Modeling Damage in Large and Heavy Electronic Components Due to Dynamic Loading

Ed Habtour, U.S. Army Materiel Systems Analysis Activity
Gary S. Drake, U.S. Army Materiel Systems Analysis Activity
Christopher Davies, ManTech International

Key Words: Reliability, Physics of Failure, failure mechanisms, fatigue, vibration, multiaxial, electronics

SUMMARY & CONCLUSIONS

The performance of the next-generation U.S. Army platforms, such as the Small Unmanned Ground Vehicles (SUGV) and Small Unmanned Aerial Vehicles (SUAV), is strongly dependent on electronics. These electronic systems may experience harsh dynamic loads due to shock and vibration. These loads may cause significant damage to electronic component packages, leads and solder joints. The damage can be due to a combination of bending moments in Printed Circuit Boards (PCBs) and/or inertias of large/heavy components. When modeling a PCB, the typical approach in electronics Physics of Failure (PoF) is to employ a two-dimensional (2-D) finite element analysis (FEA) which uses the “smeared” properties technique. Such an approach may not fully address the inertias of large/heavy components and their local stiffness.

Through PoF, the US Army Materiel Systems Analysis Activity (AMSAA) is currently investigating a 2-D combined with three-dimensional (3-D) FEA approach to assess the reliability of large/heavy electronic components. The goal is to evaluate whether this approach could detect failures earlier in the development cycle. The failure mode in these components perpetuated by the local acceleration is considered. This approach may produce more cost effective and reliable models for capturing unforeseen design defects and prevent damage instigated by an unexpected excitation mode.

1 INTRODUCTION

In military applications, electronic devices play a vital role in the success of a mission. These devices which provide control, guidance, communication, and reconnaissance are vital components in modern unmanned vehicular applications. This trend in modern warfare has increased the complexity of electronic equipment, especially in low volume, highly sophisticated, and dense electronic systems. Figures 1 and 2 show the SUGV and SUAV (Erwin, 2010). These modern systems take advantage of the remarkable advances made in low cost commercial electronics. It is becoming progressively more beneficial to use such components in military applications for improved computational performance, on-demand availability, addressing obsolescence, and providing state-of-the-art capabilities. This current movement of using commercial-off-the-shelf (COTS) electronics and devices for military applications has led to concerns about their reliability in harsh battlefield environments. Therefore the complexity of the autonomous military platforms that take advantage of COTS has resulted in an increased need to improve the reliability of these components by understanding the failure mechanisms due to dynamic loads through Physics of Failure (PoF). Typically, these types of systems are subjected to various complex loadings, including shock and vibration, during their life-cycle. These loads may impose significant stresses on the PCB substrate, component packages, leads and solder joints [1]. These stresses can be due to a combination of bending moments in the PCB and/or inertias of components. They may lead to several failures such as delamination in the PCB, solder joint fatigue, lead fracture or structural damage to components.

Figure 1 - SUGV

Figure 2 - SUAV

U.S. Government work not protected by U.S. copyright
When conducting PoF analysis of electronic systems, the large variety of package types is perhaps one of the main challenges to consider, since failure may occur due to one of several failure drivers. One of the most frequent failures in electronics is interconnect damage in heavy components with large center of mass (CM) and low profile surface mount packages (SMT). The failure in heavy components with large CM can be predominantly due to inertial loads while in light low-profile SMT packages the dominant stress source can be due to board deflection. Both of these failure drivers may compete in heavy and large electronic components such as inductors and transformers. Depending on the architecture of these components, they can also potentially alter the local vibration response significantly. It is common to increase the board stiffness to reduce the overall response of the PCB. However, increasing the board stiffness may increase local bending moments.

The current available vibration fatigue life prediction methods for large/heavy components force reliability engineers to use one of two extremes. One method is to construct a detailed 3-D FEA. This approach may be impractical when dealing with large circuit card assemblies (CCA) containing many components, each with multiple leads and solder joints and can be computationally expensive. The other extreme is to use simple empirical equations. Probably the best-known method to estimate component life under vibration is Steinberg’s model [2]. However, these models are of questionable use for engineers since they were based on personal experiences. Furthermore, they cannot be used alone to evaluate new products or emerging technologies with a high level of confidence.

This paper is concerned with a rapid analytical technique for analyzing heavy/large components that can provide an engineer high fidelity assessment while reducing the computational time. A PoF approach was developed that may improve the reliability assessment of CCAs containing large/heavy components. This approach is a hybrid-method that combines 2-D and 3-D FEA where the mechanical and inertial properties of the components at the local level are taken into consideration. These properties may be used to extract accurate natural frequency values for the CCA.

2 PHYSICS OF FAILURE ANALYTICAL APPROACH

2.1 Simplified Single Degree of Freedom Approach

As mentioned above one of the most well known simplified models in the area of PCB vibration fatigue is Steinberg’s model. Steinberg has proposed an empirical equation for designing PCB’s in vibration environments where the maximum deflection of the PWB is less than a critical displacement value, d [2]:

\[ d = \frac{0.00022 B}{C h r \sqrt{L}} \]

(1)

where B is the length of the PCB edge parallel to the component located at the center of the board in units of inches. L and h are the length of the component and the thickness of the PCB in inches, respectively. C is a constant coefficient which depends on the component type and r is the relative position factor of the component relative to the PCB. The Steinberg model assumes a dynamic single-amplitude displacement for the PCB. This model is valid only for single-degree-of-freedom (SDOF) systems. The out-of-plan root-mean-square (rms) displacement is calculated as follows [2]:

\[ Z_{\text{rms}} = \frac{9.8 G_{\text{rms}}}{f_n^2} \]

(2)

where \( f_n \) is the natural frequency and \( G_{\text{rms}} \) is the root-mean-square output acceleration. The \( G_{\text{rms}} \) can be estimated using Miles’ equation:

\[ G_{\text{rms}} = \sqrt{\frac{\pi P f_n Q}{2}} \text{ where } Q = \sqrt{f_n} \]

(3)

where P is the input Power Spectral Density and Q is the transmissibility. Steinberg states when the dynamic single-amplitude displacement at the center of the PCB is limited to the critical value d, the component is expected to achieve a fatigue life of 20 million stress reversals in a random vibration environment and 10 million stress reversals under sinusoidal vibration. One must be cautious when using this model since Steinberg’s empirical approach is based on his testing and personal experience. Thus, this empirical approach may be more accurate when applied to PCBs assembled in exactly the same manner. The drawback of his model is that it cannot be used outside the range and configuration of the assembly used in the derivation of the model. It cannot be used to evaluate new products or emerging technologies with high confidence. Nonetheless, this approach may help designers obtain a rough estimate of the fatigue life due to vibration and can be enhanced further with caution.

Electronically dense military platforms have to endure severe and complex environmental conditions that involve not only thermal but also dynamic loading resulting in high-cycle fatigue. Thus, using Steinberg’s model alone may not be an adequate approach for assessing the survivability of military devices.

In this study a 127x101.6 mm^2 PCB with six large/heavy inductors was designed, as shown in Figure 3. The inductor geometry is shown in Figure 4.

Figure 3 - CAD model of PCB with large components
the reliability of large electronic components. A more cost effective approach combining 2-D and 3-D FEA was developed to extract the natural frequency of the PCB and more accurately determine the maximum deflection, curvature, and stress of the PCB. The goal is to evaluate whether this approach could detect critical failures earlier in the development cycle. Future plans were developed to conduct vibration testing using a multi-degrees of freedom (M-DoF) electrodynamic (ED) shaker to evaluate this approach and assess the effect of large/heavy components local inertia on the fatigue life.

Figure 4 - Large inductor used in this study

2.2 Two-Dimensional Approach

A more analytical approach that can be employed other than the Steinberg model when conducting PoF analysis is simplified 2-D (plate or shell) FEA of a PCB. The mass of the components are “smeared” over their PCB footprints to reduce computational time and cost. This method was developed by Pitarresi and Primaver where they performed experimental and FEA modeling work to characterize the natural frequencies, mode shape, and transmissibility at the board level [3]. Later, they used the simple plate vibration models, using the property smearing approaches, as well as detailed finite element modeling work to characterize the natural frequencies, mode shape, and transmissibility at the board level [3]. In the case where the local inertias are significant, a traditional 3-D FEA might be necessary. Transforming a 3-D PCB model in Figure 3 to 2-D FEA using the smeared technique is shown in Figure 5. The PCB was discretized into rectangular shell elements, as shown in Figure 5. The individual elements were defined by four nodes. Typically in traditional FEA, the shell element nodes in a continuum structure have six-DoF. For PCB PoF analysis, the number of degrees of freedom was reduced to three which include one out of plane displacement, \( u_z \), and two rotations about the orthogonal axes in the plane of the board. The displacement of each node is driven by the element’s stiffness matrix. Each value of the element stiffness matrix is a function of the constitutive material properties. Because PCBs are multilayer composite construction, laminated plate theory was used to calculate the element stiffness matrix. The layers’ geometric and material properties were designed symmetrically about the middle surface of the board. Therefore, the bending-extension coupling effect was not a cause for concern and the extension and the coupling matrixes were eliminated [4]. This led to a simplified plate equation with the bending stiffness only:

\[
\begin{bmatrix}
M_x \\
M_y \\
M_{xy}
\end{bmatrix} =
\begin{bmatrix}
D_{11} & D_{12} & 0 \\
D_{12} & D_{22} & 0 \\
0 & 0 & D_{33}
\end{bmatrix}
\begin{bmatrix}
\kappa_{xx} \\
\kappa_{yy} \\
2\kappa_{xy}
\end{bmatrix}
\]

(4)

where \( \{M\} \) and \( \{\kappa\} \) are the moment resultant and board curvature vectors. The flexural rigidity (or bending) matrix, \( [D] \), is the measure of how easily the board bends. \( [D] \) can be calculated as follows:

\[
D_{ij} = \frac{1}{3} \sum_{k=1}^{n} Q_{ij}^k (h_k^3 - h_{k-1}^3) \text{ for } i,j = 1,2,3
\]

(5)

where \( i \) and \( j \) coincide with the natural axes of the material and \( n \) is the number of layers in the PCB. The distance of the outermost fiber of the \( k^{th} \) layer from the mid-surface of the plate is \( h_k \). \( Q_{ij} \) is the stiffness matrix for each layer. The reader may refer to Jones textbook [4] for additional information on how to calculate \( Q_{ij} \). For a uniform, symmetrical, homogeneous composite construction the board rigidity, \( D \), may be reduced to:

\[
D = \frac{E \ t^3}{12(1 - \nu^2)}
\]

(6)

where \( E \) is the elastic modulus, \( t \) is the board thickness and \( \nu \) is the Poisson ratio. After obtaining the \( [D] \) matrix, the board curvature can be calculated from the simplified plate equation (4). The smeared-technique was then implemented using 2-D shell elements. The first mode natural frequency for this board was 108 Hz, where mass properties of the components were included but not their stiffness.

Figure 5 - 2-D FEA using smearing method

The smeared technique represented the mass of the board and components by simply increasing the mass of the shell element under the footprint of each component. However, for large/heavy thru-hole components, some 2-D codes may include the stiffness of the leads or completely ignore it. In this approach, the mass and stiffness effects were included by locally increasing the PCB’s density and Young’s modulus, respectively. Unfortunately, such an approach does not address the inertias of large components. In the case where inertia is significant, a traditional 3-D FEA might be necessary.
2.3 Three-Dimensional Approach

In this study, modal analyses were conducted on just the inductor with various standoff heights as shown in Table 1. In this task, the component was assumed to be fixed at the leads-board interface, as shown in Figure 4. As expected the modal response dropped as the standoff height increased due to the component’s significant inertia which is typically neglected in the smeared properties technique. Resulting mode shapes are shown in Figure 6. Even more interestingly, the modal response of the components dropped significantly when the components were modeled as part of the PCB, where the boundary conditions are more representative of real systems, as shown in Figure 7. The standoff height in this analysis was 2.0 mm and the maximum response occurred in the components located at the middle of the PCB. The first vibration mode for the middle components and the components closer to the fixed edges were approximately 69 and 71 Hz. These values were lower than the local analysis of the component. This is because the leads are fixed to the PCB and are not fixed globally. The PCBs first vibration mode was at 119 Hz. The PCB modal shape is shown in Figure 8.

Table 1 - Inductor modal response for various standoff heights

<table>
<thead>
<tr>
<th>Standoff Height (mm)</th>
<th>Mode I (Hz)</th>
<th>Mode II (Hz)</th>
<th>Mode III (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5</td>
<td>108</td>
<td>1733</td>
<td>3266</td>
</tr>
<tr>
<td>1.0</td>
<td>102</td>
<td>1657</td>
<td>3161</td>
</tr>
<tr>
<td>1.5</td>
<td>98</td>
<td>1560</td>
<td>2682</td>
</tr>
<tr>
<td>2.0</td>
<td>94</td>
<td>1430</td>
<td>2200</td>
</tr>
</tbody>
</table>

Figure 6 - Inductor modal shapes when fixed at leads

In typical vibration fatigue analysis, the PCB is treated as a thin plate. Therefore, many researchers reasoned that the PCB’s natural frequency is dependent upon the geometry and the material of the board and not necessarily the components properties [5]. This might be valid for low inertia microelectronic components, where the drop in natural frequency due to the added mass of the components is compensated for by the increase in the natural frequency due to the local increase in stiffness from component mounting. However, for large/heavy components the scheme may not be applicable since the increase in the component mass and the leads stiffness might not cancel each other. The first vibration mode for the PCB described above when neglecting both the inertial and stiffness effects of the components was approximately 310 Hz. Clearly, neglecting the mass and stiffness effects of the inductor over estimated the natural frequency of the PCB. If the mass effect of the components was included only in this particular PCB, the natural frequency was 108 Hz, which was slightly less than the first vibration mode obtained from the 3-D FEA model.

Figure 7 - Middle Inductors first modal response in PCB

Figure 8 - PCB first modal response

2.4 Combined Two-Three-Dimensional Approach

In this study the simplicity of the global 2-D FEA was combined with a more detailed local 3-D FEA. The advantages of this approach are significant cost and time reduction while maintaining high fidelity analysis. In this approach, the global model was considered first. Based on the knowledge of the local PCB warpage at the component of interest, the local stiffness was determined. The warpage was evaluated by simply applying a local unit load to the local model, as shown in Figure 9. This caused the PCB to experience small dynamic deflection or warpage. The local deformation was modeled with two radii of curvature. This was accomplished through the use of Kirchhoff-plate moment-curvature equations (found in several structural textbooks), where the local radii of curvature and the local applied bending moments are related as follows:

\[
\begin{bmatrix}
M_{xx} \\
M_{yy} \\
M_{xy}
\end{bmatrix}
= \begin{bmatrix}
D_{11} & D_{12} & 0 \\
D_{12} & D_{22} & 0 \\
0 & 0 & D_{33}
\end{bmatrix}
\begin{bmatrix}
\kappa_{xx} \\
\kappa_{yy} \\
2\kappa_{xy}
\end{bmatrix}
\]

where, \(D_{ij}=E_{ij}h^3/12\) for \(i, j = 1, 2, 3\). For simplicity, isotropic material was assumed in this paper. Therefore the local stiffness can be calculated as follows:
\[
D_{11} = \frac{M_x - \nu M_y}{K_x(1 - \nu^2)}
\]  
(8)

Similarly,
\[
D_{22} = \frac{M_y - \nu M_x}{K_y(1 - \nu^2)}
\]  
(9)

The deflections were obtained from the 3-D FEA model. The curvature can be calculated by the relationship below which can be obtained in any calculus book:
\[
\kappa = \frac{d^2 y}{dx^2} \cdot \frac{1 + \frac{d^2 y}{dx^2}}{(1 + \frac{d^2 y}{dx^2})^{3/2}}
\]  
(10)

where, the elastic curve was expressed mathematically as \(y=f(x)\). The component was assumed to remain rigid and all the deformation occurs in the PWB and the leads. This assumption was made in the local model to make the displacement calculation more manageable.

The global model was constructed in the manner discussed in section 2.2. However, the local flexural rigidity matrix, \([D]\), was replaced with the ones calculated from the local 3-D FEA model. The first vibration mode was then obtained from the global model which was 120 Hz. This value was close to the full 3-D FEA model discussed in 2.3. Therefore, one may use a combined two--three-dimensional approach to reach similar results to a full 3-D FEA with the advantages of less computational time and cost reduction.

3 TESTING METHODOLOGIES

When performing electronics PoF for ground vehicles, there are two types of motion in random vibration. One motion is the induced curvature or bending in the PCB as the assembly moves in a vibratory manner (global motion). The other motion is the individual components moving with respect to the PCB due to the compliance of the components’ attachment (local motion). To accurately assess the excitations from the vehicle to the component level, some researchers have suggested modeling the dynamic response of the vehicle subsystems. This approach, however, can be an arduous task [6]. The main reason for this lies in the fact that the vehicle chassis and body are complex systems. The reaction forces and vibration velocities depend not only on the strength of excitation within the chassis but also on the coupling of the chassis and the subsystems.

Thus, one has no choice but to count on engineering judgment in estimating the boundary conditions and system inputs. A more practical approach perhaps is using experimental Frequency Response Function (FRF) data to represent the body then combine it with the FEA models. Therefore, a M-DoF ED shaker at the University of Maryland Center for Advanced Life Cycle Engineering (CALCE) will be utilized to excite the PCBs, with large insertion-mount components, to levels seen on the battlefield. The FRF experimental data will be combined with the FEA model where the solder fatigue results would be extracted with the aid of FEA.

The M-DoF ED shaker at CALCE was developed by TEAM Corporation. It consists of eight plane actuators and four out of plane actuators underneath the shaker table, as shown in Figure 10. The twelve ED shakers are mechanically coupled to the table. This architecture allows the shaker to produce a true M-DoF vibration environment. Each axis has four shakers with 200 in-lbf rotation per axis. The excitation limit is up to 30Gs with 0-3000Hz. Unlike other testing methodologies, M-DoF ED shakers will provide a clearer knowledge of the failure mechanisms in electromechanical devices. This is because they provide both qualitative and quantitative understanding of the failures not present in single-axis excitation [7]. The inputs can be controlled for all axes as demonstrated by CALCE, Figure 11. This is because the twelve shakers can be excited independently of each other. Figure 11 shows excellent control of the shape of the excitation PSD profile. This approach may help in establishing a quantitative relationship between performance in the battlefield and performance in the test.

4 FUTURE WORK

Vibration durability tests will be conducted on the M-DoF ED shaker for various orientations: out-of-plane, in-plane, simultaneous in-plane and out-of-plane excitation and sequential in-plane and out-of-plane excitations. Subsequently destructive physical analysis of failed specimens will be performed. The final step will be to develop a PoF modeling approach for vibration durability under random, multi-modal and M-DOF excitations. The natural frequencies and mode shapes will be extracted from the FEA modal analysis and
compared with testing results. For this step, the test specimens will be fabricated (PCB with heavy through-hole inductors) to conduct a M-DoF vibration durability test on the M-DoF shaker. Additionally, FEA simulation will be carried out to characterize the dynamic response of the test PCB. Response characterization will mainly include dynamic strain history collected on the test vehicle. Corresponding FEA will be conducted and calibrated, based on this experiment’s result. The characteristic flexural strain at different locations on the test board can then be either measured with gages or estimated from the FEA model, under different excitation levels. These strain time histories are then used to construct strain range distribution functions of the PCB, based on cycle counting.

Figure 11- Probability Density in Multiaxial ED Shaker

5 OUTCOMES

As shown above the damage can be a combination of bending moments in PCBs and/or inertias of large/heavy components. When conducting electronics PoF, the two-three-dimensional FEA approach may provide natural frequency results closer to full 3-D FEA while reducing cost and computational time.

Nonetheless, it is essential to understand the structural characteristics of large/heavy components in electronics devices in order to correlate the defects with the dynamic responses. As mentioned above, the main challenge in electronics packaging is the prediction of the reliability and lifetime of the critical components. Therefore, it is imperative to identify the failure mechanisms of the components through experimental analysis. However, the experimental approach has to emulate the real world operational conditions, which includes simulating M-DoF dynamic loads. This involves experimentally measuring the transient in-plane and out-of-plane displacement responses which can be accomplished with the aid of a multiaxial shaker.

This investigation will be utilized to enhance and improve existing standards for dealing with complex dynamic loading in electronics. It may also provide a means to validate and improve existing physics of failure models.

REFERENCES


BIOGRAPHY

Ed Habtour
USAMSAA
392 Hopkins Road
APG, MD 21005-5017
email: amsaa.reltools@us.army.mil

Ed is a Mechanical Engineer at AMSAA. He earned his bachelor’s degrees in Mechanical Engineering at Utah State Univ. and his master’s degree in Mechanical Engineering at Johns Hopkins. He is currently pursuing a Ph.D. in Mechanical Engineering at the Univ. of Maryland.

Gary S. Drake
USAMSAA
392 Hopkins Road
APG, MD 21005-5017
email: amsaa.reltools@us.army.mil

Gary is Team Leader of the Electronics Physics of Failure Team at AMSAA. He earned his bachelor’s degree in Mathematics at the Univ. of Maryland and his master’s degree in Administrative Science at the Johns Hopkins University.

Christopher Davies
ManTech International
392 Hopkins Road
APG, MD 21005-5017
email: amsaa.reltools@us.army.mil

Christopher is a Mechanical Engineer supporting Electronics Physics of Failure at AMSAA. He earned his bachelor’s and master’s degree in Mechanical Engineering at The Ohio State University.