ball bond interconnects. The desired fracture, called lift off, occured in the interface of the copper ball bond to the aluminium pad metallization. A sketch of the desired
lift off can be seen in figure 3.5 (b). The interface of the ball bonds and the chip acts as load-bearing area.

For the estimation of forces and average shear stresses the lift off mass, the acceleration acting on the chip and the fracture surface must be known.

The lift off mass, \(m_{\text{liftoff}}\), is the mass of the chip without the mass of the nailheads. In order to obtain exact stresses, the lift of mass was measured after fracture with a precision scale.

Accelerations of the chip, \(a_{\text{chip}}\), were measured with a LDV.

The fracture surface area, \(A_{\text{frac}}\), consisted of the interconnection area of the 34 nailheads to the metallization pad and was determined after fracture with a microscope.

The average value of shear stress in the interface of the bond interconnection can be approximated with the values of \(m_{\text{liftoff}}\), \(A_{\text{frac}}\), \(a_{\text{chip}}\) and equations 3.6 to 3.7 (see equation 3.14).

\[
\tau_{\text{estimated}} = \frac{F}{A_{\text{frac}}} = \frac{m_{\text{liftoff}} \cdot a_{\text{chip}}}{A_{\text{frac}}} \tag{3.14}
\]

With the described procedure it was possible to determine fatigue life curves of multiple metallic ball bond interconnections of copper wires to aluminium pads for Tech-A.

### 3.5.3 Experimental Results → Tech A

Experimental results obtained by Lassnig [24] which were used for lifetime modelling are shown and explained in this section. Further outcome of experimental procedures done by Lassnig are not mentioned.
Sample Specifications and Test Conditions

Typical sample specifications and test conditions like mass, fracture surface and acceleration are listed below.
Lift off masses were measured after fracture. The mean value for the lift off mass was $m_{\text{liftoff}} = 36.8 \pm 0.6$ mg.
Fracture Surface areas were investigated with a Scanning Electron Microscope. The range of recorded values for the fracture surface area was $A_{\text{frac}}^\text{min} = 0.13 \text{ mm}^2$ to $A_{\text{frac}}^\text{max} = 0.34 \text{ mm}^2$.
Acceleration was calculated from velocity measurements by a LDV. Applied accelerations ranged from $a_{\text{chip}}^\text{min} = 100 \text{ km/s}^2$ to $a_{\text{chip}}^\text{max} = 374 \text{ km/s}^2$.
A summary of the sample specifications and test conditions can be seen in table 3.6.

<table>
<thead>
<tr>
<th>$m_{\text{liftoff}}$</th>
<th>$\Delta m_{\text{liftoff}}$</th>
<th>$A_{\text{frac}}^\text{min}$</th>
<th>$A_{\text{frac}}^\text{max}$</th>
<th>$a_{\text{chip}}^\text{min}$</th>
<th>$a_{\text{chip}}^\text{max}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>[mg]</td>
<td>[mg]</td>
<td>[$\text{mm}^2$]</td>
<td>[$\text{mm}^2$]</td>
<td>[$\text{km/s}^2$]</td>
<td>[$\text{km/s}^2$]</td>
</tr>
<tr>
<td>36.8</td>
<td>0.6</td>
<td>0.13</td>
<td>0.34</td>
<td>100</td>
<td>374</td>
</tr>
</tbody>
</table>

Inserting the values from table 3.6 into equation 3.14 yielded estimated shear stress values in the range of (see equation 3.15)

$$18.4 \text{ MPa} \leq \tau_{\text{estimated}} \leq 96.2 \text{ MPa} \quad (3.15)$$

(standard errors were not taken into consideration).

Shear Test

Shear testing as explained in section 2.5.4 for Tech-A samples yielded a mean value of shear strength of $175 \pm 40$ MPa for the tested Cu-Al interconnect [24].

Fatigue Data and Wöhler Curve

The fatigue life data for Tech-A is presented in figure 3.6 showing estimated average shear stress values (see equation 3.14) versus numbers of cycles to failure. The number of cycles to failure ranges between $N_f = 10^5$ to $N_f = 10^9$. White marks framed in blue indicate samples without rupture during the test, called runouts.
Average shear stress values ranged from 23 to 50 MPa and showed strong scattering. This may be explained by variations of the shear strength (175 ± 40 MPa) due to the nature of the bonding process and irregularities of the sample preparation.

The estimated average shear stresses does not include geometry effects or complex stress states. This may lead to underestimation of stresses, like observed by Khatibi et al. [18]. Therefore FEM analysis of the shear fatigue experiment using a precise geometrical model of the sample including plasticity was performed.

On the basis of the presented results a lifetime prediction model for ball bonded interconnects of Cu wire on Al metallization is proposed in the following.

### 3.5.4 Simulation → Tech A

FEM analyses were performed in order to evaluate the validity of the geometry of the sample Tech-B and to calculate exact stress distributions of conducted experiments. The 3-dimensional simulations were carried out
with the FEM code ANSYS. In the following the performed simulations are presented.

Volumes and Geometry

The sample dimensions were determined with a light microscope and assembled to a three dimensional reconstruction of the used Tech-A sample. For the FEM analysis, the geometry was simplified. Main volumes and their material properties are listed in section 3.3.2. Due to the fact that the copper nailheads were embedded into a solder layer in order to fix them an additional volume, called solder embedding, was introduced. A brief description of the used volumes is given.

1. **Leadframe - (Cu)** A rectangular shaped volume, 2700/6450/207 µm (width, length, height), with material properties according to table 3.1.

2. **Chip Adhesive - (Sn63 Pb37)** A thin solder layer, 1920/4010/12 µm, with material properties according to table 3.2.

3. **Substrate - (Si)** A rectangular shaped volume, 1920/4010/215 µm, with material properties according to table 3.3.

4. **Metallization - (Al)** A 5 µm layer, 1920/4010 µm (width, length), coated onto the substrate with material properties according to table 3.4. In order to provide a nice mesh transition from the large rectangular shaped metallization to the small round shape of the nailheads a transition by subdividing the metallization was executed. Small squares with cylinders were placed under every nailhead.

5. **Nailheads-(Cu)** 34 nailheads with an outer diameter of 130 µm and a height of 50 µm were distributed over the metallization with material properties like in table 3.5.

6. **Solder embedding** According to the sample preparation of Tech-A the nailheads were embedded into a solder layer made of Sn62 Pb36 Ag2. A rectangular shaped volume with dimensions of 1920/4010/100 µm was modelled. To provide a nice mesh from the solder embedding volume to the small nailheads the solder layer was further subdivided into small cubes at the places of the nailheads. A bi-linear material model with kinematic hardening was chosen (see table 3.7).
Table 3.7: Material model - Solder Embedding (Sn62 Pb36 Ag2) → Bi-linear

<table>
<thead>
<tr>
<th>$\rho$ [kg/m$^3$]</th>
<th>E [MPa]</th>
<th>$\nu$ [-]</th>
<th>$\sigma_{yield}$ [MPa]</th>
<th>T [MPa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>8825</td>
<td>31.4</td>
<td>0.33</td>
<td>72.0</td>
<td>10.0</td>
</tr>
</tbody>
</table>

Model and Mesh

The geometry of the model is depicted in figure 3.7 (a) and (b). Because of the symmetry of the chip, a mirror symmetry plane was inserted, indicated in red. The sample was spatially fixed according to the idea in section 3.3.4, indicated in blue.

Discretization was achieved by an adjusted mesh method. As mentioned volumes were subdivided according to the needs of the mesh refinement. Special attention was payed to the contact properties of adjacent volumes. The contact area between the solder embedding (and its sub-volumes) and the metallization (and its sub-volumes) was realized as a frictionless gliding pair of planes. Penetration of the according volumes was prevented whereas opening of a gap between the two pairs was enabled. All other contacts were glued (perfectly rigid contact pair). See a picture of the mentioned ideas in figure 3.8 (a) and (b).

Loads

In the simulation the loads due to inertia forces were modelled as spacial constant acceleration field while the sample was fixed using boundary conditions. To evaluate the coefficients in the Basquin relation it was necessary to evaluate the stresses at two different acceleration amplitudes.
The experimentally measured acceleration amplitudes ranged from $100 \, \text{km/s}^2$ to $374 \, \text{km/s}^2$. For simulation purposes two acceleration amplitudes were chosen at $a_1 = 180 \, \text{km/s}^2$ and $a_2 = 201 \, \text{km/s}^2$. Along with section 3.3.4 the accelerations, $a_1$ and $a_2$, were used as maximum values for Static and Transient Analysis. For the transient load a sinusoidal curve was chosen with 25 load-steps, $n=25$. Equations 3.11 and 3.12 with $n=25$ and $f=20 \, \text{kHz}$ yield numerical values for the simulated loads.

$$\Delta t = \frac{1}{20.000 \cdot 25} = 2 \, \mu s \tag{3.16}$$

$$a_{1-i} = 180 \cdot \sin(i \cdot \frac{2\pi}{25}) \quad \& \quad a_{2-i} = 201 \cdot \sin(i \cdot \frac{2\pi}{25}) \tag{3.17}$$

**Geometry Check**

In order to be able to neglect resonance effects, a Modal Analysis and a Transient Analysis were performed. If the eigenfrequencies, $f_{eigen}$, are much higher than the testing frequency of $20 \, \text{kHz}$, and a stable solution in time can be observed, resonance effects can be neglected.

To determine the influence of sample preparation irregularities on the recorded data, the thickness of the embedding solder was varied and a Static Analysis was performed.

1. **Modal Analysis → Eigenfrequencies**

   Eigenfrequencies and corresponding mode shapes were searched in the range from 0 to $300 \, \text{kHz}$. In this range 7 Eigenfrequencies were found, see table 3.8.
Table 3.8: Modal Analysis → Tech-A (0 ≤ f ≤ 300 kHz)

<table>
<thead>
<tr>
<th>Mode Nr.</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
</tr>
</thead>
<tbody>
<tr>
<td>$f_{eigen}$ [Hz]</td>
<td>57.764</td>
<td>80.971</td>
<td>152.060</td>
<td>164.200</td>
<td>264.780</td>
<td>279.560</td>
<td>295.480</td>
</tr>
</tbody>
</table>

The lowest calculated eigenfrequency is 57.764 Hz, what is 189 % higher than the testing frequency.
The first mode shape likely to be excited by the sample deflection is shape 4 at 164.200 Hz. This frequency is far away from the testing frequency of 20.000 Hz. A picture of the displacement vector sum of mode shape 4 can be seen in figure 3.9.

Figure 3.9: Displacement vector sum of mode shape 4 at $f_{eigen}=164.200$ Hz.

Further mode shapes would show higher eigenfrequencies than the already observed ones and would not lead to resonance effects.
According to the Modal Analysis resonance effects can be neglected.

2. **Transient Analysis → Stable Solution**
The lower experimentally determined acceleration, $a_1 = 180 \text{ km/s}^2$, was applied according to section 3.6.4 over 4 periods. Number of load-steps was set to 25. A plot of the maximal displacement vector sum and maximal von Mises stress over time can be seen in figure 3.10 (a) and (b).
A stationary solution without increase in amplitude due to resonance was found in a transient analysis over 4 periods of loading.

3. Static Analysis → Preparation Irregularities
The manually prepared samples showed certain irregularities like variation of the solder embedding thickness. In order to evaluate whether preparation irregularities influenced the experimental results, the solder embedding thickness was varied between 0.1 mm and 0.4 mm in steps of 0.1 mm, see table 3.9. A Static Analysis with the peak acceleration of $a_1 = 180 \text{ km/s}^2$ was then performed for every step.

<table>
<thead>
<tr>
<th>Step</th>
<th>$d_{solder\ emb}$ [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.1</td>
</tr>
<tr>
<td>2</td>
<td>0.2</td>
</tr>
<tr>
<td>3</td>
<td>0.3</td>
</tr>
<tr>
<td>4</td>
<td>0.4</td>
</tr>
</tbody>
</table>

A schematic picture of the solder variation is depicted in figure 3.11.

Figure 3.11: Schematic of the solder embedding thickness variation for step 1 and 2.

A plot of the von Mises stresses, $\sigma(d_{solder\ emb})_{step\ 1-4}$, calculated with the respective solder thickness can be seen in figure 3.12.
An influence of the solder embedding thickness on the maximal von Mises stress was not identified.

Von Mises Stress

Exact stress values are important for the lifetime prediction with a Basquin relation. To describe the calculated stress state, the maximal von Mises stress is computed by means of FEM to be further used for the lifetime prediction.

1. Transient Analysis $\rightarrow$ Von Mises Stress

A Transient Analysis over one period of loading was performed for the acceleration $a_1 = 180 \text{ km/s}^2$ and $a_2 = 201 \text{ km/s}^2$. 25 load steps were chosen and modulation happened according to section 3.3.4. One period took $T = 25 \cdot \Delta t = 25 \cdot 2 \mu s$. The maximum value of the von Mises stress at the second maximum of the applied acceleration was further used for the lifetime prediction. See pictures of the simulated results for the acceleration amplitude of $a_1 = 180 \text{ km/s}^2$ in figure 3.13 and 3.14.
Figure 3.13: Displacement vector sum (blue indicates zero displacement) of the transient analysis with for $a_1 = 180 \text{km} \text{s}^{-2}$.

Figure 3.14: Maximal von Mises stress (a) and zoom of a von Mises stress concentration (b) for $a_1 = 180 \text{km} \text{s}^{-2}$.

Stressed areas were clearly those at the interconnection of Al pad and Cu ball bond. The maximal von Mises stress was observed within the wire at the edge of the ball bond, called integration point of ball bond to pad. Other regions experienced low stresses.

It was assumed that the highest stress values within the relevant stress concentration were responsible for the crack initiation and accelerated failure. These stress levels were further utilized for the lifetime modeling. See table 3.15 and figure 3.15 for a summary of the obtained stresses.
Table 3.10: Transient Analysis → Tech-A ($a_1 = 180 \frac{km}{s^2}$ and $a_2 = 201 \frac{km}{s^2}$)

<table>
<thead>
<tr>
<th>Acceleration $\frac{km}{s^2}$</th>
<th>$a_1 = 180$</th>
<th>$2a_2 = 201$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\sigma_{von\ Mises} \ [MPa]$</td>
<td>197</td>
<td>211</td>
</tr>
</tbody>
</table>

Figure 3.15: Transient Analysis → Tech-A ($a_1 = 180 \frac{km}{s^2}$ and $a_2 = 201 \frac{km}{s^2}$)

According to figure 3.13 the loading mode for Tech-A during an active accelerated fatigue test can be described with the to and fro gliding of the chip over the solder parallel to the loading. The deformation parallel to the direction of loading of Tech-A samples in the simulation was recognized as shear fatigue features at the Al pad by means of SEM micrographs of the according fracture surfaces by Lassnig [24].

3.5.5 Results of the Simulation → Tech A

Relevant eigenfrequencies of Tech-A of 57.764 Hz and higher are clearly higher than the testing frequency of 20.000 Hz. No resonance effects were found in a simulation over several applied load periods.

Influences on the experimental results by irregularities of sample preparation were investigated. According to FEM Analysis the solder embedding thickness did not affect the results.

The to and fro gliding of the sample in the direction of the loading of Tech-A samples in the simulation coincides with experimental fracture surface observations by Lassnig [24].

FEM calculation of von Mises stress values according to the performed experiments was conducted. A difference between estimated average shear stresses of 23 to 50 MPa and simulated von Mises stress of 197 MPa and
211 MPa was observed.
The relationship between von Mises stress and shear stress is given in terms of $\sigma_{\text{vonMises}} = \sqrt{3} \tau$. Thus an increase of about a factor of 2 between the FEM calculated and by shear stresses determined curves is expected. It may also be that geometrical factors (stress concentration and loading mode), which were neglected in the estimation of average shear stresses also lead to lower values of stress.

3.5.6 Lifetime Modelling → Tech A

Experimentally obtained fatigue data of the Tech-A sample were plotted in a Wöhler curve. With this data set it was possible to compute a lifetime model for the investigated ball bond interconnection of Cu wire on Al metallization. Lifetime modelling was conducted for estimated stress values and for stresses calculated by FEM.

**Lifetime Modelling - Tech A → Estimated Stresses**

According to section 3.4.1 the experimentally determined Wöhler curve can be fitted with a power law. Application of a least square method was chosen to obtain the fitting function (see figure 3.16).

![Figure 3.16: Power law fit of experimentally recorded fatigue data for Tech-A, 24.](image)

The obtained Basquin relation is (see equation 3.18):

$$\sigma = C \cdot (N_f)^b = 58,535 \cdot N_f^{-0.033}$$  \hspace{1cm} (3.18)
The data showed high scattering which was further discussed by Lassnig in [24]. In order to obtain a lifetime region a lower and upper bound were defined. Without further statistics the four highest and lowest stress values, with respect to the cycles to failure, were chosen. These data pairs were again fitted by a power law. An upper and a lower bound of a life time area were determined with this method (see figure 3.17).

Figure 3.17: Life time region of experimentally recorded fatigue data (blue curve) confined by the upper (green) and lower (purple) life time bound.

Blue indicates all experimentally recorded data. Marks with an arrow indicate runouts. Green indicates points and curve of the upper life time area bound while purple indicates the lower bound of the life time area. The Basquin relation of all points (blue curve) lies in the area confined by upper (green) and lower (purple) bound.

See table 3.11 for a summary of the conducted lifetime modeling.

Table 3.11: Summary of lifetime modelling data with estimated shear stress values for Tech-A

<table>
<thead>
<tr>
<th></th>
<th>( C_{\text{estimated}} ) [MPa]</th>
<th>( b_{\text{estimated}} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Basquin ( \rightarrow ) all points</td>
<td>58,54</td>
<td>-0,033</td>
</tr>
<tr>
<td>Basquin ( \rightarrow ) upper bound</td>
<td>74,49</td>
<td>-0,034</td>
</tr>
<tr>
<td>Basquin ( \rightarrow ) lower bound</td>
<td>35,40</td>
<td>-0,018</td>
</tr>
</tbody>
</table>
Lifetime Modelling - Tech A → FEM calculated Stresses

Experiments performed with Tech-A samples were simulated using a structural 3D finite element simulation. Analysis yielded the von Mises stress, which need to be correlated with the experiment in order to express the experimentally obtained Basquin relation in terms of FEM computed stresses.

As described in section 3.4.2, estimated stress values for the simulation, \( \sigma_{est-model}^{a1/a2} \), were calculated with the finite element model properties. With the experimentally obtained Basquin relation, these stresses were related to the number of cycles to failure, \( N_{f-model}^{a1/a2} \). The obtained couples \( (\sigma_{est-model}^{a1/a2}, N_{f-model}^{a1/a2}) \) expressed fatigue data with estimated stresses from the simulation. \( \sigma_{est-model}^{a1/a2} \) values were exchanged with the respective von Mises stresses, \( \sigma_{FEM}^{a1/a2} \). See table 3.12 for a summary of the described procedure.

\[
\begin{array}{cccccc}
 a_{model} & m_{frac-model} & A_{frac-model} & \sigma_{est-model} & N_{f-model} & \sigma_{FEM} \\
 180 & 3.65 \times 10^{-5} & 1.71 \times 10^6 - 7) & 38.76 & 1.23 \times 10^9 & 197 \\
 201 & 3.65 \times 10^{-5} & 1.71 \times 10^6 - 7) & 43.28 & 6.65 \times 10^3 & 211 \\
\end{array}
\]

The two couples \( (\sigma_{FEM}^{a1/a2}, N_{f-model}^{a1/a2}) \) for 180 km/s^2 and 201 km/s^2 yield two power law relations with two unknown parameters, \( C_{FEM} \) and \( b_{FEM} \) (see equation 3.19 to 3.21):

\[
\sigma_{FEM}^{a1/a2} = C_{FEM} \cdot (N_{f-model}^{a1/a2})^{b_{FEM}} \quad (3.19)
\]
\[
197 = C_{FEM} \cdot (1.23 \times 10^9)^{b_{FEM}} \quad (3.20)
\]
\[
211 = C_{FEM} \cdot (6.65 \times 10^3)^{b_{FEM}} \quad (3.21)
\]

Solving the system of equations for \( C_{FEM} \) and \( b_{FEM} \), yielded the parameters for the fatigue model obtained with FEM calculated stresses. The same procedure was done for the upper and lower bound parameters from table 3.11. See table 3.13 for a summary of the conducted lifetime modelling with FEM calculated stress values and figure 3.18 for a graphical illustration of the conducted life time modelling.
Table 3.13: Summary of lifetime modelling data with FEM calculated stress values for Tech-A

<table>
<thead>
<tr>
<th>Basquin → all points</th>
<th>$C_{FEM}$ [MPa]</th>
<th>$b_{FEM}$ [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>259.53</td>
<td>-0.024</td>
<td></td>
</tr>
<tr>
<td>Basquin → upper limit</td>
<td>295.79</td>
<td>-0.021</td>
</tr>
<tr>
<td>Basquin → lower limit</td>
<td>186.21</td>
<td>-0.011</td>
</tr>
</tbody>
</table>

Figure 3.18: Lifetime curves for Tech-A samples based on calculated von Mises stresses.

Blue is correlated with all experimentally recorded data. Green indicates points and curve of the upper life time area bound while purple indicates a lower bound of the life time area.

Comparison of experimentally determined lifetime models (figure 3.17) and lifetime models based on FEM analysis (figure 3.18) shows higher values of stress for the latter. As already explained before in section 3.5.5 this difference is related to the definition of the von Mises stress (see sections 2.1.3 and 3.3.6 and) and the neglecting of geometrical effects of the average shear stress approach. Another factor is the difference between the failure criteria defined for each approach. Number of cycles to failure, $N_f$, is related to the complete fracture and separation of the interface for the lifetime curves in terms of average shear stresses. In the FEM supported approach crack initiation was defined as failure criteria with the highest value of von
Mises stress considered as the relevant stress.

3.6 Experimental - Tech B

Samples named as Tech-B are devices with one copper nailhead on an aluminium pad. Within this chapter descriptions of the sample, sample preparation method, the experimental procedure and the experimental results are given. Furthermore Finite Element Simulation of the fatigue experiment and Lifetime Modeling are presented. An ultrasonic fatigue testing setup as described in section 3.1 was used to subject metallic micro-joints to fatigue loading. Testing was done according to section 3.2.

3.6.1 Sample and Sample Preparation → Tech B

Tech-B samples were provided in an unmolded condition similar to Tech-A samples. The thickness of the Copper wire was 50 µm and the thickness of Aluminium pad was 6 µm which resulted in a ball bond with a diameter of 140 µm and a height of 40 µm. See SEM micrographs of the Tech-B sample in figure 3.19.

![SEM micrographs of the Tech-B sample.](image)

Figure 3.19: Top view (a) and cross section (b) of the Tech-B sample.
Since the mass of the Cu ball bond was very small, in order to obtain sufficient high stresses required for fatigue damage in the bond interface it was necessary to increase the mass using sample preparation methods. This was achieved by application of a special soldering technique as described in the following.

Preparation

The upside down geometry of the sample preparation used by Khatibi et al [18] was useful for thick wires (400 μm) (see figure 3.5). An upside down preparation including an additional solder layer which was used by Lassnig [24] made it possible to expose several miniaturized Cu-Al ball bonded interconnects to fatigue loading. For both techniques coupling between the sample holder and the micro-component was provided by the bond interconnection. Loading occurred due to the inertia of the micro-component mass.

In case of Tech-B samples which consisted of one ball bond interconnection with a wire bond of 50 μm, loading by an upside down geometry would not work due to the poor stability of only one thin wire bond.

An alternative preparation would be to place the micro-component in an upright position onto the sample holder. In an upright set-up the interconnection would be loaded due to the inertia of the copper ball bond mass during the accelerated fatigue testing. Nevertheless due to the small mass of the ball bond the cyclic stress amplitude induced in the interconnection would be too low. Thus it was necessary to increase the mass of the ball bond by using a special preparation method, to increase the inertia and obtain sufficient high stresses to induce fatigue fracture in the interconnect during the accelerated testing.

In the following details of the sample preparation technique are given and a schematic of the procedure is illustrated in figure 3.20.

The copper wire bonds of the unmolded Tech-B samples were cut from the leadframe side and bend to a vertical position at the ball bond side (figure 3.20 (a)). A thin piece of Aluminium foil with dimensions of 5 μm x 5 mm x 5 mm was cut in a suitable shape in order to cover the surrounding surface of the nailhead and let the ball bond and the copper wire free (figure 3.20 (b)). Using special templates, this assembly was fixed by clamping the foil to the chip. The correct position of the foil and its cutting were checked with a stereo microscope. Prior to the soldering all parts were degreased and cleaned by using aceton. The copper nailhead and the wire were covered with a soldering flux by using a fine brush.

Commercial solder wire of Sn63 Pb37 was used in order to adhere an
additional mass to the copper nailhead. A thin solder ring was prepared out of the solder wire by cutting a thin slice of about 200 \( \mu m \) thick from the wire in which a small hole was made. The obtained ring was placed around the nailhead on top of the Al foil. The Al foil provided a defined distance between the Al metallization pad and the solder material (figure 3.20 (c)). Soldering of the ring to the copper nailhead was performed by using a programmable reflow oven. Adjustable parameters were the temperature (in \(^\circ C\)) and duration (in s) of a preheating phase, \( T_1 \) and \( t_1 \), the temperature and duration of the actual heating phase, \( T_2 \) and \( t_2 \), and the duration of the cooling period. Satisfying soldering results were achieved with a setting of \( T_1 = 180^\circ C, t_1 = 150 \text{ s} \) and \( T_2 = 210^\circ C, t_2 = 170 \text{ s} \). After the soldering process the sample was left in order to cool down slowly. Active cooling by the reflow oven was not performed. According to producer information this heat exposure should not effect the bond connection properties.

The solder ring formed a ball after it was heated over the liquidus (figure 3.20 (d)). Due to the good reactivity of copper and solder the nailhead was strongly bonded to the solder ball, while there was no adhesion between the solder ball to the aluminium foil and the underlying metallization due to lack of wetting. Hence it was possible to remove the aluminium foil after the assembly cooled down with a pair of tweezers. See a sketch of the preparation process in figure 3.20.

![Figure 3.20](image)

**Figure 3.20:** Sketch of the preparation for Tech-B samples.

The solder ball was soldered to the copper nailhead and the wire. There was no connection between Aluminium and solder as shown in the Scanning Electron Microscope (SEM\(^1\)) micrograph of a prepared sample in figure 3.21.

---

\(^1\)SEM: Supra 55 VP, Zeiss
Figure 3.21: SEM micrograph of a cross section for a prepared Tech-B sample.

Typical properties of the solder ball can be seen in table 3.14.

<table>
<thead>
<tr>
<th>$r_{solder ball}$ [mm]</th>
<th>$m_{solder ball}$ [mg]</th>
<th>Material$_{solder ball}$ [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.36</td>
<td>1.6</td>
<td>Sn63 Pb37</td>
</tr>
</tbody>
</table>

With this procedure and the sample in an upright position, it was possible to increase the loading mass to a level which allowed accelerated testing of the bond micro-interconnect. Finally the lead frame of the soldered Tech-B sample was glued to the free end of the sample holder. See a sketch of the final preparation in figure 3.22 (a). The bond interconnection got loaded due to the inertia of the ball bond and the solder ball.

3.6.2 Experimental Procedure → Tech B

For the accelerated fatigue testing the prepared Tech-B sample was attached to a sample holder and mounted to an ultrasonic fatigue system$^2$ Thus

$^2$US resonance system: Ultrasonic transducer, SE 50/30 6-20 4TR2, Telsonic

$^3$Ultrasonic Generatro: DG-2000, Telsonic
during the loading the sample holder and the attached micro-component were excited to longitudinal vibrations. Due to its inertia the solder ball experienced an acceleration relative to the moving sample holder. Hence cyclic shear stress was induced in the interconnect area. Coupling between the solder ball and the chip which was glued to the sample holder was provided by one ball bond interconnection. The desired fracture, called lift off, occurred in the interface of copper ball bond to aluminium pad metallization. A sketch of the final preparation and the desired lift off of can be seen in figure 3.22 (a) and (b). The interface between ball bond and the chip acted as load-bearing area.

Figure 3.22: Sketch of the cross section of the solder ball sample preparation (a) and the desired lift off mode (b).

For the estimation of forces and average shear stresses the lift off mass, the acceleration acting on the solder ball and the fracture surface must be known. The testing processes and the obtained values are described in the following.

**Experimental Setup Calibration**

To determine the acceleration acting on the solder ball, $a_{solderball}$, a LDV\(^4\) was used. In order to correlate velocities measured by the LDV with strain values of the sample holder measured by a strain gage\(^5\), a calibration procedure like described in section 3.2.2 was performed. This was done because strain values were easier to obtain during an active experiment than permanent measurement by a LDV.

Velocity was measured with a LDV parallel to the direction of motion at the center of the solder ball. A sketch of the described procedure can be seen in figure 3.23.

\(^4\)LDV: OFV 534 & OFV 2500-2, Polytec
\(^5\)Strain Gage: Type 3/120LY19, $k=2.04$, HBM
This procedure yielded the peak velocity and the frequency, which were used to calculate the peak acceleration acting on the solder ball according to equation 3.8. Simultaneously to the velocity measurements with the LDV, strain was measured with a strain gage at the middle of the sample holder. Plots of measured velocity versus strain and calculated acceleration versus strain are depicted in figure 3.24 (a) and (b).

The depicted range of velocity and acceleration versus strain in figure 3.24 includes the experimentally applied settings in the accelerated testing of the micro-interconnection. This procedure made it possible to perform strain measurement during the experiment and correlation of these values with accelerations acting on the solder ball.
Lift off Mass, Fracture Surface and Average Shear Stresses

The lift off mass, \( m_{\text{liftoff}} \), was the mass of the solder ball and the nailhead. In order to obtain exact stresses, the lift of mass was measured after fracture with a precision scale 6.

The fracture surface area, \( A_{\text{frac}} \), consisted of the interconnection area of the nailhead to the metallization pad and was determined after fracture with a microscope 7. For later calculations a mean value was formed to represent the area of the fracture surface.

The average value of shear stress in the interface of the bond interconnection was approximated with the values of \( m_{\text{liftoff}} \), \( A^\text{mean}_{\text{frac}} \), \( a_{\text{solderball}} \) and equations 3.6 and 3.7 (see equation 3.22).

\[
\tau_{\text{estimated-B}} = \frac{F}{A^\text{mean}_{\text{frac}}} = \frac{m_{\text{liftoff}} \cdot a_{\text{solderball}}}{A^\text{mean}_{\text{frac}}} \tag{3.22}
\]

Cycles to failure

In order to determine the number of cycles to failure of an accelerated fatigue test, video recording and time measurement were performed. Recording of the video and time measurement were started simultaneously with the fatigue experiment. After the lift off occurred the time of loading, \( t_{\text{loading}} \), was determined and with the testing frequency, \( f_{\text{test}} \), the number of cycles was calculated, \( N_f = t_{\text{loading}} \cdot f_{\text{test}} \).

With the described procedure it was possible to determine fatigue life curves of a single metallic ball bond interconnection of copper wires to aluminium pads for Tech-B.

3.6.3 Experimental Results → Tech B

Using the above mentioned methods fatigue life curves of a single Cu-Al ball bond connection were obtained.

Sample Specifications and Test Conditions

Typical sample specifications and test conditions like mass, fracture surface and acceleration are listed below and a summary can be seen in table 3.15.

6Scale: AT 250, Mettler  
7Microscope: Axioplan, Zeiss
**Lift-Off-Mass**  Lift-off masses were measured after fracture with a precision scale. The minimum and maximum values for the lift-off mass were $m_{\text{liftoff}}^{\text{min}} = 1.24$ to $m_{\text{liftoff}}^{\text{max}} = 2.07$ mg.

**Fracture Surface**  The fracture surface area, $A_{\text{frac}}$, consisted of the interconnection area of the nailhead to the metallization pad and was investigated with a SEM. A mean value of $A_{\text{frac}}^{\text{mean}} = 1.14 \cdot 10^{-2} \pm 0.06 \cdot 10^{-2}$ mm$^2$ was measured.

**Acceleration**  Acceleration was calculated from velocity measurements by a LDV. Fatigue data was recorded in a range of cycles to failure from $N_f = 10^5$ to $N_f = 10^9$. Applied accelerations ranged from $a_{\text{solderball}}^{\text{min}} = 52 \text{ km/s}^2$ to $a_{\text{solderball}}^{\text{max}} = 107 \text{ km/s}^2$.

Table 3.15: Sample specifications and test conditions.

<table>
<thead>
<tr>
<th>$m_{\text{liftoff}}^{\text{min}}$ [mg]</th>
<th>$m_{\text{liftoff}}^{\text{max}}$ [mg]</th>
<th>$A_{\text{frac}}^{\text{mean}}$ [mm$^2$]</th>
<th>$\Delta A_{\text{frac}}^{\text{mean}}$ [mm$^2$]</th>
<th>$a_{\text{solderball}}^{\text{min}}$ [km/s$^2$]</th>
<th>$a_{\text{solderball}}^{\text{max}}$ [km/s$^2$]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.24</td>
<td>2.07</td>
<td>1.14 $\cdot 10^{-2}$</td>
<td>0.06 $\cdot 10^{-2}$</td>
<td>52.89</td>
<td>107.34</td>
</tr>
</tbody>
</table>

Inserting the values from table 3.15 into equation 3.22 yielded estimated average shear stress values in a range of (see equation 3.23)

$$5.37 \text{ MPa} \leq \tau_{\text{estimated}} \leq 20.51 \text{ MPa}$$  \hspace{1cm} (3.23)

(standard errors were not taken into consideration).

**Shear Test**  Shear testing (section 2.3.2) was performed for Tech-B samples by Infineon Technologies AG. Mean shear strength was $150 \pm 10$ MPa for the tested single Cu-Al interconnect.

**Fatigue Data - Wöhler Curve**  In order to obtain Wöhler curves with estimated shear stresses versus cycles to failure for Tech-B, individual lift-off masses and mean values for the fracture surface were used and computed with equation 3.22. The measured data of successfully tested samples can be seen in table 3.16 and a graphical illustration of the fatigue life data for Tech-B is presented in figure 3.25.
The number of cycles to failure ranges between $N_f = 10^5$ to $N_f = 10^9$. White marks framed in red indicate samples without rupture during the test, called runouts.

Estimated average shear stresses of the obtained Wöhler curve for Tech-B are between 13 MPa and 6 MPa at cycles to failure of $10^5$ to $10^9$. The recorded data showed moderate scattering and it was not necessary to define a lifetime range like done for Tech-A. Shear tests performed with Tech-B samples yielded shear strength of $150 \pm 10$ MPa. Moderate scattering of experimentally obtained fatigue data may be related to the low scattering of shear strength for Tech-A ($150 \pm 10$ MPa) in comparison with Tech B ($175 \pm 40$ MPa).

Like for Tech-A samples the estimated average shear stresses approach do not include geometry effects or complex stress states and may lead to underestimated stresses [18]. Therefore FEM analysis of the shear fatigue experiment using a precise geometrical model of the sample including plasticity was performed.
Table 3.16: Measured data of successfully tested Tech-B samples.

| sample | comment | \( a_{\text{solderball}} \) \([m]\) | \( m_{\text{lift off}} \) \([kg]\) | \( t_{\text{loading}} \) \([s]\) | \( t_{\text{testing}} \) \([-]\) | \( N_f \) \([-]\) | \( A_{\text{frac}} \) \([m^2]\) | \( \sigma_{\text{estimated}} \) \([MPa]\) |
|--------|---------|-----------------|-----------------|-----------------|-----------------|-----------------|-----------------|
| 1.3    | lift off | 52.891           | 1,52E-06        | 1601            | 20012           | 3,20E+07        | 1,10E-08        | 7.03            |
| 3.3    | lift off | 66.963           | 1,61E-06        | 534             | 20015           | 1,07E+07        | 1,27E-08        | 9.43            |
| 9.3    | lift off | 64.436           | 2,07E-06        | 321             | 20015           | 6,42E+06        | 1,20E-08        | 11.66           |
| 1.4    | runout   | 52.891           | 1,34E-06        | 5400            | 20012           | 1,00E+09        | 1,13E-08        | 6.20            |
| 1.4    | lift off | 80.230           | 1,34E-06        | 460             | 20015           | 9,21E+06        | 1,13E-08        | 9.40            |
| 2.4    | lift off | 66.963           | 1,70E-06        | 751             | 20015           | 1,50E+07        | 1,09E-08        | 9.95            |
| 4.4    | lift off | 71.701           | 1,24E-06        | 5400            | 20015           | 1,08E+08        | 1,05E-08        | 7.77            |
| 11.4   | runout   | 58.745           | 1,53E-06        | 65280           | 20014           | 1,00E+09        | 1,14E-08        | 7.86            |
| 12.4   | lift off | 69.490           | 1,77E-06        | 6               | 20015           | 1,20E+05        | 1,28E-08        | 10.75           |
| 3.5    | lift off | 69.490           | 1,75E-06        | 5               | 20015           | 1,00E+05        | 1,05E-08        | 10.63           |
| 12.5   | runout   | 64.436           | 1,62E-06        | 59741           | 20015           | 1,00E+09        | 1,14E-08        | 9.13            |
| 15.5   | lift off | 73.912           | 1,97E-06        | 82              | 20015           | 1,64E+06        | 1,11E-08        | 12.73           |
Fracture Surface

Foot prints of copper ball bonds on the aluminium pad metallization were investigated by means of a SEM\(^8\) after fatigue fracture. This analysis was supposed to reveal the dominant fracture mode of the tested ball bond interconnects. Fracture surface interpretation was done according to [14], [31] and [33]. Bright areas in the following micrographs display electrical charged regions which occurred due to a rest of flux material of the soldering.

Fracture Surface - Tech-B-a  A micrograph of the entire foot print (chip-side) for the Tech-B-a sample can be seen in figure 3.26.

![Fracture Surface - Tech-B-a](image)

Figure 3.26: Entire fracture surface Tech-B-a, chip-side.

Fatigue fracture characteristics can be observed at the periphery of the bond while characteristics of a ductile fracture are visible in the center of the footprint. On the left and right edge of the ball bond horizontal lines can be observed. These tracks show the direction of the bonding process. Fracture mainly occurred in the Al metallization. The upper and lower fracture surface periphery of the footprint Tech-B-a is depicted in more detail in figure 3.27 and figure 3.28.

\(^8\)SEM: Supra SS VP, Zeiss
Parallel lines, called striations, around the center of the fracture can be observed in figure 3.27 and 3.28. This can be seen as evidence for a fatigue fracture at the ball bond edge. The center of the fracture surface can be seen in figure 3.29.
A dimple like structure can be observed in figure 3.29. This structure is typical for a ductile fracture.

In the following several SEM micrographs of fracture surfaces are shown. Fatigue fracture at the bond periphery indicated by striations and a ductile fracture indicated by a dimple structure can be seen in all of the regularly fractured ball bond connections as explained for sample Tech-B-a.

**Fracture Surface - Tech-B-b**

A micrograph of the entire foot print for the Tech-B-b sample can be seen in figure 3.30.
The lower periphery of the fracture surface of the footprint Tech-B-b is depicted in figure 3.31.

The center of the fracture surface can be seen in figure 3.32.
Figure 3.32: Fracture surface in the center of the footprint Tech-B-b.

**Fracture Surface - Tech-B-c** A micrograph of the entire footprint for the Tech-B-c sample can be seen in figure 3.33.

Figure 3.33: Entire fracture surface Tech-B-c.

The lower periphery of the fracture surface of the footprint Tech-B-c is depicted in figure 3.34.
Figure 3.34: Fracture surface at the lower periphery of the footprint Tech-B-c.

The center of the fracture surface can be seen in figure 3.35.

Figure 3.35: Fracture surface in the center of the footprint Tech-B-c.

**Fracture Surface - Tech-B-d**  Like for the mechanical shear test for ball bonds different failure modes were investigated for the accelerated fatigue testing. Failure at sample Tech-B-d occurred due to cratering and lift off of
the entire metallization under the ball bond. A micrograph of the entire foot
print for the sample Tech-B-d can be seen in figure 3.36. This failure mode
occurred very seldom and did not affect the experimental output.

Figure 3.36: Entire fracture surface Tech-B-d.

Fracture of single ball bonds in Tech-B samples mainly occurred in the
Aluminium metallization. Significant formation of IMC was not observed.
Formation of dimples in the center of the footprint indicates that final
fracture occurred due to a ductile fracture between ball bond and the pad
after crack propagation decreased the interconnection area.

3.6.4 Simulation → Tech B

FEM analysis was performed in order to evaluate the validity of the geometry
of Tech-B samples and calculate exact stress distributions of conducted
experiments. The 3-dimensional simulations were carried out with the FEM
code ANSYS. In the following the performed simulations are presented.

Volumes and Geometry

Like mentioned in section 3.3.2 the sample dimensions were determined with
a light microscope and assembled to a three dimensional reconstructed model
of Tech-B samples.

For the FEM analysis, the geometry was simplified. Preparation of Tech-B,
with the sample in an upright position on the sample holder and the solder ball together with the nailhead being the liftoff mass, allowed to neglect geometrical details for the chip. Layer thicknesses was modelled according to microscope measurements but lateral dimensions were reduced to a square. These simplifications were performed in order to save computational time without any effects on the accuracy of the simulation.

Main volumes and their material properties are listed in section 3.3.2. Due to the preparation an additional Volume, the Solder Ball, was introduced. A brief description of the used volumes is given.

1. **Leadframe-(Cu)** A rectangular shaped volume, 2000/2000/265 µm (width, length, height), with material properties according to table 3.1.

2. **Chip Adhesive-(Sn63 Pb37)** A thin solder layer, 2000/2000/20 µm, with material properties according to table 3.2.

3. **Substrate-(Si)** A rectangular shaped volume, 2000/2000/170 µm, with material properties according to table 3.3. Due to the adjusted mesh refinement the substrate was subdivided at the site of the nailhead.

4. **Metallization-(Al)** A 6 µm layer, 2000/2000 µm (width, length), coated onto the substrate with material properties according to table 3.4 was modelled. In order to provide a nice mesh transition from the large rectangular shaped metallization to the small round shape of the nailhead a transition by subdividing the metallization was executed. Small squares with cylinders were placed under every nailhead.

5. **Nailhead-(Cu)** One nailhead was placed with an outer diameter of 140 µm and a height of 40 µm in the center of the modelled metallization with material properties like shown in table 3.5.

6. **Solder Ball** According to the sample preparation a solder ball, made of Sn63 Pb37, with a diameter of 0,7 mm was attached to the nailhead. A bi-linear material model with kinematic hardening was chosen (see table 3.17).

<table>
<thead>
<tr>
<th>ρ [kg/m³]</th>
<th>E [MPa]</th>
<th>ν</th>
<th>σyield [MPa]</th>
<th>T [MPa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>8825</td>
<td>31.4</td>
<td>0.33</td>
<td>72.0</td>
<td>10.0</td>
</tr>
</tbody>
</table>
Model and Mesh

The geometry of the model is depicted in figure 3.37 (a) and (b). Because of the symmetry of the assembly, a mirror symmetry plane was inserted, indicated in red. The sample was fixed spatially by means of boundary conditions according to the idea in section 3.3.4, indicated in blue.

Figure 3.37: Picture of the used geometry (a), the symmetry area (red) and fixed boundary condition (blue) (b) for finite element modeling of Tech-B.

Discretization was achieved with an adjusted mesh method. Volumes were subdivided according to the needs of the mesh refinement. Special attention was payed to the contact properties of adjacent volumes. The contact area between the solder ball and the metallization (and its sub-volumes) was realized as a frictionless gliding pair of planes. Penetration of the volumes was prevented whereas opening of a gap between the two pairs was enabled. All other contacts were glued (perfectly rigid contact pair). See a picture of the mentioned ideas in figure 3.38 (a) and (b).

Figure 3.38: Picture of the complete meshed model (a) and a zoom of the adjusted mesh (b) for Tech-B.
Loads

In the simulation the loads due to inertia forces were modelled as spatial constant acceleration field while the sample was fixed using boundary conditions. To evaluate the coefficients in the Basquin relation it was necessary to evaluate the stresses at two different accelerations. The experimentally measured acceleration amplitudes ranged from $50 \text{ km/s}^2$ to $110 \text{ km/s}^2$. For simulation purposes two accelerations were chosen at $a_1 = 50 \text{ km/s}^2$ and $a_2 = 60 \text{ km/s}^2$. These accelerations, $a_1$ and $a_2$, were used as maximum values for Static and Transient Analysis. For the transient load a sinusoidal curve was chosen with 25 load-steps, $n=25$. Equations 3.11 and 3.12 with $n=25$ and $f=20$ kHz yield numerical values for the simulated loads, see equations 3.24 to 3.25.

\[
\Delta t = \frac{1}{20.000 \cdot 25} = 2 \mu s \tag{3.24}
\]

\[
a_{1-i} = 50 \cdot \sin(i \cdot \frac{2\pi}{25}) \quad \& \quad a_{2-i} = 60 \cdot \sin(i \cdot \frac{2\pi}{25}) \tag{3.25}
\]

Geometry Check

In order to be able to neglect resonance effects, a Modal Analysis and a Transient Analysis were performed. If eigenfrequencies, $f_{eigen}$, are much higher than the testing frequency of 20 kHz and a stable solution is observed in time, resonance effects can be neglected.

1. Modal Analysis $\rightarrow$ Eigenfrequencies

Eigenfrequencies and corresponding modeshapes were searched in the range of 0 to 1 MHz. Due to the fact that a Modal Analysis in ANSYS always means a linear analysis, limitations for the modelling of contact properties occurred. Contact pairs with the ability of opening a gap between each other while penetration is forbidden form a nonlinear behaviour and were not considered in Modal Analyses. In order to evaluate the eigenfrequencies of the solder ball geometry, three different modal simulations with different contact properties between solder ball and metallization were performed and interpreted.

(a) Penetrating Contact properties between the volumes of the solder ball and the metallization were totally omitted. This lead to the effect that the volumes (metallization and solder ball) were able to penetrate each other when being deflected. No stabilizing effect was given by the metallization with respect to the solder ball.
Eigenfrequencies were supposed to be lowest possible due to the minimal coupling between the volumes. Three eigenfrequencies with corresponding modeshapes were found between 0 and 1 MHz. A summary of the obtained results can be seen in table 3.18.

Table 3.18: *Penetrating* Modal Analysis → Tech-B ($0 \leq f \leq 1$ MHz)

<table>
<thead>
<tr>
<th>Mode Nr.</th>
<th>$f_{eigen}$ [Hz]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>27.433</td>
</tr>
<tr>
<td>2</td>
<td>256.970</td>
</tr>
<tr>
<td>3</td>
<td>404.660</td>
</tr>
</tbody>
</table>

Figure 3.39: Displacement vector sum (a) and vertical displacement (b) of the penetrating Modal Analysis for Tech-B.

The lowest calculated eigenfrequency of 27.433 Hz was 37% higher than the testing frequency of 20.000 Hz. The first mode shape likely to be excited by the sample displacement during the accelerated test was shape 1 at 27.433 Hz. A picture of the displacement vector sum and vertical displacement of mode shape 1 can be seen in figure 3.39 (a) and (b). A gap between solder ball and metallization can be seen on the one side, while penetration of the volumes can be investigated on the other side of the nailhead. The deflection can be described as a tilting of the solder ball. This can be interpreted as tilting of the solder ball relative to the movement of the sample holder during the accelerated fatigue experiment.

(b) **Gliding** The contact pair between volumes of the solder ball and the metallization was performed as purely gliding. Opening of a gap was prevented while gliding of one volume over the other was allowed. This behaviour is supported by a Modal Analysis. The solder ball became deflected and was stabilized by
the metallization.
Due to the increased coupling between the volumes of the solder ball and the metallization eigenfrequencies were expected to be higher than for the *penetrating* Modal Analysis. Three eigenfrequencies with corresponding modeshapes were found between 0 and 1 MHz. A summary of the obtained results can be seen in table 3.19.

<table>
<thead>
<tr>
<th>Mode Nr.</th>
<th>1</th>
<th>2</th>
<th>3</th>
</tr>
</thead>
<tbody>
<tr>
<td>$f_{eigen}$ [Hz]</td>
<td>63.237</td>
<td>348.200</td>
<td>406.90</td>
</tr>
</tbody>
</table>

Figure 3.40: Displacement vector sum (a) and vertical displacement (b) of the gliding Modal Analysis for Tech-B.

The lowest calculated eigenfrequency of 63.237 Hz was 216 % higher than the testing frequency of 20.000 Hz. The first mode shape likely to be excited by the sample displacement during the accelerated test was shape 1 at 63.237 Hz. A picture of the displacement vector sum and the vertical displacement of mode shape 1 can be seen in figure 3.40 (a) and (b). No gap between solder ball and metallization can be investigated.

Like observed in the *penetrating* analysis the deflection can be described as a tilting of the solder ball. This deformation is equivalent to a tilting effect parallel to the displacement in the fatigue experiment.

(c) **Fixed** By gluing the volumes of solder ball and metallization the coupling between these volumes was increased to its maximum. Neither gliding nor opening of a gap was allowed. Stronger
stabilizing effects than observed for the gliding Modal Analysis were achieved in the simulation. Eigenfrequencies were supposed to be higher than for the penetrating and the gliding Modal Analysis. Three eigenfrequencies with corresponding modeshapes were found between 0 and 1 MHz. A summary of the obtained results can be seen in table 3.20.

Table 3.20: Fixed Modal Analysis → Tech-B (0 ≤ f ≤ 1 MHz)

<table>
<thead>
<tr>
<th>Mode Nr.</th>
<th>1</th>
<th>2</th>
<th>3</th>
</tr>
</thead>
<tbody>
<tr>
<td>$f_{eigen}$ [Hz]</td>
<td>80.288</td>
<td>383.250</td>
<td>546.490</td>
</tr>
</tbody>
</table>

![Figure 3.41: Displacement vector sum of the fixed Modal Analysis for Tech-B.](image)

The lowest calculated eigenfrequency of 80.288 Hz was 301 % higher than the testing frequency of 20.000 Hz. The first mode shape likely to be excited by the sample displacement during the accelerated test was shape 1 at 80.288 Hz. A picture of the displacement vector sum of mode shape 1 can be seen in figure 3.41. No gap and no gliding between solder ball and metallization can be investigated.

Further mode shapes would correspond to higher eigenfrequencies than the already observed ones and would not lead to resonance effects. Three different Modal Analyses were performed with different coupling between solder ball and metallization. Eigenfrequencies were found at 27.433 (penetrating), 63.237 (gliding) and 80.288 Hz (fixed). All frequencies show a distinct difference to the experimentally applied
loading frequency of 20.000 Hz. With respect to the contact properties in the respective models, the *gliding* Modal Analysis seemed to match the desired sample properties best. It is argued that the eigenfrequency for a model with a gliding contact pair with the ability of opening a gap should be between the found frequencies for the *penetrating*, 27.433 Hz, and the *gliding* Analysis, 63.237 Hz. A difference between eigenfrequency and testing frequency is proofed and according to the Modal Analysis resonance effects can be neglected.

2. **Transient Analysis → Stable Solution**

The lower experimentally determined acceleration, \(a_1 = 50 \frac{km}{s^2}\), was applied over 4 periods. Number of load-steps was set to 25. A plot of maximal displacement vector sum and maximal von Mises stress over time can be seen in figure 3.42 (a) and (b).

![Figure 3.42: Maximal displacement vector sum (a) and maximal von Mises stress (b) for \(a_1 = 50 \frac{km}{s^2}\).](image)

A stationary solution without increase in amplitude due to resonance was found in a transient analysis over 4 periods of loading.

**Von Mises Stress**

Exact stress values are important for the lifetime prediction with a Basquin relation. To describe the FEM calculated stress state, the maximal von Mises stress is computed to be further used for the lifetime prediction.

1. **Transient Analysis → Von Mises Stress**

A Transient analysis over one period of loading was performed for the acceleration \(a_1 = 50 \frac{km}{s^2}\) and \(a_2 = 60 \frac{km}{s^2}\). 25 load steps were chosen and modulation happened according to section 3.3.4. One period took \(T = 25 \cdot \Delta t = 25 \cdot 2 \mu s\). The maximum value of the von Mises stress at
the second maximum of the applied acceleration was further used for the lifetime prediction. See pictures of the graphically displayed results of the simulation for $a_1 = 50 \frac{km}{s^2}$ in figure 3.13 and 3.44.

Figure 3.43: Displacement vector sum (blue indicates zero displacement) (a) and plot of the von Mises stress (b) of the transient analysis with for $a_1 = 50 \frac{km}{s^2}$.

Figure 3.44: Plot of the von Mises stress for the total Tech-B sample (a) and von Mises stress at the metallization (b) for $a_1 = 50 \frac{km}{s^2}$.

Intensively stressed areas are clearly those at the interconnection of Al pad and Cu ball bond. The maximal von Mises stress can be observed at the edge of the ball bond, called integration point of ball bond to pad. Other regions experienced low stresses. It was assumed that the highest stress values within the relevant stress concentration were responsible for the crack initiation and accelerated failure. These stress levels were further utilized for the lifetime modelling. See table 3.45 and figure 3.45 for a summary of the obtained stresses.
Table 3.21: Transient Analysis → Tech-B ($a_1 = 50 \frac{km}{s^2}$ and $a_2 = 60 \frac{km}{s^2}$)

<table>
<thead>
<tr>
<th>Acceleration $\frac{km}{s^2}$</th>
<th>$a_1 = 50$</th>
<th>$2a_2 = 60$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\sigma_{von\ Mises}$ [MPa]</td>
<td>194</td>
<td>227</td>
</tr>
</tbody>
</table>

Figure 3.45: Transient Analysis → Tech-B ($a_1 = 50 \frac{km}{s^2}$ and $a_2 = 60 \frac{km}{s^2}$)

According to figures 3.43 (a) and (b) the loading mode for Tech-B during an active accelerated fatigue test can be described as tilting of the solder ball parallel to the loading. Stresses in the ball bond interconnect due to the tilting occur in areas of alternating tension and compression at the periphery of the ball bond.

This deformation does not show a loading direction in the ball bond interface like observed for Tech-A samples. According to SEM micrographs in section 3.6.3 fatigue features at the periphery of the center of the fracture surface were observed which coincides with the observed tilting loading mode in this section.

These micrographs may also confirm the assumption of crack initiation due to maximal von Mises stresses at the integration point of ball bond to pad at the bond periphery. Characteristics of a ductile fracture can be observed in the center as indication of the final rupture of the ball bond.

3.6.5 Results of the Simulation → Tech B

Relevant eigenfrequencies of Tech-B of 27.433 Hz and higher are clearly higher than the testing frequency of 20.000 Hz. No resonance effects were found in a simulation over several applied load periods.

Deformation due to tilting of the solder ball in the FEM simulation
may be confirmed with experimental fracture surface observations by means of SEM for Tech-B samples.

FEM calculations of von Mises stress values corresponding to the conducted experiments were performed. Comparing the stresses obtained by the estimated approach (6 MPa to 13 MPa) with those calculated by means of FEM (197 MPa and 211 MPa) reveals a difference between the respective stress levels. An explanation for higher von Mises stresses can be given similar to Tech-A. Neglecting the complex loading mode, state of stress, different failure criteria and the definition of von Mises contribute to the difference between the two approaches.

3.6.6 Lifetime Modelling → Tech B

Experimentally obtained fatigue data of Tech-B samples is plotted in a S-N diagram. With this data set it was possible to compute a lifetime model for the investigated single ball bond interconnection of Cu wire on Al metallization. Lifetime modelling was conducted for estimated shear stress values and for stresses calculated by FEM.

Lifetime Modelling - Tech B → Estimated Stresses

According to section 3.4.1 the experimentally determined fatigue data (see figure 3.25) can be fitted with a power law. Application of a least square method was chosen to obtain the fitting function (see figure 3.46).
Figure 3.46: Basquin power law relation fit of experimentally recorded fatigue curve data for Tech-B.

The obtained Basquin relation is (see equation 3.26):

$$\sigma = C \cdot N_f^b = 20,299 \cdot N_f^{-0.048}$$

(3.26)

The data showed moderate scattering and no further action was necessary in order to obtain a lifetime area like done for Tech-A. See table 3.22 for a summary of the conducted lifetime modelling.

Table 3.22: Summary of lifetime modelling data with estimated shear stress values for Tech-B

<table>
<thead>
<tr>
<th>Basquin → all points</th>
<th>$C_{estimated}$ [MPa]</th>
<th>$b_{estimated}$ [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>20,299</td>
<td>-0.048</td>
</tr>
</tbody>
</table>

**Lifetime Modelling - Tech B → FEM calculated Stresses**

Experiments performed with Tech-B samples were simulated using a structural 3D finite element simulation. Analysis yielded the von Mises stress, which need to be correlated with the experiment in order to express the experimentally obtained Basquin relation in terms of FEM computed stresses.
Estimated average stress values, $\sigma_{\text{est-model}}^{a_1/a_2}$, were calculated with the finite element model specifications according to equations 3.6 and 3.7. With the experimentally obtained Basquin relation, these stresses were related to the number of cycles to failure, $N_{f\text{-model}}^{a_1/a_2}$. The obtained couples ($\sigma_{\text{est-model}}^{a_1/a_2}$, $N_{f\text{-model}}^{a_1/a_2}$) expressed fatigue data with estimated stresses from the simulation. $\sigma_{\text{est-model}}^{a_1/a_2}$ values were exchanged with the respective von Mises stresses, $\sigma_{\text{FEM}}^{a_1/a_2}$. See table 3.23 for a summary of the described procedure.

Table 3.23: Correlation of estimated stress values and stress values calculated by means of FEM for Tech-B

<table>
<thead>
<tr>
<th>$a_{\text{model}}$ [km/s$^2$]</th>
<th>$m_{\text{frac-model}}$ [kg]</th>
<th>$A_{\text{frac-model}}$ [m$^2$]</th>
<th>$\sigma_{\text{est-model}}^{a_1/a_2}$ [MPa]</th>
<th>$N_{f\text{-model}}^{a_1/a_2}$</th>
<th>$\sigma_{\text{FEM}}^{a_1/a_2}$ [MPa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>50</td>
<td>1,62·10$^{-6}$</td>
<td>7,85·10$^{-9}$</td>
<td>10,39</td>
<td>1,28·10$^6$</td>
<td>194</td>
</tr>
<tr>
<td>60</td>
<td>1,62·10$^{-6}$</td>
<td>7,85·10$^{-8}$</td>
<td>12,40</td>
<td>2,86·10$^4$</td>
<td>227</td>
</tr>
</tbody>
</table>

The two couples ($\sigma_{\text{FEM}}^{a_1/a_2}$, $N_{f\text{-model}}^{a_1/a_2}$) for $a_1=50$ km/s$^2$ and $a_2=60$ km/s$^2$ yielded two power law relations with two unknown parameters, $C_{\text{FEM}}$ and $b_{\text{FEM}}$ (see equation 3.28 and 3.29).

\[
\sigma_{\text{FEM}}^{a_1/a_2} = C_{\text{FEM}} \cdot (N_{f\text{-model}}^{a_1/a_2})^{b_{\text{FEM}}} \tag{3.27}
\]

\[
a_1 = 50 \frac{\text{km}}{\text{s}^2} \quad \rightarrow \quad 194 = C_{\text{FEM}} \cdot (1,28 \cdot 10^6)^{b_{\text{FEM}}} \tag{3.28}
\]

\[
a_2 = 60 \frac{\text{km}}{\text{s}^2} \quad \rightarrow \quad 227 = C_{\text{FEM}} \cdot (2,86 \cdot 10^4)^{b_{\text{FEM}}} \tag{3.29}
\]

Solving the system of equations 3.28 and 3.29 for $C_{\text{FEM}}$ and $b_{\text{FEM}}$, yielded the parameters for the fatigue model obtained with FEM calculated stresses. See table 3.24 for a summary of the conducted lifetime modeling and figure 3.47 for a graphical illustration.

Table 3.24: Summary of lifetime modelling data with FEM calculated values for Tech-B

<table>
<thead>
<tr>
<th>Basquin → all points</th>
<th>$C_{\text{FEM}}$ [MPa]</th>
<th>$b_{\text{FEM}}$ [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>346,995</td>
<td>-0,041</td>
</tr>
</tbody>
</table>
Figure 3.47: Lifetime curves for Tech-B samples based on calculated von Mises stresses.

Similar to the results obtained for Tech-A a considerable difference between lifetime models obtained by average shear stresses (see figure 3.46 and the curve computed by means of FEM (see figure 3.47) is observed for Tech-B samples. Underestimation of the geometrically induced dominant loading mode and complex state of stress may have contributed to the discrepancy as well as neglecting the stress concentration, definition of von Mises stress and different failure criteria.

### 3.7 Comparison of Tech A and Tech B Samples

Lifetime curves for Tech-A and Tech-B specimens were individually shown in section 3.5.6 and 3.6.6. Two methods based on estimated average shear stresses and von Mises stresses calculated by FEM were used in order to obtain lifetime models according to the Basquin power law relation. In the following a comparison between Tech-A and Tech-B is given by means of performed shear tests, lifetime modelling using estimated shear stresses and lifetime modelling using FEM calculated stresses.
3.7.1 Shear Test

Shear tests were performed according to section 2.5.4 for both samples (Tech-A and Tech-B) by Infineon Technologies AG. According to producer information the bonding parameters and material combinations of the ball bonds for both samples were comparable. Shear strength of the ball bonds was 175 ± 40 MPa for Tech-A and 150 ± 10 MPa for Tech-B respectively which also indicates a comparable bond connection.

3.7.2 Estimated Stresses

Experimentally obtained lifetime curves were expressed in terms of estimated average shear stresses for Tech-A and Tech-B. The models were based on a Basquin power law relation where $\sigma$ is the estimated average shear stress. The constants $C_{\text{estimated}}$ and $b_{\text{estimated}}$ were determined by a least square fit. Obtained curves for both samples are displayed in figure 3.48.

![Figure 3.48](image)

Figure 3.48: Life time curves of experimentally obtained fatigue data in terms of average shear stresses for Tech-A, indicated in blue (all data), green (upper lifetime area bound), purple (lower bound) and Tech B, indicated in red.

For both samples geometrically caused effects were not taken into consideration. The difference between the lifetime curves of Tech-A and
Tech-B may be related to considerable effect of the sample geometry on the experimental results as discussed in sections 3.5.6 and section 3.6.6. Thus it can be concluded that reasonable comparison of lifetime models obtained in this work can only be done for experiments using the same sample geometry.

Fatigue fracture for both samples mainly occurred in the thin Al metallization. Thin-coated metallic layers show different material properties than bulk materials [34] [3]. Due to this fact and the lack of published fatigue data for thin Al layers a comparison of obtained lifetime models for Tech-A and Tech-B with lifetime models from literature is neither reasonable nor applicable.

3.7.3 FEM Calculated Stresses

The experimentally obtained lifetime curves were expressed in terms of von Mises stresses for Tech-A and Tech-B by means of FEM analysis. The constants \( C_{FEM} \) and \( b_{FEM} \) were determined by correlating the experimentally obtained average shear stress values with the simulation. A plot displaying all lifetime models for Tech-A and B can be seen in Figure 3.49.

Contrary to experimentally determined lifetime curves based on the
estimated average shear stress, similar lifetime models in terms of von Mises stresses were obtained for Tech-A (blue) and B (red). As expected the sample geometry seems to have a pronounced influence on the results of fatigue tests, which can be considered and calculated by means of FEM simulation. The obtained lifetime models for Tech-A and Tech-B yield a comparable quality of thermosonically bonded Cu-Al interconnects in accordance with producer information and shear tests.
Summary

The main object of this thesis was to develop an accelerated test method for the reliability assessment of Cu-Al ball bonds and to propose lifetime prediction models for these interconnects by means of experimental procedures and Finite Elemente Method (FEM) simulation. Due to the fact that common methods for qualification of ball bonds either do not measure desired interconnect properties or are too time consuming, new technologies in order to assess the reliability of interconnects in microelectronic devices are required.

Developed by the research group Micromaterials at the University of Vienna the present work uses a mechanical setup to induce cyclic stresses in various types of metallic interconnects at a frequency of 20 kHz. This method was successfully applied for the lifetime estimation of thick Al wire wedge bonds (400 μm diameter) on Si chips. The wire bonds were subjected to cyclic loading and failed due to wire lift off. This failure mode is driven by thermal mismatch and describes the loss of electrical connection between contacts due to fracture of the bond interconnect. Wire lift off is the dominant failure in actual service life and conventional thermal and power cycling tests of microelectronic devices.

Obtained lifetime curves showed a good correlation with lifetime curves from conventional tests. This proofs that the proposed method is a time and costs saving method in order to evaluate fatigue parameters of thick Al wedge bonds [18].

The basic principle of the accelerated mechanical shear fatigue testing technique is based on an ultrasonic resonance fatigue setup. The micro-component is attached to the system and forced to longitudinal vibrations. Coupling of an active mass, which may be the mass of a chip, and the moving system is provided by a metallic interconnect. Due to the relative acceleration between the active mass and the oscillating system cyclic shear stresses are induced in the interconnect area, leading to fatigue failure.
The main aim of this work was to apply the accelerated testing method to extremely small Cu-Al ball bond interconnects. Using tailored specimen preparation, a method should be developed in order to obtain wire bond lift off failure of the Cu ball bond on the Al metallization pad. Lifetime curves in terms of experimentally obtained average shear stresses should be determined.

Under given experimental conditions, the vibrational response of the test samples with different geometries should be analysed by means of modal, static and transient FEM simulations.

Based on FEM calculations and experimental data, lifetime prediction models based on the Basquin relation should be established.

Specimens, called Tech-A and Tech-B, chosen for the present study were microelectronic devices as used in automotive applications. According to producer information the samples showed comparable bonding properties.

Samples called Tech-A consisted of 34 Cu ball bonds, which were thermosonically bonded to a thin Al metallization layer. Wire and ball bonds consisted of a 50 µm Cu wire, resulting in a ball bond with a diameter of approximately 130 µm and a bonding height of 40 µm. Lassnig performed experimental measurements yielding lifetime curves for Tech-A specimens [22]. In this work these results were used in order to propose a lifetime predicting model for multiple ball bond interconnects in terms of experimentally recorded average shear stresses and to perform structural FEM simulations for further lifetime modelling in terms of equivalent von Mises stresses.

Due to scattering of the experimental data obtained for Tech A specimens a lifetime range was defined which included an upper and lower level. This range was between 28 to 50 MPa at $10^5$ cycles and 25 to 36 MPa at $10^9$ in a Wöhler diagram. Geometrical effects related to the specimen preparation method were not considered in this approach.

In order to determine influences of the geometry on above mentioned experimental results, simulations using the FEM code ANSYS were performed, which included 3-dimensional models of the Tech A samples. These calculations resulted in lifetime curves based on calculated von Mises stresses covering a range of 160 to 230 MPa at $10^5$ cycles, and 150 to 190 MPa at $10^9$ cycles to failure.

Tech-B samples consisted of a single 50 µm Cu wire bonded to a thin Al metallization. Thermosonic bonding produced a ball bond with a
diameter of approximately 140 µm and a bond height of 40 µm. The mass of the ball bond was 4 µg and the bonding interface showed an interconnection surface of 0.01 mm².

Because of the very low mass and the relative high interface area of the ball bond the initial specimen geometry did not provide the required geometry, active mass and inertia in order to induce sufficient cyclic shear stresses for fatigue loading in the interface of the ball bonded interconnect.

The main experimental challenge of the present work was to find an applicable sample preparation method for experimental measurement of fatigue life curves of Tech-B samples. Experimental success for this specific specimen geometry and metallization was achieved with a special soldering technique. A suitable sample geometry and ratio of mass to contact area was obtained by utilizing the fact that Cu can readily be soldered, whereas the underlying Al pad does not react with conventional Pb-Sn solders. Thus the active mass of the Cu ball bond could be increased while the solder material did not wet the metallization pad. Using this specially designed specimen preparation technique, lifetime curves could be obtained for single ball bonded specimens of Tech-B. Experimentally measured lifetime curves in terms of average shear stresses showed values of 13 MPa at $10^5$ cycles and 6 MPa at $10^9$ loading cycles.

Figure 3.50 illustrates the experimental procedure for determination of fatigue lifetime of multiple and single Cu-Al ball bond interconnects by means of accelerated mechanical testing. It can be seen that the obtained lifetime curves based on experimentally determined average shear stress for Tech A and Tech B specimens show different levels. Since the bonding quality of both sample types is almost similar (similar bonding parameters and shear strength values), the difference in the fatigue response is related to the effect of sample geometry as confirmed by FEM simulations.

Similar to Tech-A, the lifetime curve obtained for Tech B based on experimentally determined average shear stress does not include different loading modes and complex stress states caused by the sample geometry. On the basis of experimentally obtained results and FEM supported calculations a lifetime model was proposed including effects of the sample geometry for Tech-B. Obtained lifetime curves showed a range of equivalent stresses of 150 MPa to 250 MPa between $10^5$ and $10^9$ loading cycles.

In contrary to the estimated approach equivalent von Mises stresses and lifetime models obtained with FEM simulation show comparable values for the samples of Tech-A and Tech-B. The procedure of lifetime prediction
for Cu-Al ball bonded interconnects using experimental results and FEM simulations for both specimen geometries is illustrated in figure 3.51. The lifetime range in terms of equivalent von Mises stresses defined for Tech-A samples of 160 to 230 MPa at $10^5$ and 150 to 190 MPa at $10^9$ totally confines the lifetime curve for Tech-B ranging from 220 MPa to 150 MPa between $10^5$ to $10^9$ cycles to failure.

Including geometry effects by structural FEM simulations, analysis of the geometrically induced cyclic loading modes, and considering the equivalent state of stress in the samples, lead to a comparable lifetime regime of both samples and yields a comparable quality of ball bond connection for both types of technologies, in conformity with producer information and shear tests.

Fatigue fracture for both samples was observed by means of SEM microscopy and occurred mainly in the 6 $\mu$m thin Al metallization. Thin-coated metallic layers show different material properties as bulk materials. Due to this fact and the lack of published fatigue data for thin Al layers a
comparison of obtained lifetime models for Tech-A and Tech-B with lifetime models from literature is not possible.

Figure 3.51: Schematic of FEM supported lifetime modelling.

In the frame of this work it was possible to obtain lifetime curves and propose lifetime predicting models for extremely small Cu-Al ball bonded interconnections. In cooperation with Lassnig [24] these results were obtained for the first time worldwide. Challenges were accomplished by the development and application of a tailored sample preparation, the use of a fast mechanical fatigue testing setup and FEM analysis.

Sample preparation and FEM analysis have to be improved in order to be applicable for industrial use. Improvements may be achieved by the use of micromanipulators and more suitable soldering templates for the experimental procedure. Better knowledge about the complex multilayer structure is required to optimize results obtained by FEM simulation.
Zusammenfassung


Die Methode zur beschleunigten Lebensdauerprüfung von metallischen Mikroverbindungen basiert darauf, ein elektronisches Bauteil an ein Ultraschall-Resonanz-System zu befestigen und das entstandene System zu longitudinalen Schwingungen anzuregen. Die Kopplung zwischen jenem Teil


Die verwendeten Proben, genannt Tech-A und Tech-B, sind mikroelektronische Leistungsbauteile, die in der Automobilindustrie zu Steuerungszwecken Anwendung finden. Herstellerangaben zufolge sind die jeweiligen "Bond"-Parameter vergleichbar, was auch durch Schertests bestätigt wurde.


Für die experimentellen Wöhler-Kurven wurde eine vereinfachte Methode zur Berechnung von Schubspannungen verwendet. Aufgrund der Streuung der Messergebnisse war es sinnvoll für diese Probe einen Lebensdauer-Bereich zu definieren, der von 28 bis 50 MPa bei $10^5$ Lastzyklen zu 25 bis 36 MPa bei $10^9$ reicht.

Dieser Ansatz zog keinerlei geometriebedingte Einflüsse in Betracht, weshalb mit dem FEM Programm ANSYS strukturelle Analysen durchgeführt wurden, die ein 3-dimensionales Modell der Probe beinhalteten. Lebensdauerkurven die mittels FEM berechnet wurden lagen zwischen 230 MPa und 150 MPa bei Lastzyklen von $10^5$ bis $10^9$.  

106


Im Gegensatz zu diesen Ergebnissen zeigen von Mises-Vergleichsspannungen und Lebensdauermodelle, die mittels FEM berechnet wurden, vergleichbare Werte. Der Lebensdauerbereich definiert für Tech-A von 160-230 MPa bei 10⁵ bis 150-190 MPa bei 10⁹ Zyklen schließt die Lebensdauerkurve für Tech-B, mit Werten von 220 MPa bis 150 MPa, vollständig ein.

Durch die Einbeziehung geometrieführender Faktoren in der Finite-Elemente-Simulation ergaben sich für die daraus erhaltenen Lebensdauerkurven vergleichbare Werte, was auch Herstellerangaben und Schertests entspricht. In den Abbildung 3.50 und 3.51 ist eine Übersicht der angewendeten
Verfahren, Methoden und Ergebnisse dargestellt.

Ermüdungsbrüche traten bei beiden Proben hauptsächlich in der 6 µm dünnen Aluminium-Metallisierung auf. Da sich die Ermüdungseigenschaften von dünnen Schichten entscheidend von denen massiver Körper unterscheiden und keinerlei Ermüdungsdaten in der Literatur über dünne Aluminium-Schichten gefunden wurden, ist ein Vergleich der in dieser Arbeit gemessenen Ermüdungsparameter mit Werten aus der Literatur nicht möglich [34][3].

Durch diese Arbeit war es möglich Lebensdauerkurven von Cu-Al-"Ball Bond"-Verbindungen experimentell zu bestimmen und entsprechende Lebensdauermodelle auf Basis der Basquin Relation zu erstellen. In Zusammenarbeit mit Alice Lassnig stellen die erhaltenen Ergebnisse die erste, erfolgreiche Ermüdunguntersuchung für kleine Cu-Al-"Ball Bond"-Verbindungen im "High Cycle Fatigue"-Bereich (10⁵ bis 10⁹ Lastzyklen) weltweit dar. Experimentelle Messungen und anschließende Analysen wurden durch spezielle Probenpräparationen, der Verwendung eines Ultraschall Resonanz Ermüdungssystems und der Finite Elemente Methode durchgeführt.

Bibliography


Curriculum Vitae

Personal Information:
Surname / First name: Trasischker Wolfgang
Address and Contact: Pichlern 42
9562 Himmelberg
Austria
Phone: +43 (0) 650 8209986
E-mail: wolfgangtra@gmx.at
Nationality: Austrian
Date of birth: 20 April 1985

Studies and Education:

Academic Studies
October 2006 – April 2011
Physics
University of Vienna

Academic Residence
February – July 2009
Complutense University – Faculty of Physics
Madrid, Spain

Schooling
1999 – 2004
Secondary technical college Villach
Civil engineering
Final exam with distinction

1995 – 1999
Grammar school Feldkirchen

Diploma Thesis:
March 2010 – April 2011
“Accelerated Lifetime Estimation of Copper Ball Bonds using Experimental Methods and Finite Element Analysis”
University of Vienna in cooperation with Infineon Technologies Austria – Comet-K Project