# FRACTURE ANALYSIS OF EXHAUST MANIFOLD STUD OF MAHINDRA TRACTOR THROUGH FINITE ELEMENT METHOD (FEM) – A PAST REVIEW

MAHESH M. SONEKAR<sup>1\*</sup>, DR. SANTOSH B. JAJU<sup>2</sup> <sup>1\*</sup> P G Student – M.Tech (CAD /CAM)<sup>2</sup> Prof. Mechanical Engineering Department

G H Raisoni College of Engineering & Technology, NAGPUR, (India) - 440016

Email: <sup>1\*</sup> <u>info.cicet@gmail.com</u>

<sup>2</sup> <u>sbjaju@gmail.com</u>

## Abstract

The present paper is a review of the past work done in the field of analysis on fracture of various casting components, engine parts and other area of applications using FEM (ANSYS). Failures were observed even after designing the components with maximum stress value well below yield / ultimate stress. Later it was concluded that the failure is because of variation of load with respect to "time" not being taken in to account. Tests were then carried out for time varying loads. Results proved that the component fails at values below yield stress when subjected to time varying load. It was also observed that below a specific stress value components were not failing at all. This stress value was termed as endurance limit. For example yield stress for general steel is around 250 N/mm<sup>2</sup> and endurance limit 160 N/mm<sup>2</sup>.In general while using FEM technique for failure analysis, a finite element routine would be first used to calculate the static and dynamic displacement and stresses under the maximum compression and tension loading, which were then used for critical points evaluation. Calculation based on failure life and accurate loading histories permit components to be optimized for durability without the need for expensive and time-consuming testing of series of prototypes. According to the various literature reviews on failure analysis, FEA based fatigue analysis of metals (iron, steel, aluminium etc.), and mostly the casted components has been well established. But same is not true with non metals (like polymers etc.). The nonlinear fatigue behavior is not yet completely understood and still in research phase. This paper proposes the use of Finite Element Method (FEM) with ANSYS and ABAQUS as one of the helping software for better result analysis in most of the failure applications.

Keywords: FEM, Fracture, Endurance limit, Optimization, Fatigue failure, Analysis.

# 1. Introduction

Tractor as the most important agricultural machinery has main share in planting, retaining and harvesting operations and then in mechanization sector. Hence in order to reach sustainable agricultural machinery and also its quantity must be reached to optimum level. Very few cases of fatigue failure analysis of agricultural vehicles have been reported till date. In the future study, on failure of exhaust manifold stud of tractor, fatigue and longetivity of the manifold stud will be carried out in the FE code Analysis ANSYS.

Fracture failure of mechanical components is a process of cyclic stress/strain evolution and redistributed in the critical stresses volume. It may be imagined that due to stress concentration the local material yield firstly to redistribute the loading to the surrounding material defects or surface, then follows with cyclic plastic deformation and finally crack initiates and the resistance is lost.

Therefore the simulation for cyclic stress/strain evolution and redistributions are critical for predicting fatigue/ fracture, improving the accuracy of fracture life prediction of mechanical components. For carrying out the fracture analysis it is necessary to determine the maximum load, and for calculating the maximum load various physical parameters would be taken in consideration:

- FE mesh of the component
- Numerical analysis & Validation



Fig. 1.1 Exhaust Manifold Stud under analysis

# 2. Review of past work carried out on Failure Analysis

James R. Dale [1] The paper Connecting Rod Evaluation shows the forging processed components, their entire property during process and investigated the weight and cost reduction opportunities of steel forged connecting rods. Analysis focused on comparing and then optimizing a rod design using crackable forged steel (C-70). Using finite element analysis (FEA) techniques, the author, was able to reduce the weight by 10% and by using "crackable" C-70, reduce the costs by 25% (over current forged steel connecting rods) and ostensibly 15% less than a PF rod with similar or better fatigue behavior.

Table 1.1 Chemical Composition of Powder Forged Materials								
	Cu	С	Mn	S	Fe			
HS150 <sup>TM</sup>	3.06	0.50	0.31	0.12	Bal/			
HS160 <sup>TM</sup>	3.03	0.57	0.33	0.12	Bal.			

The study identifies the fatigue strength as the most significant design factor in the optimization process.

Table 1.2 Taligue Test Result (Connecting Rous, 1 – -2)							
	HS150TM	HS160TM	C-70				
Fatigue Limit @ 90% (MPa)	363	352	283				
Scatter (Mpa)	8	13	48				

Table	1.2 Fatigue	Test Result	(Connecting Rods, $r = -2$	)

The implication of the above data is that PF materials demonstrate improved fatigue strength on the order of 25-33% over C-70 material of the same design. As a result of the lower scatter a more robust and compact product may be obtained through powder forging.

Mansour Rasekh, Md. Reza Asaid, Ali Jafari, Kamran Kheiralipour [2] The work carried on Tractor part (Mf-285) connecting rod, using Finite Element Method to obtain the maximum stress in different sections of connecting rod and proper profile for that was studied for future optimization.

The result shows the maximum pressure stress was between pin end and rod linkage, and between bearing cup and connecting rod linkage. Results of FEM method and results of experimental equations were similar 9Maximum difference was only 13%) which shows the accuracy of the modeling, meshing and loading using FEM. Also the common stresses in carbon steel connecting rods is between 160 to 250 MPa. It can be extract that cause of high fail of forged component is over stresses of common range.



Fig 1.2 Stress distribution in crank end resulted from preloading force considering Van Misses.

**M. Omid, S. S. Mohtasebi, S A Mireei and E. Mahmoodi [3]** In this paper a finite element routine was used to calculate the static and dynamic displacement and stresses under the maximum compression and tension loadings in the connecting rod of universal tractor (U650), which were then used for critical points evaluation. Fatigue analysis and longevity after a 1000000-cycle load, assessed through using ANSYS software.



Fig 1.3 a) Von Misses stresses in compressive loading b) node 46

**Prasanta Sahoo, Biplab Chatterjee, Dipankar Adhikary [4]** Their study considers an elastic-plastic contact analysis of a deformable sphere with a rigid flat using finite element method. The effect of strain hardening on the contact behavior of a non-adhesive frictionless elastic-plastic contact is analyzed using commercial finite element software ANSYS. The result of strain hardening effect shows a generalized solution cannot be applicable for all kind of materials as the effect of strain hardening differently influenced the contact parameters. With the increase in the value of hardening parameter this effect also increases. With the increase in strain hardening the resistance to deformation of a material is increased and the material becomes capable of carrying higher amount of load in a smaller contact area.

**Jesse Doty** [5] As braking forces are applied at the wheel of an automobile, those forces must be transmitted to the suspension of the vehicle. A cliper abutment bracket must take the frictional forces generated by the brake system and not yield or overly deflect. Using ABAQUS a finite element analysis was performed on a caliper abutment bracket. This analysis found the maximum stress of 45Ksi and its location. The maximum deflection at the point of outer pad loading was found to be 0.04887in. in the direction of the load and was compared to hand calculations done using Castigliano's method to find deflection at the same point.



Table 1.2 Comparison of hand calculations and ABAQUS model results. These are deflection values, in inches for the point where outer brake pad force is applied.

Ugine & ALZ [6] The paper deals with the recent progress in the development of thermomechanical fatigue design tools using FEA related to the design of stainless steel exhaust manifold. A numerical method was proposed for the design and the lifetime prediction of stainless steel exhaust manifold under thermal fatigue load. An accurate elastoviscoplastic behavior model has been identified and is available to simulate the stresses and strains in a part submitted to thermal fatigue. The use of a virtual thermomechanical fatigue design approach permits to optimize the design of the manifold and limits both the number of prototype and motor bench tests, and finally, reduces the risk of failure.

Kenneth A. Ramsey [7] This article discusses two popular parts of modern day structural dynamics technology; the experimental portion which is referred to as experimental modal analysis or modal testing, and the analytical portion, which is referred to a Finite Element Analysis (FEA) or Finite Element Modeling (FEM). It discusses how experimental and analytical methods are used to solve design various problems and the importance of using modal parameters to link testing and analysis. Finally, it shows how structural modification techniques are used as a complement to both methods and how all of the tools may be combined on an inexpensive desktop computer. The article concludes with an example showing how experimental modal analysis, structural dynamics modification and finite element analysis were used to analyze the dynamic properties of a test structure.

**W. Kajzer, A. Kajzer, J. Marciniak [8]** The paper presents results of numerical analysis in metatarsal bone 'I'- compression screw system. The aim of the work was determined stresses, strain and displacement in the inserted screws. The analysis was carried out on the metatarsal bone 'I' – compression screws system. The influence of the loads and displacements on the bone – screws system on the results of numerical analysis was analyzed. In order to carry out calculations, 2 model of diverse mechanical properties of screw – Ti-6Al-4V alloy – model 1, stainless steel (Cr-Ni-Mo) – model 2 and two load steps were selected. The obtained results could be used for clinical practice. They can be applied in selection of stabilization methods or rehabilitation as

Fig 1.4 Mesh and contour plot of Von Mises stress for solid model using tetrahedral elements.

well as in describing the biomechanical conditions connected with type of bone fracture obtained from medical imaging.



Fig 1.5 Checking the position of the screw in 4 planes: axial, Lateran, 45° pronation, 45° supination



Fig 1.6 Insertion of the screw & FEM analysis of bone - compression screw made of Ti-64i-4V alloy.

### 3. Conclusion

In this paper we have proposed FEM from a variety of aspects, such as references, features, analytical treatment and technologies. Moreover we have illustrated several representative platforms for the scope of FEM in future applications. In FEM the theoretically calculated results are compared with the measured ones (result validation) and found a good conformity. The result obtained is promising and has the potential as an alternative processing method for failure analysis for different mechanical components. Still there is an essential need for an efficient use of FEM technique with other multiuser software so that accuracy of result could be maintained for better analytical treatment. In order to accurately analyse and optimize the fracture part, there is a need to experiment with multiple fractures parts taking live examples from the companies.

#### 4. Future work

Our review suggests that in forthcoming efforts, analysis of mechanical components (casted components, engine parts etc.,) could be best possible with Finite Element Method with ANSYS or ABAQUS as one of the helping software. In future we explore the failure analysis of exhaust manifold stud and such other components which tends to fail, causes major losses to companies. Our future work is to propose a better analysis technique using FEM with ANSYS for concluding the fracture analysis and design optimization of exhaust manifold stud.

#### 5. References

- [1] James R. Dale "Connecting Rod Evaluation" ASME Journal of Mechanical Engineering January 2005, Vol- 123.
- [2] Mansour Rasekh, Md. Reza Asaid, Ali Jafari, Kamran Kheiralipour "Obtaining maximum stresses in different parts of tractor (Mf-285) connecting rods using Finite Element Method" Australian Journal of Basic and Applied Sciences, 3(2): 1438-1449, 2009., ISSN 1991-8178.
- [3] M. Omid, S. S. Mohtasebi, S A Mireei and E. Mahmoodi "Fatigue analysis of connecting rod of U650 tractor in the Finite Element Code ANSYS" Journal of Applied Sciences 8 (23): 4338-4345, 2008., ISSN 1812-5654.
- [4] Prasanta Sahoo, Biplab Chatterjee, Dipankar Adhikary "Finite Element based Elastic-Plastic Contact Behaviour of a Sphere against a Rigid Flat? Effect of Strain Hardening" International Journal of Engineering & Technology (IJET) 2(1) 2010., Vol 2 Issue 1: ISSN : 0975-4024
- [5] Jesse Doty "FEA Analysis of a Caliper Abutment Bracket" ME 404 March 17<sup>th</sup> 2008.
- [6] Ugine & ALZ "Thermomechanical fatigue analysis of stainless steel exhaust manifolds" Christian SIMON, Pierre Olivier Santacreu Centre de Recherche d'Isbergues, Isbergues, France.
- [7] Kenneth A. Ramsey "Experimental Modal Analysis, Structural Modifications and FEM Analysis on a Desktop Computer" International Journal of Science, Feburary 1983.
- [8] W. Kajzer, A. Kajzer, J. Marciniak "FEM analysis of compression screws used for small bone treatment" Journal of Achievements in Materials and Manufacturing Engineering (AMME)., Vol 33 Issue 2 April 2009.