International Journal of Recent Technology and Engineering (IJRTE)
ISSN: 2277-3878 (Online), Volume-2 Issue-1, March 2013

Design Modification for Failed Grill Bracket using Finite Element Analysis

K.S. Kulkarni, R.S.Bindu

Abstract: Grill is a part placed on vehicle located in front of the Engine cooling module by means of bracket. The purpose of the Grill is to protect the cooling module from front impact and at the same time provide appropriate aesthetic value to the vehicle. This report presents the failure analysis of grill bracket of Engine cooling module of a Truck using Finite Element Analysis. The Grill bracket has failed in the field before warranty period so it has to be replaced with new one. Replacing old bracket means economic loss to the company. The Failure of this Grill bracket is analysed using Finite Element Analysis. 3 D models were created using Pro –E CAD softwares and Finite element analysis was done using Medina and Permas softwares. After doing Finite Element Analysis it was observed that high stresses were coming at failure location on the Grill bracket. The high stresses were mainly observed due to accelerations loads. Several proposals for alternate designs were created considering the packaging data, availability of the standard materials and manufacturing feasibility. These alternate designs were again checked by finite element analysis. The most optimized design was finalized through this process. The finalized design showed 60% lower stress values at failure location compared to current design. New proposed design was found to pass the given warranty period (100000 miles). Thus Finite element analysis proved to be very suitable tool for the situation where quick solution is expected.

Index Terms: FEA, Grill

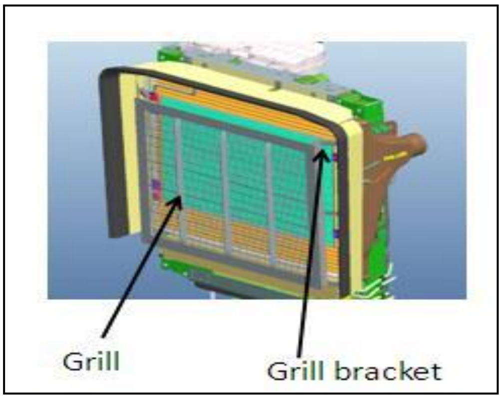
I. INTRODUCTION

The main task of an Engine cooling system is to control the temperature of the engine, the components and the media involved to an optimum level [1].In order to protect the engine from overheating and being destroyed, it is necessary to ensure that sufficient heat is transferred from the engine to the atmosphere. The resultant temperatures in the engine and the fluids have a significant impact on the comfort and environmental friendliness of the vehicle. The customer durability and functional requirement should be achieved to meet vehicle performance is target for Engine cooling system, sub system and components.

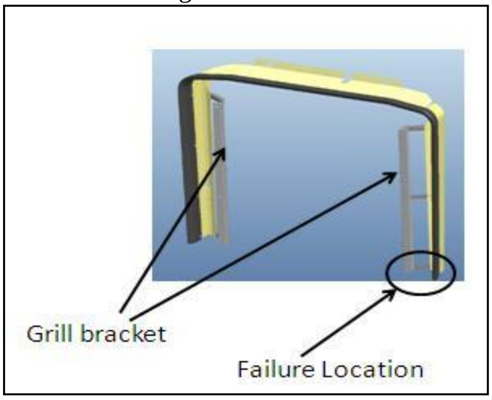
Automotive Engine cooling Subsystem or cooling module is a integrated system which consists of Radiator, Charge air cooler, condenser, oil cooler etc. For maximum efficiency, all components are optimally matched. The modules are put together according to the vehicle design concept, reducing development, production and logistics expenses.

II. PROBLEM STATEMENT

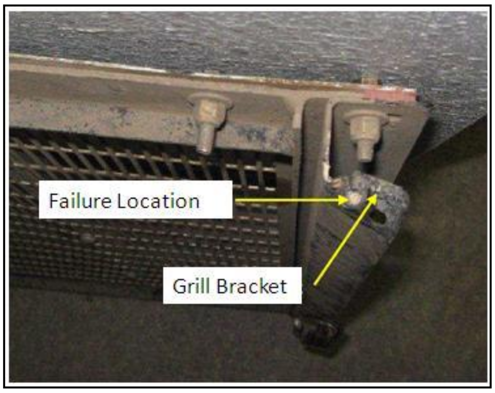
Stationary grill is the part placed on the front side of the Cooling module which protects the cooling module from front side impact. Grill is mounted on the cooling module by means of frames and bracket.



Cooling Module and Grill



Bracket and Recirculation shield



Grill and Failed bracket

Revised Manuscript Received on 30 March 2013.
* Correspondence Author
K. S. Kulkarni*, PG Student, Department of Mechanical Engineering, D Y Patil College of Engineering, Akurdi, Pune (M.H), India.
Dr. (Mrs.)R. S. Bindu,, Associate Professor and HOD, Department of Mechanical Engineering, D Y Patil College of Engineering, Akurdi, Pune (M.H), India.

Above Picture show the assembly of the Grill on the Cooling module. The Grill is mounted on two brackets, one on left and one on right side of the assembly. The Bracket is then connected to the Charge Air cooler tanks by bolted joints.

Many incidents of failure of Grill brackets were observed during running conditions of the vehicle. So a quick failure analysis was required to find out the reason for failure of the bracket and to design a new bracket which will sustain for required warranty period.

The Cooling Module Manufacturing Company is supplying Cooing Module assembly along with the Grill brackets to the Truck Manufacturing Company (OEM). The Grill is supplied by another supplier to the truck manufacturing company. The assembly of the Grill on Cooling Module is done at the OEM location. The Grill bracket was failed in warranty period so it was very urgent to stop these failures. So to stop the failure of the Grill bracket it was proposed to modify the grill bracket design.

J. N. Reddy[2] explained that finite element analysis (FEA) is a powerful engineering tool that can solve many kinds of engineering problems to as high a degree of precision as necessary. In essence, the finite element is a mathematical method for solving ordinary & partial differential equations. Because it is a numerical method, it has the ability to solve complex problems that can be represented in differential equation form. As these types of equations occur naturally in virtually all fields of the physical sciences, the applications of the finite element method are limitless as regards the solution of practical design problems Mechanical, fluid, heat transfer, electrical, and magnetic problems either separately or in combination can be solved by FEA. If a problem can be described by a set of differential equations, it can generally be solved by FEA.

Different Steps in FEA are –

1) Pre-Processing – In Pre Processing the problem is divided in many small components called elements which connected to each other by nodes. These elements are defined by means of shapes functions which represent its physical behavior. The elements thus form are assembled to form a global stiffness matrix. Then Boundary conditions and loading are applied to assembly of the elements (FE model)

2) Solution – Fe model thus created in Pre Processing is solved to the nodal results. The Nodal results can be displacement, stress values etc for the structural problems or temperature, heat fluxed for heat transfer problems.

3) Post Processing – The Results obtained the solution phase can be viewed in post processing phase. Using FEA the current Design can be analysed so that the failure mode can be identified first and then alternate design proposals can be prepared and analysed with help of FEA.

III. INVESTIGATION OF FAILURE OF CURRENT DESIGN

Model Creation - 3D models of the Cooling module and Grill assembly was created in Pro-E Software. The CAD model was then imported in Medina Pre Processing software. In Medina Pre Processor FE Model was created. All different components of the Cooling module wiz Radiator, Charge air Cooler , Surge Tank , Supporting channel and Grill Assembly were created using Solid 3D elements and assembled using rigid 1d Elements. Higher order tetrahedron (tet10) and single order hexahedron (H8) elements were used for solid modeling. Appropriate Material properties for each component were applied and constrains were applied which would simulate the real support conditions for the cooling module.

Loading - The loading for the Analysis was critical. In cooling module the Heat exchangers are undergoing multiple loads at a time. Different loads acting on cooling module are temperature load due to hot fluid, internal pressure of the fluids and acceleration load due to dynamics of the vehicle.

The Grill is not exposed to hot pressurized fluid. So the Grill bracket failure is due to acceleration loads only. It was decided to do the Finite Element Analysis with Acceleration loads. The Acceleration load values are decided based on the test bench setup values. The Test bench acceleration values are calculated using road load test data, complied for different road conditions. Actual loads coming on the vehicle are dynamic loads but for the Finite Element analysis Static Acceleration loads are considered.

Why Static Analysis is done even the loads are Dynamic in nature? Reasons for this question are given below.

1) Static Analysis is quick and easy as compare to Dynamic analysis. There are several variants to be analyzed, baseline and alternate design proposals. To do Dynamic analyses for several variants would take much time as compared to static analysis. This is relative comparison analysis as we are interested in relative improvement in the bracket stress values. The static Analysis would be best to do relative comparison as it would give results comparison faster than Dynamic analysis. So considering the solution time Static analysis is preferred to Dynamic analysis for this situation.
2) Company has internal rules for the Static acceleration analysis for which the loads are calculated based on dynamic loads from road load data. The Finite Element analysis done using Static Acceleration loads are validated for the test bench results. By Validating the FEA results with the test bench results, company has calculated the limiting stress values which are used acceptable stress values for the Static acceleration load analysis. These static acceleration values are applied in X, Y and Z direction. The Stress value are calculated for each of the above load values.

Design Acceptance Criteria –

It is possible to calculate different types of stresses wiz, Von Misses stresses, Principal stresses and shear stresses in post processing software easily. For this analysis Von Misses stresses are considered as these stresses are recommended for ductile material [3].

The Von Mises Criteria states that failure of a component occurres when the energy of distortion reaches the same energy for yield/failure in uniaxial tension test. Mathematically, this is expressed as,

$$\sigma_{von} = \sqrt{1/2 \{ (\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2 \}}$$

where

σ_{von} = Von Misses Stress.

$\sigma_1, \sigma_2, \sigma_3$= Principal Stresses.

According to Von Mises Criteria the maximum Von Mises stress should be less than the material yield strength for the durability of the component. Grill Bracket is made of Standard Steel. The Yield strength for the SS grade 40 is 275MPa. (ASME Specification of Steel – A1011/A 1011M -05a)

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

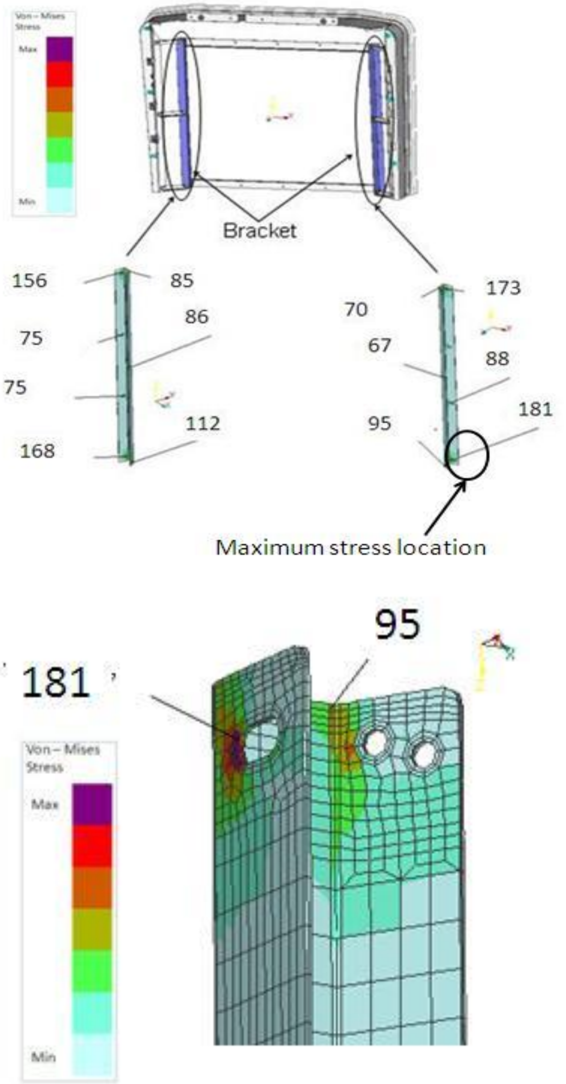
Company has internal allowable stress values for these kinds of components and analysis. These internal allowable stress values are decided based on Finite elements result and test correlation done for several similar components. The Allowable stress limits are not very exact to the standard yield strength but are based on the FE modeling and standard loading conditions used within the company and are valid only for these components.

The stress values listed in this report are relative stress values with respect to the allowable stress values. Maximum allowable stress value in percentage is 100. All stress values greater than 100% are consider to be critical stress values and action need to be taken to reduce these stress values.

Acceleration Analysis Results –

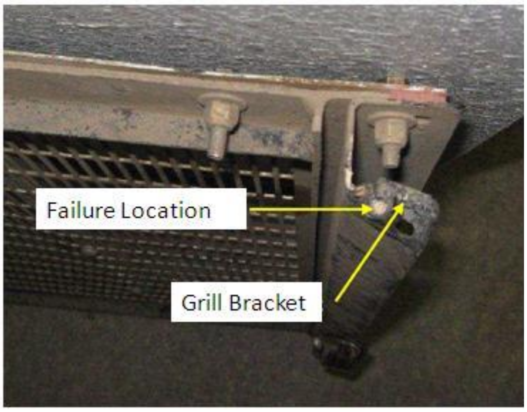
Von Mises Stresses are evaluated at Grill bracket for Accelerations loading. High stress values are observed for the acceleration in X and Y direction which show close resemblance with actual failure location in real part. This is shown in the picture below.

Next picture shows the Von mises stress values on the Grill bracket at different locations due to Load in X direction. Maximum value of 181% stress is can be seen at the right bottom corner.



Grill bracket stress plot on FE model.

Above figure shows close up view for the maximum stress location on Grill bracket. The Maximum stress location is the actual failure location which can be seen in the next figure. These two Figures show that the Finite Element Analysis done for this Grill bracket for current design is able to predict the failure location.



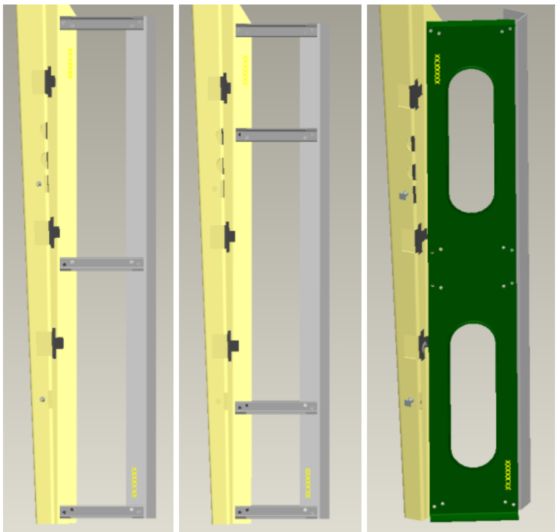
Grill bracket failure on physical part

IV. ALTERNATE DESIGN PROPOSALS

Alternate Designs were proposed and checked by the FE analysis in similar ways as done for Current design.

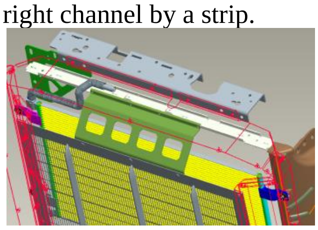
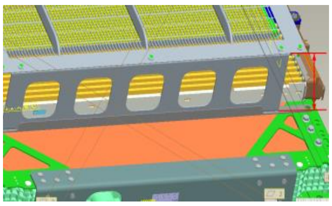
Proposal 1 - Addition of one more mounting bracket with same two rivet pattern to add more strength to grill mounted bracket.

Proposal2 - Replacing the Mounting Bracket by plate. The plate will add more stiffness to the Grill assembly.



Baseline proposal 1 proposal 2

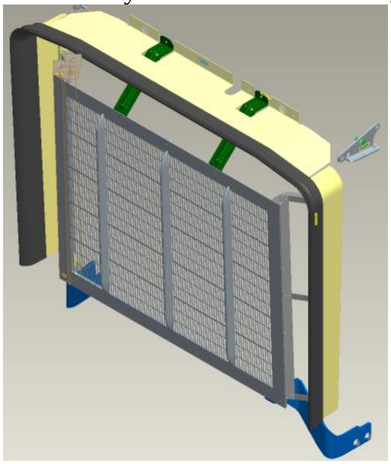
Proposal3 - Adding Extra support from the Top side and Bottom side to the grill assembly. Adding plate on the top side which connect grill assembly and core end plate of charge air cooler. Bottom plate will be connected to left and right channel by a strip.

Proposal 3 top support **Proposal 3 Bottom support**

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

Proposal 4 - Adding top support plates which are connected to top channel and adding bottom support plate with no straight span and directly connected to left and right channel



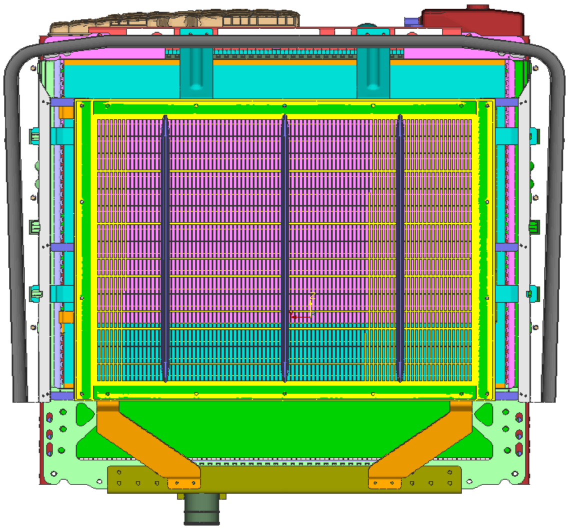
Proposal 4

All Above mentioned proposals for alternate designs were checked by FE analysis. Same Loading and boudary conditions as for Current design are used for analysis so that the results are directly comparable with the baseline.

It was found that Proposal 4 was relatively better than other variants. Stresses on the Grill bracket was reduced by 30% in variant 4 but the stresses on the top support was more than acceptable limit. To reduce the stresses on the top support further modification in alternate design proposal 4 were made.

These changes include changing the thickness of the top support, two bolted connection on each of the top supports and changes in the bottom support to accommodate the other components in the assembly. The modified and finalized design show 60% lower stress on the grill bracket. The Top support and bottom supports were also below the acceptable limit.

The Baseline design and Optimized design were tested along with cooling module for acceleration loads on test bench. The Baseline design failed for but Optimized design passed the lifecycle criteria.



V. SUMMARY AND CONCLUSION

FEA Results of the Current Design are matching with the actual failed component. The location and Stress values found using FEA is showing clear indication that the boundary conditions, Loading, Element types and assumptions made for FE analysis of sufficient to predict the failure for given condition. Out of several design proposals, best suitable Proposal is selected by validating it using FEA. Same Conditions are used for checking Design proposals in FEA as used for Baseline variant. Optimized Design Proposal show 60% reduction in the stress values at Grill bracket failure location when compared to Current Design. Top Support and Bottom Support added are also optimized for stress values. Maximum stress values for top support and bottom support are within allowable range of the Material.

Use of Finite Element analysis was found highly effective in terms of time required to do quick changes in the design and to predict the relative improvement in the successive designs.

REFERENCES

1. C. A. Mesa, 'The Engine Cooling System'.
2. J. N. Reddy, 'An Introduction to finite element methods'
3. Nitin S Gokhle, Sanjay S. Deshpande, Sanjeev V Bedekar, Anand N Thite. 'Practical Finite Element Analysis'
4. J. Devlukia, J. Davies, 'Experiment and Analytical Techniques for Assessing the Durability of Automotive Structures' SAE Paper , 871968, Oct 1987
5. L.C. H. Ricardo, R.J.P.C. Miranda, S. Delijacov, 'Procedures to Apply Finite Element Analysis in Automotive Structures', 2003 SEM Annual Conference & Exposition on Experimental and Applied Mechanics , 2003
6. S. Abdullah, N.A. Al-Asady, A.K. Ariffin and M.M.Rahaman, 'A review of Finite element analysis approaches in Durabilty assessment of Automotive components', Journal of Applied Sciences, 8: 2192-2201, 2008.

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)