Review Study of Analysis of Industrial Brackets using FEA Software

Amol B. Deore¹, Sagar M. Gaykar², Prof. Susheel S. Pote³
¹,²,³Sandip Foundation's Sandip Institute of Engineering and Management, INDIA

ABSTRACT
In Current review study the stress analysis of industrial bracket is considered. It is necessary to ensure the static load carrying capability of the industrial bracket. Stress analysis will be carried out for the given geometry of the attachment bracket. Finite element method is used for the stress analysis. Stress analysis helps in prediction of fatigue life of structural component of bracket structure.[1]

Keywords--- Industrial Brackets, Fatigue life, Finite Element Method

I. INTRODUCTION
Connector type elements widely used as structural supports for connections in bracket structure. Failure of joints may lead to the catastrophic failure of the whole structure. Finite element analysis studies and experimental data help the designer to safeguard the structure from catastrophic failure. Attachment can be some of the most fracture critical components in bracket structure, and the consequences of structural failure can be very severe in case of industrial brackets.[1]

II. FINITE ELEMENT ANALYSIS
The finite element method (FEM) is a numerical technique for solving problems which are described by partial differential equations or can be formulated as functional minimization. A domain of interest is represented as an assembly of finite elements. Approximating functions in finite elements are determined in terms of nodal values of a physical field which is sought. A continuous physical problem is transformed into a discretized finite element problem with unknown nodal values. For a linear problem, a system of linear algebraic equations should be solved. Values inside finite elements can be recovered using nodal values.[1] A fatigue crack normally initiates from the location maximum tensile stress in the structure, further fatigue life estimation can be carried out to predict the life of the airframe component.[1]

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 analyzed with help of FEA. [2]

III. STATIC ANALYSIS
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 analysis 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]
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, we can calculate 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. 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.[2]

IV. CASE STUDY REVIEW

1. In Design Modification for Failed Grill Bracket using Finite Element Analysis, K.S. Kulkarni, R.S. Bindu studies The Failure of this Grill bracket is analysed using Finite Element Analysis. 3D models were created using Pro-E CAD software and Finite element analysis was done using Medina and Permas software. 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. 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 life cycle criteria. 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.[2]

2. Stress Analysis for Wing Attachment Bracket of a six seater Transport Airframe Structure Harish E.R.M, Mahesha K, Sartaj Patel done Stress analysis of the wing fuselage lug attachment bracket is carried out and maximum tensile stress is identified at one of the rivet hole of I-spar plate. FEM approach is followed for the stress analysis of the wing fuselage lug attachment bracket. A validation for FEM approach is carried out by considering a plate with a circular hole. Maximum tensile stress of 1373N/mm
2 (i.e, 140 kg/mm2) is observed in the I-spar plate. Several iterations are carried out to obtain a mesh independent value for the maximum stress. A fatigue crack normally initiates from the location maximum tensile stress in the structure, further fatigue life estimation can be carried out to predict the life of the airframe component.[1]
3. Finite Element Analysis of Cross Member Bracket of Truck Chassis Balbirsingh R. Guron, Dr. D.V. Bhope, Prof. Y. L. Yenarkar says The component bracket in the cross member of chassis is one the most important parts of heavy trucks. Chassis has steel sections, longitudinal ones throughout the length of the vehicle and joined together by transverse parts called the cross members. Rear ones bear payloads and the front one has to bear the engine and transmission. For suspension, handling, body alignment, etc. they need to be strong, besides being complicated, costly and time consuming to replace, if damaged. Stresses induced lead to the failure of the component brackets in the cross members. This study aims to investigate the critical points of stresses that lead to or induce failure. To analyze, finite element method (FEM) is used & commercially available packages MSC/NASTRAN with PATRAN are used for modeling and stress analysis. Modifications have been made to current bracket & these modifications have resulted in reduction in stress values leading to safe design.

V. CONCLUSION

Use of Finite Element analysis is highly effective in terms of time required to do quick changes in the design and to predict the relative improvement in the successive designs. Here we studied the analysis method to solve problems in Industrial brackets. It is the effective method for design and optimization of brackets.

REFERENCES