Creating an User Interface that Provides Optimized Distribution of Microelectronic Components on the Printed Circuit Board Used in Automobile Lighting Systems

Fahri Berk Bilbay¹, Barış Ediz², Enes Çal³, Mustafa Cemal Çakır⁴

¹,²,³ R & D Department, Marelli Türkiye, Bursa, Türkiye
⁴Mechanical Engineering, Uludağ University, Bursa, Türkiye
E-mail: bilbayberk@gmail.com

Abstract—Printed circuit boards (PCBs) can be in various sizes depending on the application. In addition, depending on the application, the microelectronic components (transistors, capacitors, resistors, integrated circuits, etc.) and the location of these components on the PCB may vary. This situation makes it very difficult to define the design concept of the PCBs by considering certain loading conditions. Vibration loadings can cause damage on the PCBs and also on the electronic components. It is very important to take into account of certain dynamic test conditions in automotive lighting industry when designing and validating the PCB’s.

The aim of this study is to take into account of the road conditions when designing PCBs used in automotive lighting systems and to distribute the components to the appropriate locations by using the design of experimental (DoE) method. In the study, a finite element model of the PCB with all its components was created and correlated with physical testing by using the dynamic test scenario frequently used in the industry. The locations of the components on the PCB were determined as the design parameters, the stresses and fatigue damage obtained after random vibration analysis as the response functions in the DoE method. As a result, an interface has been created where the designer can choose the component locations on the board based on test specifications with the virtually/physically verified analyses, rather than based on user experience. This interface also allows the designer to perform a pass/fail check of the PCB for fatigue damage and obtaining the optimum component locations.

Index Terms—Printed Circuit Board (PCB); Design of Experiment (DoE); Microelectronics Reliability; Optimization; IC Test and Validation

I. INTRODUCTION

Due to the anisotropic material structure of the printed circuit boards (PCBs), the variety of electronic components (transistors, resistors, diodes, integrated circuits, etc.), harsh test conditions and the sensitivity of electronic components on the board to damage due to vibrations, it is important that the PCB concept design must be accomplished by taking these factors into account.

Printed circuit boards, which are widely used in the automotive industry, may be exposed to various dynamic loads and vibrations at different frequencies during their service life. Vibration loadings on electronic cards may occur during transportation, handling, manufacturing and customer’s end usage. Therefore, structural reliability evaluation of the vibration-induced circuit board in the electronic packaging has become an important issue in the industry. In addition, the temperature generated during the operation of the LED components on the PCBs used in the automotive lighting industry and the vibration exerted by the harsh environments have a combined effect. For this reason, it is very important to consider both situations when performing virtual analysis or physical test conditions of the PCBs.

PCBs have a sandwich structure consisting of an insulating composite layer between two conductive copper layers. The material property of the printed circuit board cannot be defined as isotropic due to this special structure. In addition, variables such as the types of hundreds of components on the circuit board, material properties, types of the solder leads (which connect components to PCB surface), and the thickness of the sandwich structure make it very difficult to create and verify the finite element (FE) model of the circuit board [1]. Cracks may occur on the core layer or on the solder leads if the PCB design cannot withstand the loads under test conditions during the design phase and this makes the card unusable.

In the literature, it is possible to find studies on improving the circuit board model by the use of finite analysis and experimental design methods.

Burmitskii et al.[2] accomplished a modal analysis of the detailed PCB model with electronic components and simplified PCB model by comparing the first two modes. In simplified model, electronic components such as resistors, capacitors and LEDs were eliminated from the model and added their masses to the PCB. As a result, a 3.5% difference was seen between the first two modes of the detailed and simplified model. Ren et al. [3] performed modal analysis using free boundary conditions in the FE model to obtain the natural frequencies and vibration modes of the PCB. FE results were compared with the experimental modal test to verify the analysis result. PCB with different thicknesses were compared and stated that the error rate between analysis and test models were below 10%. Somashekar et al. [5] obtained the natural frequencies and of PCB by using the finite element method (FEM) to investigate the dynamic properties of printed circuit boards and avoid many test attempts and confirmed them with physical tests.
Maia et al. [6] conducted experimental tests with three different solder lead designs. It was applied the same test scenario in a virtual environment and stated that the physical/virtual test correlation was at a good level. Che et al. [7] created a FE model of the PCB and solved the random vibration scenario in the frequency range of 20 to 2000 Hz. In the model, especially solder lead that connecting the integrated circuit and the PCB together was detailed. The same scenario as a physical test was accomplished and compared with fatigue life. As a result, it was seen that the solder leads were the critical region and the damage primarily occurred in this region. The cracks in the physical test were also seen at these solder leads. García et al. [8] used a FE model of PCB to simulate the vibration loads used in space vehicles during the takeoff. Random vibration analysis scenario was solved to examine the fatigue life of the solder leads. As a result, it was stated that the damage in the physical fatigue test occurred at the solder leads, as predicted by the finite element analysis (FEA) result with the good correlation. Verma et al. [9] performed modal and random vibration analyzes for the PCB. Several random vibration scenarios were used and predicted the fatigue life of the PCB. The result of random vibration analyzes showed that the maximum stresses mostly occurred in the solder lead areas of the components.

Tinja et al. [10] created FE and physical test models to examine the behavior of the PCB under static and dynamic loads. Sherlock software was used to define the layout of the components on the PCB and the materials of these components in a FE model. Aoues et al. [11] determined various design variables and examined their effects on the fatigue life of solder leads. Parameters such as temperature, soldering lead length and component thickness were chosen as design variables. As a result, the effect of design parameters on PCB fatigue damage were investigated. Han [12] conducted vibration and PSD analyzes of the PCB in different components positions. The analysis of PCB was solved and compared with fatigue life of each position. As a result, it was stated that maximum stresses occur in the solder leads and that these stresses directly affect fatigue life. It was also stated that it is possible to optimize components positions since the maximum stresses are clearly visible. Gao et al. [13] performed modal and fatigue analyzes in FEM and modeled the electronic components on the PCB in a simple way. The electronic components on the PCB were located in three different locations and examined their reliability. As a result, it was stated that the locations of the electronic components on the PCB greatly affected the stress distributions and fatigue life on the PCB. Gao [14] used FEM to obtain the modal frequencies of a printed circuit board and extracted its first 6 modes. The boundary condition coordinates of the PCB defined as design parameters and modal frequencies as response functions. As a result, it was stated that the most appropriate connection center coordinates were obtained after optimization.

Bilbay et al. [15] created a FE model for the PCB that was correlated with physical testing. With the LED light on, three strain gauges were placed in the areas where the transistors were located, and instantaneous strain data were read and recorded. FEA was performed for the PCB by including the temperature map with the LED light on under the same boundary conditions, and the strain values in the same regions were obtained. As a result, it was obtained a correlated virtual PCB material model with the physical test. Kalyani et al. [16] conducted modal analysis of PCB with isotropic and anisotropic material properties to examine the material characteristics of PCB. The elasticity module of the PCB defined as a design variable and compared its modules in three directions. As a result, it was stated that the effect of the elasticity modules in the X and Y directions on the analysis are much more effective than the Z direction. Xu et al. [17] conducted modal, and fatigue analyzes by using the FEM to obtain the most suitable PCB model and determined certain design variables (material properties of the PCB) and response functions. As a result of the optimization work, it was obtained the most suitable PCB model for the safety of solders in critical areas.

The aim of this study is to take into account of the road conditions when designing PCBs used in automotive lighting systems and to distribute the components to the appropriate locations by using the design of experimental (DoE) method. In the study, a finite element model of the PCB with all its components was created and correlated with physical testing by using the dynamic test scenario frequently used in the industry. The locations of the components on the PCB were determined as the design parameters, the stresses and fatigue damage obtained after random vibration analysis as the response functions in the DoE method. As a result, an interface has been created where the PCB designer can choose the component locations on the board based on test specifications with the virtually/physically verified analyses, rather than based on user experience. This interface also allows the designer to perform a pass/fail check of the PCB for fatigue damage and obtaining the optimum component locations.

Fig. 1 (a) Internal structure of rear lamp (b) Assembled and illuminated rear lamp
The PCB used in the automobile rear lamp system in the study is given in Figure 1(a) along with the environmental elements. The system can be seen as assembled and illuminated with LED lights in Figure 1(b). PCB in this study shown in Figure 2(a) is used to illuminate the logo in the middle of the assembled rear lamp. PCB layers are formed by sandwiching insulating substrate layer (FR4) between conductive copper layers and this structure shown in Figure 2(b). Additionally, the copper layer was covered with a solder mask to protect it from external factors such as dust or small amount of damage.

II. MATERIAL & METHODS

In the study, uniaxial tensile test was performed on FR4 samples in order to obtain the anisotropic properties of the FR4 material (used in the PCB core layer) and define it in the FE model. Experimental modal analysis (EMA) was performed on 160 x 400 mm plates and the modal shape accuracy was determined with modal assurance criterion (MAC). The modes and the shapes of FR4 plates were obtained using the FE model under the same conditions. Both EMA and FEM results were evaluated and FR4 material properties were validated.

The FE model of the PCB was created in Sherlock software. The types, locations, and materials of the components on the PCB, as well as the connection leads were included in the model. It is very important in automotive lighting for PCBs to include pre-stresses in the model when LEDs running condition. The temperature distribution was obtained by using a thermal camera and mapped to the PCB model. Finite element analysis was carried out using the sine sweep loading condition for the PCB model while LEDs open conditions. In order to validate the FE model of the PCB, the PCB was tested on a shaker using the same boundary conditions and the same loading conditions, thus the results were compared.

Finally, the design variables and the response functions were determined to create a DoE model of the PCB. The samples of DoE were solved and the effects of the each parameters were investigated. Regression was performed based on the experimental design results and the equations related to the design parameters were obtained for each response function.

The workflow was summarized by a basic flow chart shown below in Figure 3.

A. Obtaining FR-4 Material Properties

The physical properties of some single-crystal substances depend on the crystallographic direction in which measurements are made. For example, the elastic modulus and electrical conductivity may have different values in the [100] and [010] directions. The property of variability in these directions is called anisotropy and is related to the variance of the atomic spacing in the crystallographic direction. Substances in which the measured properties are independent of the direction of measurement are isotropic. The core material used in the circuit board (FR4) is considered anisotropic due to its structure. Elastic properties in different directions need to be defined separately to using it effectively in the finite element model [18].

The output of the tensile test is recorded as force against extension. These force-deformation properties depend on the sample size and defined as:

\[ \sigma = \frac{F}{A_0} \]  

Here, F represents the force (N), \( A_0 \) represents the cross-sectional surface area of the sample (\( \text{mm}^2 \)). Engineering strain (\( \varepsilon \)) is defined by the equation below.

\[ \varepsilon = \frac{l_1 - l_0}{l_0} = \frac{\Delta l}{l_0} \]  

Here, \( l_0 \) is original length before any load is applied, \( l_1 \) is instantaneous length, \( \Delta l \) is deformation elongation or length change at any moment depending on the original length.

The degree to which a structure deforms or strains depends on the magnitude of the applied stress. For most materials in tension and at relatively low levels of strain, the relationship between stress and strain is proportional to each other and is known as Hooke's law and defined as:

\[ \sigma = E \varepsilon \]
In the study, different samples were prepared in both directions to obtain the elastic modulus of the tensile test samples in X and Y directions. Tensile tests in two different directions were performed and the anisotropic properties of the material obtained. Detailed dimensioning information of 2 mm thick samples were produced according to ISO 527 tensile test standard can be seen in Figure 4(a). For the uniaxial tensile test of FR-4 samples, the 34TM-50 model testing machine of the INSTRON brand, shown in Figure 4(b) was used based on the ISO 527 tensile test specification. The ends of the samples were placed in the machine apparatus and compressed. The strain-stress curve was obtained with the help of touch sensor extensometer. Test were performed with ten samples each of X and Y axes with 2 mm/min speed. The elastic modulus of each sample was calculated by the machine software and its average was defined in the FE software.

Fig.4 (a) ISO 527 Tensile test specimen (b) Uniaxial tensile testing machine

B. Material Property Validation of FR-4 Plate

Many structures are affected by vibration and they need to be investigated and controlled because of the effects of vibration. The dynamic properties of the structures must be measured in order to carry out these studies. In addition, experimental studies that measure the modal properties of structures are called experimental modal analysis (EMA). Modal testing and modal analysis are used to characterize the dynamic behavior of structures as a result of the forces to which they are subjected. EMAs are necessary to understand how structures vibrate, resist to forces and other structural mechanics.

It can be seen in Figure 5 that the force is applied to the structure by an hammer. The amount of impact applied to the structure was measured in the time domain by using the force measuring tip located at the end of the hammer. As the force was applied, the structure exposed to vibration, the response of the structure to the impact is measured in the time domain with the help of accelerometers located on the structure. Measurements performed in the time domain enables the conversion of the impact and the response functions to the frequency domain by using the fast fourier transform (FFT) [19].

As a result, the mode shapes, the natural frequencies and the damping ratios of the structure can be determined with the help of the FFT. The basic systematic of the EMA process is simply evaluated in Figure 5.

Fig.5 EMA Systematics

The structure of the plate tested by EMA was assumed to be time-invariant and linear, also must obey Maxwell’s reciprocity theorem [20]. The base principle of EMA is finding the ratio between observed excitation at one location while applying force at another location as a function of frequency. The ratio is called the frequency response function (FRF) and known as a complex mathematical function. Mathematical definition of a single-input relationship is given below:

\[
\begin{bmatrix}
X_1 \\
X_2 \\
. \\
. \\
X_p \\
\end{bmatrix} = \begin{bmatrix}
H_{1q} \\
H_{2q} \\
. \\
. \\
H_{pq} \\
\end{bmatrix} \ast F_q
\]  

(4)

Xp is a response in the point of p, Fq is excitation in the point of q, and Hpq is FRF between points of p and q. The process can be achieved by selecting two points on the plate and changing the location of the excitation (hammer) and the response (accelerometer) points. That can be expressed as:

\[
H_{pq} = \frac{X_p}{F_q}
\]

(5)

Boundary conditions of the structure can be either free-free or fixed. Also, there might be signal processing errors during the process such as, leakage, filtering, windowing, averaging, and aliasing [21].

In this study, the fixed position accelerometer and the movable hammer EMA was performed on the FR-4 plate. The test setup is seen in Figure 6. First of all, the FR-4 plate was hung with rubber ropes in free boundary conditions. The plate was divided into 55 equal points and an accelerometer was placed on it and the position of the accelerometer was kept constant. The hammer was hit on these points and the FRF graph of each point was obtained with the help of data reading systems. Finally, the results obtained from EMA were examined with the finite element model and the frequency and mode shapes of the plate were compared.
In the finite element model, the FR-4 plate was defined as 160 x 400 mm rectangular plate with the thickness of 2 mm. Second-order brick elements (C3D20) were used in the model and the modal analysis was solved at room temperature (23 °C) with the free boundary conditions. The FE model of the plate can be seen in Figure 7.

FR-4 material properties were obtained as a result of the uniaxial test performed in the study, they were defined in the FE model and given in Table I. The first six resonance modes of the structure were requested as the outputs of the results.

Table I. Material properties of FR-4 plate

<table>
<thead>
<tr>
<th>Property</th>
<th>Symbol</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>ρ</td>
<td>g/cm³</td>
<td>1,9</td>
</tr>
<tr>
<td>Thermal Expansion Coefficient</td>
<td>αx, ay, az</td>
<td>E-05 °C⁻¹</td>
<td>1,4, 1,4, 1,5</td>
</tr>
<tr>
<td>Elastic Modulus</td>
<td>Ex, Ey, Ez</td>
<td>GPa</td>
<td>25,7, 23,9, 22</td>
</tr>
</tbody>
</table>

It is observed that in all systems exposed to vibration, the vibrations slowly weaken and disappear completely over time. This is caused by the vibration dampers in the systems absorbing mechanical energy. These effects, called damping, working against the speed of the system and creating damping forces. General formulation of damped free vibration;

\[
\ddot{x} + 2D\omega_n\dot{x} + \omega_n^2x
\]  

\(D\) in the equation is the damping factor and is shown as in equation below.

\[D = \frac{\delta}{2w_n} = (\frac{1}{\sqrt{2\pi}})w_n\]  

\(\delta\) in the above equation is the damping constant and is given in below.

\[
\delta = (\frac{1}{\sqrt{2\pi}})
\]  

Under the condition of \(D < 1\) (the condition for vibration movements to occur), the solution of the differential equation is as follows;

\[x(t) = A\exp(-D\omega_n t) \cos(\omega_n t - \phi)\]  

Here \(\omega_n\) defined as in the equation below follows and is called damped vibration frequency.

\[
\omega_n = \omega_n\sqrt{1 - D^2}
\]  

Modal verification is an important matrix system used to examine, verify and predict the similarity of mode shapes obtained after modal tests. Modal assurance criterion (MAC) enables the comparison of physical testing with finite element analyzes using computer-aided software and provides information about the accuracy of the test by making a percentage comparison. \(T_0\) define the comparison between two mode shapes, the modal assurance criterion value is expressed as follows \[22\].

\[
MAC(\{\phi_1\},\{\phi_2\}) = \frac{[(\phi_1)^*\dot{\phi_2}]^2}{[(\phi_1)^*\dot{\phi_2}](\phi_1)^*(\phi_2)}
\]  

The modal assurance criterion matrix varies between 0 and 1, and these values indicate the similarity of the mode shapes obtained from finite element analysis and physical testing. The MAC value is close to 0 if these two vectors are linearly independent of each other \[23\].

C. Validation of Printed Circuit Board

In automotive lighting systems, LED lights in the rear lamp create stresses on the PCB due to the operation of electronic components such as transistors and those components conducting heat around them. For this reason, it is important to examine and include these pre-stresses caused by heat into model.

Head lamp or rear lamp lighting systems are exposed to various vibrations and loads while the vehicle on the road. Obtaining road data on the vehicle takes quite a long time and also those load data can be variable. Therefore, sine sweep, or random vibration analysis is generally preferred in the industry to apply equivalent loads in the laboratory.
In this study, the printed circuit board was physically tested using sine scan data on the shaker and with the LED lights on, and it was validated in a virtual environment by creating a FE model. The same load and the same boundary conditions were established in both the test setup and the FE model.

Most solid materials tend to expand when heated and contract when cooled. The expression for the change in length with respect to temperature for a solid material is as follows:

$$\frac{l_f - l_0}{l_0} = \alpha_l(T_f - T_0)$$

(12)

In the above equation, $l_0$ and $l_f$ respectively, represent the initial and final length quantities during the change from temperature $T_0$ to temperature $T_f$. The parameter $\alpha_l$ is called the linear coefficient of thermal expansion. It indicates the extent to which a material expands during heating and is a material property known to have temperature units [(°C)-1 or (°F)-1]. The heating or cooling process affects all dimensions of an object, and as a result, a change in the volume of the object occurs. The volume change of the body due to temperature can be calculated as follows:

$$\frac{\Delta V}{V_0} = \alpha_v \Delta T$$

(13)

In the above equation, $\Delta V$ and $V_0$ are the volume changes of the body and the initial volume, and $\alpha_v$ means the volume coefficient of thermal expansion. For material structures where thermal expansion is isotropic, $\alpha_v$ is approximately 3$\alpha_l$.

Thermal stresses are stresses that accumulate in the body as a result of temperature changes. Understanding the basis and nature of thermal stresses is very important because such stresses can lead to damage or unwanted permanent deformation. If a homogeneous and isotropic solid body is heated evenly and then cooled, there will be no stress because there will be free expansion or contraction in the object. On the other hand, thermal stresses develop in the body if rigid supports restrict the part's movement in the axial direction. The magnitude of these stresses resulting from a temperature change from $T_0$ to $T_f$ is:

$$\sigma = E \times \alpha_l \times (T_f - T_0) = E \times \alpha_l \times \Delta T$$

(14)

In the above equation, $E$ is the modulus of elasticity, and $\alpha_l$ is the coefficient of linear thermal expansion. If the final temperature is greater than the initial temperature ($T_f > T_0$), the stress exerts a compressive effect ($\sigma < 0$) because the rod expansion is limited. If the final temperature is less than the initial temperature ($T_f < T_0$), tensile stress is applied to the object ($\sigma > 0$). Also, the stress required to compress (or stretch) the body is the same stress that allows it to expand (or contract) freely with the change of temperature $T_0 - T_f$ in equation 14 [18].

In this study, FLIR SC 620 thermal camera was used to measure the temperature on the surface of the PCB under LED lights operating condition as shown in Figure 8(a). The PCB was connected to a 13.5 Volt power supply and the LED lights were illuminated for 30 minutes to stabilize the temperature distribution. Then, measurements were made with a thermal camera and the result of the thermal camera measurement was included and mapped on the FE mesh structure as shown in Figure 8(b).

**Fig. 8**  (a) Thermal camera measurement setup (b) PCB Thermal measurement result

In sine sweep analysis, a sinusoidal oscillation is applied to a system (for example, a mechanical component). In sine sweep, the components respond sensitively to resonance frequencies. Resonance can cause damage or destruction of components in PCB or other devices and cause malfunction problems.

Sine sweep testing is a simple method for measuring resonance frequencies. The system to be tested is stimulated with a sinusoidal signal in this method. The method makes it possible to detect the resonances since the sine sweep continuously passes through many different frequencies. Therefore, sine sweep analysis becomes a very important tool for structural analysis tests.

The sine sweep excitation of unit amplitude can be defined using equation below.

$$\ddot{u}(t) = s \times \left[ 2\pi \int_0^t f(t) \, dt \right]$$

(15)

In the equation, $f(t)$ is the instantaneous frequency which depends on the specific sweep type [24].

In this study, sine sweep test was performed using the “Dongling ES-20-320” model shaker test machine shown in
Figure 9(a) to obtain the modal frequencies and the FRF curve of the PCB. The numbers represent parts used during the sine sweep test. In Figure 9(b), number 1 represents the plate which is connected to ground, number 2 represents fixture which connects PCB to the plate, and number 3 represents the PCB.

PCB was fixed to the fixture with a M3 screw between the hole in its center and the fixture. The fixture was connected to the plate with two M4 screws located at the bottom side. The plate and the ground were connected to each other with two M8 screws as seen in Figure 9(b). The test was performed at room temperature and with the LED lights running condition.

A controller accelerometer was placed on the plate to check the continuity of the 1g acceleration value applied to the shaking plate, and an accelerometer was placed to measure the response acceleration amplitudes on the PCB as seen in Figure 9(b). Accelerations corresponding to the swept frequencies were recorded during the test.

The parameters of the sine sweep test are as follows:

- The driven amplitude value of the system was defined as 1g.
- The test start frequency was defined as 5 Hz and the end frequency was defined as 500 Hz.
- The sweep rate was defined as 1 Hz per second and linear, and the test lasted 8 minutes and 15 seconds.

![Figure 9](image.png)

Various finite element software such as Ansys Sherlock and Ansys Mechanical was used to verify the PCB physical test results in a virtual environment. Sherlock is a software that performs static and dynamic FEA for PCBs. Moreover, reliability and strength analyzes of PCBs can be performed. This module is used for structural analysis of the PCBs used in automotive, aerospace, and many other fields. PCBs include many various components, such as transistors, integrated circuits, capacitors, LEDs, resistors, and diodes. Factors such as the large number of components on the PCB and the variety of the locations of components, makes the modeling with classical FE software too difficult. Sherlock module makes the work much easier and saves time by its library containing wide range of the components and wide range of materials for electronic board design [15].

![Fig.10](image.png)

Importing the PCB layout to Sherlock, which includes specific features of the PCB, is a very efficient method, especially for saving time. Based on the workflow chart of Figure 10, the materials of all components were synchronized with the local library by importing the PCB layout.

![Fig.11](image.png)

Component types (transistor, capacitor, diode, resistor etc.), locations of the components, PCB stack up, materials of components and leads (modulus, density, CTE, etc.) were integrated into model by importing the PCB layout. In the model, 0.1 mm sized second order hex (C3D20) elements were used for leads, 0.5 mm sized second order hex (C3D20) elements were used for PCB layers, and 0.5 mm first order hex (C3D8) elements were used for the components.

The PCB model created in the Sherlock module was exported to Ansys Mechanical. Mesh structure, mesh type, material properties of each component and the lead types were also exported. Figure 11(a) shows the PCB used in the study, and Figure 11(b) shows the PCB layout transferred to Sherlock.

Material properties of each component, the leads and the PCB layers were updated by using Sherlock software library and defined in the model and can be seen in Table I, Table II and Table III.
In order to set up the sine sweep test in a virtual environment, the fixture and the test plate of the PCB were created in a FE model as seen in Figure 12. The numbers represent parts used during the sine sweep analysis. In Figure 12, number 1 represents the plate which is connected to ground, number 2 represents fixture which connects PCB to the plate, and number 3 represents the PCB.

M3 screw connection between the PCB and the fixture and M4 screw connection between the fixture and the test plate were created and the connections were fixed in all axes.

Since the sine sweep test of the PCB was performed with the LED lights on condition, a certain temperature distribution occurs on the PCB and must be included in the model for the accuracy of the analysis. The thermal distribution obtained by the thermal camera as seen in Figure 8(b) was mapped on the FEA mesh. The temperature distribution on the PCB in Figure 8(b) was included in the model and used as the load condition. In this way, effects of the temperature difference were included in the model.

The components transferred from Sherlock software to Ansys Mechanical come into contact with each other in the model, even there aren’t any contacts between the components. For this reason, bonded contact surfaces were created between all components with a gap tolerance of 0,1 mm.

FEA was performed at room temperature with boundary conditions same as the sine sweep test. In order to perform sine sweep analysis, 495 solution intervals were taken between 5 Hz and 500 Hz. The system was excited by 1g acceleration in the +Z direction, and a damping ratio of 0,003 was used which obtained by EMA test. Linear sine sweep type was applied and the amplitude and the resonance frequencies in the Z direction were taken as a response from the accelerometer in Figure 12.

Element sizes and the types of each component used in the FE model were given in Table IV. Since leads are examined in the study, especially small element sizes and second-order elements were used.

### Table IV. Material properties of leads and copper layers

<table>
<thead>
<tr>
<th>Component</th>
<th>Element Size (mm)</th>
<th>Element Number</th>
<th>Element Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>PCB Layers</td>
<td>0,5</td>
<td>326431</td>
<td>C3D20</td>
</tr>
<tr>
<td>Fixture</td>
<td>2</td>
<td>325651</td>
<td>C3D10</td>
</tr>
<tr>
<td>Test Plate</td>
<td>5</td>
<td>10858</td>
<td>C3D20</td>
</tr>
<tr>
<td>Leads</td>
<td>0,1</td>
<td>23025</td>
<td>C3D20</td>
</tr>
</tbody>
</table>

#### D. Design of Experiment (DoE) Model Creation of PCB

Sensitivity analysis is defined as the study of how uncertainty in the output of a model can be attributed to different sources of uncertainty in the model input [25]. There are various types of sensitivity analyses in the literature: "variance-based sensitivity analysis", "polynomial-based sensitivity analysis", and "metamodel of optimal prognosis". An independent data samples should be used to evaluate the prognostic quality of the metamodel. Therefore, selected metamodel with the best prognosis quality is called the metamodel of optimized prognosis (MoP) [26].

One of the best small sampling methods developed is the advanced latin hypercube sampling method. Latin hypercube sampling builds a highly dependent joint probability density function for the random variables in the problem, which allows very high accuracy in the response parameters by using only a small number of samples. In addition, latin hypercube sampling is an expert on simulating correlated or uncorrelated input variables with any probability distributions [15].

In this study, DoE was created to examine the position-dependent changes of PCB components. The positions of each PCB components were determined as the design variable. According to literature studies [6], [7], [8], [9] it was seen that the leads connecting electronic component to the copper layer are the most critical part under the load. For this reason, the 3σ maximum stress at the leads obtained from random vibration analysis and it was chosen as the response function.
As seen in Figure 13, six different regions were defined where locations of three types of components (transistor, diode and integrated circuit) are the variables. Range of location for each region was given in Figure 13(a). DoE was accomplished for each component in these regions and the response surfaces were obtained. As a result, the stress and the damage of critical leads of each electronic component shown in Figure 13(b) obtained in these regions.

Random vibrations are vibrations whose amplitude at a given time cannot be precisely predicted. They can be visualized as a combination of many different sinusoidal vibration profiles with different amplitudes, frequencies and phases. Thus, random vibrations simulate the physical operating conditions more realistically and, thanks to their rich frequency content, can simultaneously stimulate multiple resonance frequencies in a structure, unlike sinusoidal or single-frequency excitation that provide more precise structural diagnosis.

Head or rear lighting systems are exposed to various vibrations and loads while the vehicle on the road. Although the collection of loads on the vehicle takes very long time, random vibration analysis that is creating equivalent damage in a shorter time period is generally preferred in the industry. Random signals give an overview of the severity of vibration. However, in order to understand the effects of the vibration on the structure, the frequency spectrum of this vibration must be determined. These spectrums are obtained using the power spectral density (PSD), which is an ideal tool for identifying the random vibrations.

PSD are expressed by converting the time-dependent signal into the frequency domain with the FFT method and taking its square, and its unit is expressed as the frequency-dependent square of the measured output. If the measured unit is acceleration, the PSD unit is "G²/Hz" and is shown by equation 16. In equation, N is the number of samples, S is the sampling rate, and δf is the resolution of the PSD data [27].

\[ \text{PSD amplitude} = (\text{FFT amplitude})^2 \times \frac{1}{\delta f}, \quad \delta f = \frac{S}{N} \]  

Random vibration analysis was used in the experimental design of this study. It was observed in the literature [28] that, analyzes performed on the axis perpendicular to the PCB (Z-axis) are more challenging. In this study, PSD data, which is used specifically for PCBs in the industry, was used in DoE analyzes on the Z axis and can be seen in Figure 14.

![Fig.14  PSD data of DoE analysis](image)

In the study, the three-band technique by using miner’s cumulative damage ratio was used to calculate the fatigue damage accumulated on the critical leads that were determined as the response function in DoE. The first step is to determine the number of stress cycles needed to produce a fatigue failure. When the end of the lead is connected to the other parts of the structure without any fillet, the computed alternating stress has to account for stress concentration effects. The stress concentration factor K can be used in the stress equation or in defining the slope b of the S-N fatigue curve for alternating stresses. The stress concentration should be used only once in either place. The approximate number of stress cycles \( N_1 \) required to produce a fatigue failure in the lead for the 1σ, 2σ and 3σ stresses can be obtained from the following equation:

\[ \sum_{x=1}^{3} x_{\sigma} \times N_1 = N_2 \times \left( \frac{\bar{\sigma}}{S} \right)^b \]  

Miner’s cumulative fatigue damage ratio is based on the idea that every stress cycle uses up part of the fatigue life of a structure, whether the stress cycle is due to sinusoidal vibration, random vibration, thermal cycling, shock or acoustic noise [29]. Miner’s fatigue damage cycle ratio calculation is as follows:

\[ R_i = \sum_{i=1}^{n} \frac{n_i}{N_i} \]  

The PSD profile in the Figure 14 was solved for 28800 seconds (8 hours), and the damage accumulated on the structure was calculated by taking the inverse of the fatigue life calculated from equation 17 for each sigma value. Bilinear S-N curve as low cycle fatigue and high cycle fatigue was used on copper leads and given in the Figure 15 [30].
"Latin Hyper Cube" was used as the sampling method in DoE. For each DoE, 100 samples were created and were solved. In the DoE model, sample solutions were solved with Ansys Mechanical and the results were collected by Ansys Optislang. Regression was performed using polynomial metamodel and the results were investigated with the help of statistical tools.

In the DoE, statistical tools such as coefficient of prognosis (CoP), the response surfaces, the pareto chart, the coefficient of importance (CoI) and the correction coefficient (CC) were used to make sense of the data obtained after regression.

In the study, equations of response functions depending on design parameters were obtained for each component type and region as a result of the regression. These equations calculate 3σ stresses on leads depending on the X and Y location of the components in the random vibration analysis for DoE. Fatigue damage at the leads was calculated by using 3σ stresses, equation 17 and equation 18. All these data were collected in Excel software and an interface was created that allows the user to instantly check whether the leads are safe or not according to the desired locations. If damage occurs on the leads, the locations of the component can be defined as design parameters and fatigue damage as objective, and optimization can be done by using the "Solver" feature with "Evolutionary Algorithm" in Excel.

In the interface, empty boxes were designed where locations could be entered for each region and component types. Location constraints were defined in these boxes within the range for each region. If an out-of-limit value is entered, a pop-up screen appears and gives a warning. Based on the locations of component type and region, the 3σ stress values obtained from the regression are calculated with the formulation and displayed in the interface tab. Fatigue life is calculated in the background and the damage amounts of each desired component are displayed on the interface by using Miner’s rule. If the damage amount is more than 1, the box is shown in red and it means that the leads are not safe. If the damage amount is less than 1, it means the leads are safe and the box is shown in green. Finally, the interface is visualized with an image showing the location of the components along with the component name and the coordinates of the corresponding location.

### III. RESULTS & DISCUSSION

#### A. Verification of FR-4 Material Property

The natural frequency modes and the modal shapes obtained from the EMA test of the FR-4 plate, which was used as the core material in PCB, were compared with the FEA results and the material properties of the FR4 plate were validated. The results obtained from the physical test (EMA) and the virtual analysis (FEA) were given in Table V. It was seen that there was a maximum error of 5% between the EMA and the FEA.

<table>
<thead>
<tr>
<th>Mode 1</th>
<th>Mode 2</th>
<th>Mode 3</th>
<th>Mode 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>EMA</td>
<td>33,5 Hz</td>
<td>98,4 Hz</td>
<td>192 Hz</td>
</tr>
<tr>
<td>FEA</td>
<td>33,3 Hz</td>
<td>93,4 Hz</td>
<td>190 Hz</td>
</tr>
<tr>
<td>Error</td>
<td>0,5 %</td>
<td>5 %</td>
<td>1 %</td>
</tr>
</tbody>
</table>

The first and second mode shapes obtained from EMA and FEA results are given in Figure 16 and Figure 17.

![Fig.16 (a) Mode 1 Shape – EMA (b) Mode 1 Shape – FEA (c) Mode 2 Shape – EMA (d) Mode 2 Shape – FEA](image1)

![Fig.17 (a) Mode 3 Shape – EMA (b) Mode 3 Shape – FEA (c) Mode 4 Shape – EMA (d) Mode 4 Shape – FEA](image2)

The similarity of the modal shapes obtained from the EMA test and the FEA was examined and modal assurance criteria (MAC) was given in Table VI. It is seen that the similarity of the modal shapes of the EMA and the FEA were over 85%, when the Table VI was examined.
Table VI. MAC comparison of the EMA and the FEA for FR-4 Plate

<table>
<thead>
<tr>
<th></th>
<th>FEA Mode 1</th>
<th>FEA Mode 2</th>
<th>FEA Mode 3</th>
<th>FEA Mode 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>EMA</td>
<td>0,995</td>
<td>0,000106</td>
<td>0,259</td>
<td>0,0296</td>
</tr>
<tr>
<td>Mode 2</td>
<td>0,00336</td>
<td><strong>0,963</strong></td>
<td>0,0019</td>
<td>0,000435</td>
</tr>
<tr>
<td>EMA</td>
<td>0,175</td>
<td>0,00474</td>
<td><strong>0,897</strong></td>
<td>0,0165</td>
</tr>
<tr>
<td>Mode 3</td>
<td>0,0288</td>
<td>0,00271</td>
<td>0,0452</td>
<td><strong>0,867</strong></td>
</tr>
</tbody>
</table>

As a result, it was observed that there was a maximum of 5% deviation in the frequency and at least 85% similarity of the mode shapes between the EMA and FEA results of the FR-4 plate.

B. Verification of Printed Circuit Board

Transmissibility-frequency curve was obtained as outputs from both the sine sweep test and the sine sweep FEA. Transmissibility is the ratio between the output and the input amplitude of system. In Figure 18, PCB resonances were given as name of a, b and c. In the figure it was seen that the other peaks with low transmissibility values came from the test plate and the fixture, thus those resonances were eliminated.

Fig.18 Sine sweep test and FEA curve

Comparison of sine sweep test and sine sweep FEA was shown in the Table VII.

<table>
<thead>
<tr>
<th>Modes</th>
<th>Frequency (Hz)</th>
<th>Transmissibility (g/g)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>FEA</td>
<td>Test</td>
</tr>
<tr>
<td>a</td>
<td>51</td>
<td>53</td>
</tr>
<tr>
<td></td>
<td>76,2</td>
<td>80</td>
</tr>
<tr>
<td>c</td>
<td>437</td>
<td>433</td>
</tr>
</tbody>
</table>

As a result, it was found that there was a maximum deviation of 3,8% on a frequency basis between the FEA and the test when the sine sweep results were examined. This amount of deviation was found to be lower than the other literature studies [2], [3], [4] consequently a verified PCB model was obtained by the FEM.

The shapes of the three natural frequency modes obtained from the PCB sine sweep FEA were shown in the Figure 19.

Fig.19 PCB natural frequency modes (a) First Mode (b) Second Mode (c) Third Mode

In the PCB FE model, element mesh with size of 0,5 mm, 0,75 mm, 1 mm and 2 mm were created for mesh sensitivity analysis. The first three natural frequency modes found by the use of each element size were compared and shown in Figure 20. It was seen that the modal frequencies were converged by the use of elements sizes smaller than 0,75 mm when the mesh sensitivity analysis study was examined.

Fig.20 Element size independence of FE model for PCB
C. Result of Design of Experiment (DoE) and Creating User Interface

It has been observed that changing the location of the components on the PCB does not affect the natural frequency of the entire PCB according to DoE. In the Figure 21, it can be seen that there is an interaction between the responses and the design parameters only for the same type of components. The red boxes highlighted as the total effect in Figure 21 also shows the CoP, which is, the accuracy rate of the regression model. The accuracy of the regression performed in the study was found to be 90.7% for the transistor in the second region and over 95% for the others. Two-dimensional response surfaces in different regions for each component are shown in the boxes in the Figure 21.

In the Figure 21, the first letters of the responses indicate the component types (D-Diode, IC-Integrated Circuit, T-Transistor), and the second numbers indicate the region where it is located according to Figure 13. In the design parameters, the first letter indicates the component type, the second number indicates the region where it is located, and the third letter indicates the axes of the locations.

PCB design was created as an example by selecting random locations in the Figure 22. According to this example design, the integrated circuits in the second and third region do not pass the fatigue scenario and the PCB fails. Here, the user can manually change the location of the failed integrated circuits within the limit ranges until it becomes safe.

Another option is to use Excel's "Solver" optimization algorithm by defining the maximum damage as objective and the location in X and Y axes as design parameters. The best locations for all components in other regions can also be found by defining constraints to the locations of each component.

In the example, IC2 maximum damage was chosen as an objective for minimizing the damage and IC3 maximum damage constrained to stay below 1. The X and Y locations of D4, D5, T1, IC2, IC3 and IC6 were defined as design parameters. Evolutionary algorithm was used for optimization. Limit range values were defined as constraints for all design parameters and also constraints between 0 and 1 were defined for all maximum damages to ensure that the PCB remained in the safe zone. As a result of optimization, the best design parameters according to the given objective were obtained and shown in the Figure 23.
The interface optimized the component locations of the PCB design in the example where the PCB was failed, as a result PCB has been made safe by reducing the maximum damages from 25.4 to 0.55 and 2.59 to 0.67. Furthermore, the user has achieved one of the best PCB designs using data verified by testing and finite element analysis.

The predicted optimum model was solved in real-run finite element analysis and compared maximum stresses of each leads in the Table VIII. It was observed that there was a maximum deviation of 9% between the predicted result and the real-run result.

Table VIII. Comparison of maximum stresses between predicted design and real-run

<table>
<thead>
<tr>
<th></th>
<th>D4</th>
<th>D5</th>
<th>T1</th>
<th>IC2</th>
<th>IC3</th>
<th>IC6</th>
</tr>
</thead>
<tbody>
<tr>
<td>Predicted Max Stress</td>
<td>4.68</td>
<td>31.2</td>
<td>5</td>
<td>62</td>
<td>63.6</td>
<td>59</td>
</tr>
<tr>
<td>Real-Run Max Stress</td>
<td>4.35</td>
<td>32.2</td>
<td>4.9</td>
<td>68</td>
<td>62.2</td>
<td>55.4</td>
</tr>
<tr>
<td>Deviation (%)</td>
<td>7</td>
<td>3</td>
<td>3</td>
<td>9</td>
<td>2</td>
<td>6</td>
</tr>
</tbody>
</table>

IV. RESULTS & DISCUSSION

In the study, a finite element model of the PCB with all its components was created and correlated with physical testing by using the dynamic test scenario frequently used in the industry. The locations of the components on the PCB were determined as the design parameters, the stresses and fatigue damage obtained after random vibration analysis as the response functions in the DoE method. As a result, an interface has been created where the PCB designer can choose the component locations on the board based on test specifications with the virtually/physically verified analyses, rather than based on user experience. This interface also allows the designer to perform a pass/fail check of the PCB for fatigue damage and obtaining the optimum component locations. Results were summarized below:

- It was observed that there was a maximum of 5% deviation in the frequency and at least 85% similarity of the mode shapes between the EMA and FEA results of the FR-4 plate.
- It was found that there was a maximum deviation of 3.8% on a frequency basis between the FEA and the physical test when the sine sweep results were examined.
- A high-accuracy user interface was created as a result of DoE and regressions in order to place the components to optimum location on the PCB with the verified physical test and FEA analysis.
- The predicted best design model obtained in the optimization was compared with real-run FEA analysis, and it was seen that there was a maximum error rate of 9%.

This study is promising for the automotive lighting community and for the industries working with PCB automated designs due to the following reasons:

1. There are many design variables on the PCB, the thickness of the composite layers or the types of solder leads. Sensitivity analyses or optimizations of these variables can be performed with low error rates using the method in this study.
2. Optimization of PCB with different sizes or shapes can also be performed using this method.
REFERENCES


[22] Tümer S., T., Uçak kanadında titreşim analizi (M.Sc. Thesis), Manisa Celal Bayar University, Department of Mechanical Engineering, Manisa, Türkiye, 2016.

[23] Altunel F., Modal updating of a helicopter structure using a newly developed correlation improvement technique (M.Sc. Thesis), Middle East Technical University, Department of Mechanical Engineering, Ankara, Türkiye, 2009.


