IJESRT

INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

CAE ANALYSIS OF AUTOMOTIVE LAMP

Vaibhav Wakchaure*, G.V.R Seshagiri Rao
*1,2BM College of Technology, Indore, India

ABSTRACT

This Paper work is focused on the different FE analysis performed for the automotive lamps. Front fog lamp is designed according to the mounting position given in the panel and the light output requirement. In this paper different type of CAE analysis performed to qualify the physical test requirement the objective of this analysis is to find the optimum size design of lamp to qualify the test & output as per customer requirement. In this paper Static, vibration & fatigue analysis performed on the lamp model.

Keywords: FE Analysis, Static, Fatigue

INTRODUCTION

Front fog lamp is designed according to the mounting position given in the panel. The component of the fog lamp reflector & lens & mounting bracket is designed to meet the optical & physical test standard. In this paper the static analysis is performed to reduce mounting bracket displacement. The Vibration and Fatigue analysis is performed to see weather current design is qualify the physical test requirement.

FRONT FOG LAMP

Front fog lamps provide a wide, bar-shaped beam of light with a sharp cutoff at the top, and are generally aimed and mounted low. They may produce white or selective yellow light, and are intended for use at low speed to increase the illumination directed towards the road surface and verges in conditions of poor visibility due to rain, fog, dust or snow. They are sometimes used in place of dipped-beam headlamps, reducing the glare back from fog or falling snow, although the legality varies by jurisdiction of using front fog lamps without low beam headlamps.Moreover, these types of lamps are having very small space to design the component. While design these types of lamps always keep in mind the optical requirement & thermal issue. According to these two requirements as per standard the fog lamp material are design. Most of the time fog lamp meets the optical requirement but for thermal issue should be very specific while selecting the material for it. Normally these types of lamp are very small in size so while designing the lamp every accept we have to keep in mind. Now day's companies are asking such type of lamps with more optical requirement. So while design the lamp we have

optimizes the design so that it will meet the all prospects. Optimize design become very important in lamp design with in small space you have to design all the requirement.

In this paper CAE analysis performed to meet the testing requirement. Lamp design should be withstand the vibration coming from the engine & bumper as well as from the jerks from the roads. In automotive industry now a day's FEA analysis are very famous to design the component to avoid the failure.

WHY ANALYSIS FRONT FOG LAMP

Front fog lamp is mounted on the front bumper of the vehicle. Lamp design should sustain the vibration coming from the engine & shocks from the uneven road on it. To sustain these condition standard is made by the vehicle manufacturer so that in design stage itself these condition considered to avoid the failure of component.

According to these standard the lamp get design & get qualify in development stage itself. To do the physical testing in design stage proto sample required for the test which is more costly. to avoid this type of testing now a day's CAE analysis performed on the lamp design considering all the testing standard so that if any failure happen in the design it will get change to meet the testing requirement.

To avoid the proto sample cost & to sustain the lamp design from the vibration from engine & from road condition CAE analysis performed on the lamp assembly.

OBJECTIVE OF THE PRESENT WORK

The main objective of present study is to perform the Different CAE Analysis as per physical standard mentioned in the DVP to avoid the failure & reduced product launching time as per SOP. In this paper first perform the static analysis by applying the static load. Second the modal analysis to find out the natural frequency. Third Frequency response analysis & Fourth Fatigue analysis as per the Test standard.

LITERATURE REVIEW

K D Wang has suggested novel approach to determine the noise & vibration characteristics. This is a coupled fields finite element based method. It include fluid structure interaction in the model to quantify pressure waves that is generated by the structural vibration. Computational fluid dynamics & finite element techniques are integrated to analytically determine noise vibration characteristics of a system. Mathematical formulation for a simple noise vibration is presented. Mechanics of the fluid structural interaction is explained. A gray-iron steel ring is selected for the study. Firstly, modal analysis is made on the steel ring to determine modes shape and their corresponding frequencies. Secondly the new method is used to analytically determine its noise and vibration characteristics, Including frequencies, deformation, pressure and sound intensity when it is subjected to an impulse. The analytical results are compared with measurement. It can be expanded to become an effective design tool for improving noise & vibration characteristics in part.

Jin yi-min describe the application of MSC product in Beigi Futin Vehicle Co.Ltd and then describe how to finite element methods can be used for analysis and evaluation of minivan body structure. Including Static, dynamic, fatigue, crashworthiness analysis, optimization and sensitivity and so on. In this paper, Author use MSC. Nastran calculate the strength and stiffness in both bending and torsion load case summarized the strength and stiffness evaluation standard for minivan and then done the normal model analysis in order to consider the influence of tyre unbalance and engine idle excite. the calculations and analysis were verified by test

FEA INTRODUCTION

The finite element method (FEM) rapidly grew as the most useful numerical analysis tool for engineers and applied mathematicians because of it natural benefits over prior approaches. The main advantages are that it can be applied to arbitrary shapes in any number of dimensions. The shape can be made of any number of

materials. The material properties can be non-homogeneous (depend on location) and/or anisotropic (depend on direction).

The way that the shape is supported (also called fixtures or restraints) can be quite general, as can the applied sources (forces, pressures, heat flux, etc.).

The FEM provides a standard process for converting governing energy principles or governing differential equations in to a system of matrix equations to be solved for an approximate solution. For linear problems such solutions can be very accurate and quickly obtained.

Having obtained an approximate solution, the FEM provides additional standard procedures for follow up calculations (post-processing), such as determining the integral of the solution, or its derivatives at various points in the shape.

The post-processing also yields impressive colour displays, or graphs, of the solution and its related information. Today, a second post-processing of the recovered derivatives can yield error estimates that show where the study needs improvement. Indeed, adaptive procedures allow automatic corrections and re-solutions to reach a user specified level of accuracy. However, very accurate and pretty solutions of models that are based on errors or incorrect assumptions are still wrong. When the FEM is applied to a specific field of analysis (like stress analysis, thermal analysis, or vibration analysis) it is often referred to as finite element analysis (FEA). FEA is the most common tool for stress and structural analysis.

Various fields of study are often related. For example, distributions of non-uniform temperatures induce non-obvious loading conditions on solid structural members. Thus, it is common to conduct a thermal FEA to obtain temperature results that in turn become input data for a stress FEA. FEA can also receive input data from other tools like motion (kinetics) analysis systems and computation fluid dynamic (CFD) systems.

STEPS IN FINITE ELEMENT METHOD

The basics steps involved in any finite element analysis consist of the following: Pre-processor Phase

Discrete & select element type It involve dividing the body in to a equivalent system of finite element with associated nodes and choosing the most appropriate element type of model most closely the actual physical behaviour. Assume a shape function to represent the physical behaviour of an element .Develop equations for an element. Arrange and assemble the elements to present the entire system. Construct the global

stiffness matrix. Apply boundary conditions, initial conditions and loads Solution Phase.

Solve a set of linear or non-linear algebraic equations simultaneously to obtain nodal results, such as displacement values at different nodes or temperature values at different nodes in a heat transfer system.

Postprocessor Phase:-Obtain other important information including stress values, heat fluxes and so on

Static Analysis

The basic finite element equation to be solved for structures experiencing static loads can be expressed as:

$$Ku = P$$

Where K is the stiffness matrix of the structure (an assemblage of individual element stiffness matrices). The vector u is the displacement vector, and P is the vector of loads applied to the structure. The above equation is the equilibrium of external and internal forces.

The stiffness matrix is singular, unless displacement boundary conditions are applied to fix the rigid body degrees of freedom of the model.

In this model Reflector is coming out from the lamp assembly because of more displacement of a mounting bracket when apply the static load. To overcome this defect the static analysis performed on lamp assembly with the current design & modified design.

Boundary condition with old mounting bracket

Static load 98 N is apply on the lamp mounting bracket on upward & downward direction. The lamp is kept fix six degree of freedom by applying the RBE2 & SPC on the bracket as shown in below fig.

Fig. FE Model

Results

Results showing the on mounting bracket   maximum displacement 8.45 mm observed as shown in the below fig. Because of this disbarment reflector is coming out from the mounting bracket.
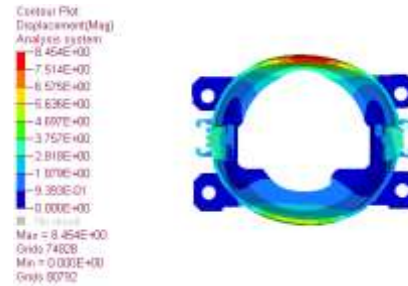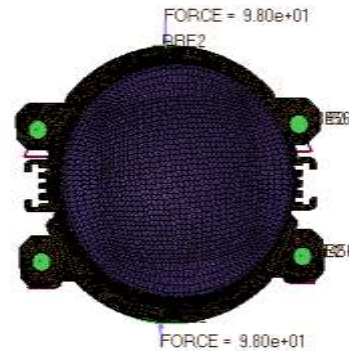

Fig. Maximum Displacement on Old Mounting Bracket

Boundary Condition With New Mounting Bracket

Static load 98 N is apply on the lamp mounting bracket on upward & downward direction. The lamp is kept fix six degree of freedom by applying the RBE2 & SPC on the bracket as shown in below fig

.

Fig. FE Model

Results

Results showing the maximum displacement observed on the mounting bracket   1.7 mm.
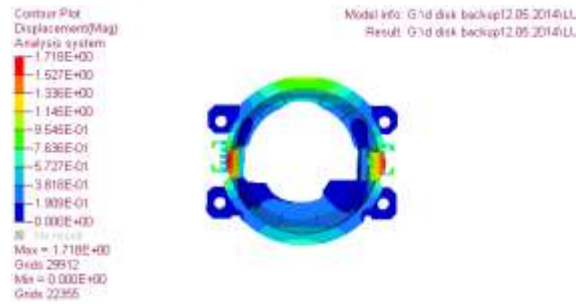
Fig. Maximum Displacement on New Mounting Bracket

If we compare both the results new mounting bracket shows less displacement than old mounting bracket. Because of the less displacement it get tighter to reflector and lens assembly. This is how by using the static analysis the defect remove in design stage only.

Modal Analysis

A modal analysis calculates the frequency modes or natural frequencies of a given system, but not necessary its full time history response to a given input. The natural frequency of a system is dependent only on the stiffness of the structure and mass which participate with the structure (including self-weight) and the boundary condition.
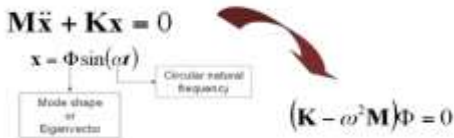
Modal analysis is performed on the lamp assembly to find out the eigenvalue & eigenvector to avoid the resonance condition.

Boundary condition for Modal analysis

The lamp kept as it going to mount in the vehicle. Mounting bracket four points fix six degree of freedom by creating RBE2 & SPC element.

Fig. FE Model

Results

First mode of the lamp is coming 206.17 Hz. This modal analysis results is helpful while doing the fatigue analysis. To avoid the resonance condition this type of analysis is necessary. Modal analysis results are also useful to decide the constraint points.
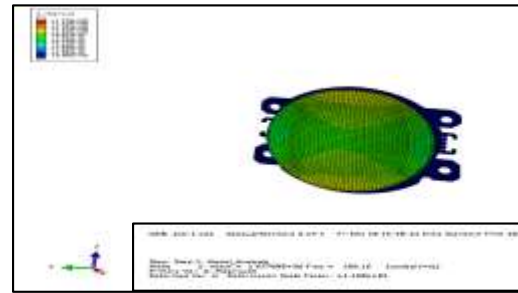

Fig. Modal Analysis First Mode Results

Frequency Response Analysis

Frequency response analysis is used to calculate the response of a structure about steady state oscillatory excitation. Typical applications are noise, vibration and harshness analysis of vehicles, rotating machinery, and transmissions. The analysis is to compute the response of the structure, which is actually transient, in a static frequency domain.

The loading is sinusoidal. A simple case is a load that has an amplitude at a specified frequency. The response occurs at the same frequency, and damping would lead to a phase shift. The loads can be forces, displacements, velocity, and acceleration. They are dependent on the excitation frequency Ω.

The results from a frequency response analysis are displacements, velocities, accelerations, forces, stresses, and strains. The responses are usually complex numbers that are either given as magnitude and phase angle or as real and imaginary part. The direct and modal frequency response are implemented. In this case the FRA is performed on the lamp assembly to see the component response on the applied load.

Boundary condition with old mounting bracket

The FE model is created & constraint points is define as shown in the fig.

The test is to be performed on the following test standard.

Displacement = 1.5 mm
Frequency Range = 10 to 55 Hz.
Direction = X direction

Fig. FE Model

Results

Results showing the maximum stress on the lamp Reflector as show in in fig. The maximum stress is observed 4.31e-03. This stress results is useful to perform the fatigue analysis.
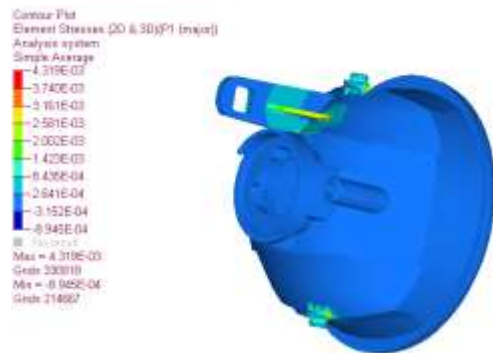

Fig. Maximum Principle stress plot on lamp Assy.

Fatigue Analysis

Often machine members subjected to such repeated stressing are found to have failed even when the actual maximum stresses were below the ultimate strength of the material, and quite frequently at stress values even below the yield strength. The most distinguishing characteristics is that the failure had occurred only after the stresses have been repeated a very large number of times. Hence the failure is called *fatigue failure*

A fatigue failure begins with a small crack, the initial crack may be so minute and cannot be detected. The crack usually develops at a point of discontinuity in the material, such as a change in cross section, a keywayorahole.

Onceacrackis

initiated, the stress concentration effect become greater and the crack progresses more rapidly. As the stressed area decreases in size, the stress increase in magnitude until, finally, the remaining area is unable to sustain the load and the component fails suddenly.

A fatigue failure, therefore, is characterized by two distinct regions. The first of these is due to progressive development of the crack, while the second is due to the sudden fracture. The zone of sudden fracture is very similar in appearance to the fracture of a brittle material, such as cast iron, that has failed in tension.

A fatigue failure almost always begins at a local discontinuity such as a notch, crack, or other area of stress concentration. When the stress at the discontinuity exceeds the elastic limit, plastic stain occurs. For fatigue fracture to occur, there must exist cyclic plastic strains. Thus we shall need to understand the behaviour of materials subjected to cyclic plastic deformation.

Crack Initiation analysis (S-N)

Total Life analysis (S-N) based on the nominal stress-life method using rain flow cycle counting and Palmgren-Miner linear damage summation. Various analysis parameters may be chosen such as mean stress correction methods and confidence parameters. Both component and material S-N curves may be accessed. Material S-N curves allow for specification of material surface finish and treatment.

Crack Initiation analysis (ε-N)

The local strain method using cyclic stress-strain modelling and Number elastic-plastic correction. The mean stress correction method, surface finish and treatment Factors may be adjusted to investigate the effect of these fatigue dependent parameters.

Crack Growth

Analysis using linear elastic fracture mechanics (LEFM) and cycle-by-cycle modelling of crack closure due to overloads, the effect of chemical environment, the loading rate and history effects. On-line displays of crack progress report the rate of crack growth, and Post processing menus enable interpolation of results.

Vibration Fatigue

Analysis calculates fatigue lives directly from Power Spectral Density Functions (PSDF or PSD) using the S-N method. This is a very powerful capability when it is not convenient to analyse a structure in the time domain, making it necessary to do a random vibration analysis.

In this paper we select the Vibration fatigue process to do the fatigue analysis. The FE results from the FRA is the input for the fatigue analysis. The main objective of the analysis to find out the component life under applies load.

To see the existing design is going to qualify the physical test the fatigue analysis performed on the lamp component.

Loading Cycle

The loading cycle is need to define the in the fatigue analysis. The FRA performed on the loading the same load cycle is created as shown in fig.

Fig. Loading Cycle

S-N Curve

To perform the Fatigue analysis the Material specification is required. In this case by creating the S-N curve component material is define.
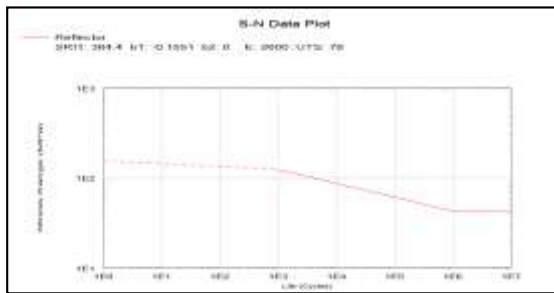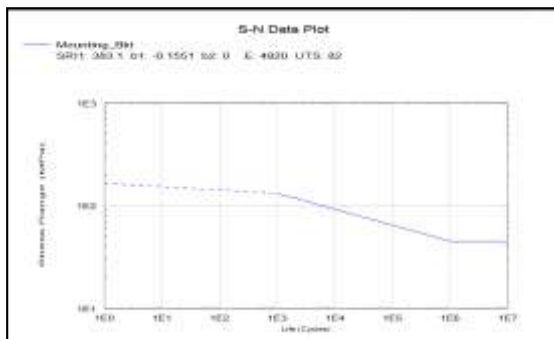

Fig. S-N Curve for Reflector & lens

Fig. S-N Curve for Reflector & lens

Results

Results showing the infinite life for the Reflector Lens & mounting bracket by applied load as shown in fig.

Fig. Life Response Results on Reflector

Fig. Life Response Results on Mounting Bracket

CONCLUSION

Once we have the Fatigue results with us we came to know that existing design of the lamp component is going to sustain the applied load. Once it sustain the applied load then the Physical test is also going to pass.

This is how the CAE analysis performed on the lamp component achieve see the it is going to sustain the physical stage in design stage only.

REFERENCES

1. Chandrupatla, Tirupathi R .and Belegunda, Ashok, Introduction to Finite Element in Engineering, 3Edition, New Jersey, (2002)
2. Wikipedia,"FiniteElementMethod",en.wikipedia.org/wiki/finite_element_method,Nov.2006
3. Wikipedia,"AlexanderHrennikoff",en.wikipedia.org/wiki/alexander_Hrennikoff,Sep.2006
4. Wikipedia,"StructuralAnalysis",en.wikipedia.org/wiki/Structural Analysis, Oct.2006
5. Machine Design II, Prof. K.Gopinath & Prof. M.M.Mayuram, Indian Institute of Technology Madras.
6. http://www.mscnastran.com/2014/
7. Effects of inclusion size and stress ratio on fatigue strength for high-strength steels with

fish-eye mode failure Chengqi Sun, Zhengqiang Lei, Jijia Xie, Youshi Hong, International Journal of Fatigue 48 19–27, ELSEVIER (2013).
8. STANDARD HANDBOOK OF MACHINE DESIGN, Joseph E. Shigley.

Author Biblography

	Mr.Vaibhav Ramnath Wakchaure StudentB.M.Collegeof Technology,Indore RGPV,University Email: Vebs1001@gmail.com