Harmonic Analysis of Connecting rod using ANSYS

Mr. Sanket S. Thakur[1], Prof. Gajendra Patil [2]
[1, 2] Department of Mechanical Engineering,
[1, 2] PHCET, Rasayani.

Abstract-*Connecting rod is connection between the crankshaft and piston. Major stress induced in connecting rod during its operation is combination of axial and bending stress , bending stress are produced due to centrifugal effects and axial stress are produced because of cylinder gas pressure. The inertia force arising due to reciprocating action of connecting rod causes both tensile as well as compressive forces; therefore durability of this component is of great importance.*

This work deals with development and analysis of connecting rod by alternative method to multi-body dynamics approach.. Harmonic analyses have been carried out. It has been observed that 58.33 Hz is more critical frequency for harmonic analysis which corresponds to 3500 rpm. Same frequency can be tracked by proposed approach and outcomes of Von-mises stress. Maximum principal stress signifies the things more clearly as compared to Von-mises stress as it is compression tension dominant problem.

Keywords-Connecting Rod, FEA, Harmonic Analysis, ANSYS, Frequency Response

I. INTRODUCTION

1.1 Background

The Automobile engine connecting rod is a basic component which is used for high volume production. It is connected between reciprocating piston and rotating crankshaft and transmits the thrust of the reciprocating piston to the rotating crankshaft. Each vehicle that has an internal combustion engine must contain at least single connecting rod that depends on the number of cylinders present in the engine. Connecting rod for car applications are regularly produced by fashioning from either created steel or powdered metal. They could likewise be casted or forged. As it may, castings could have blow-openings which are unfavourable from strength and exhaustion perspectives.

The connecting rod must not reach their primary flexible body natural frequency, but slightly it complete for unknown reasons; they ought to have the capability to survive it sufficiently long for framework to settle down. In the incident that the connecting rod failed, it could twist bringing about the piston to attach on the cylinder wall or fracture bringing about the cylinder wall being gagged or broke from the effect of crushed connecting rod. Both of these defects may bring about costly repair or totally destroy the system, obliging it to be supplanted, consequently, it can see that the connecting rod is key element which should be designed suitably.

For analysis and investigation part can be consider as rigid body or can be changed over into a flexible body by utilizing finite element technique. Analysis of rigid body regards the body as though it can't deform implies that the part won't break. This is all right for common investigation and for small solid parts which are most likely not going to be misshape or break; so far for point by point analysis of stretched thin parts, it immediately outcomes in the arrangement close to that of real arrangement. Because of complex nature of dynamic system, flexible and rigid parts are utilized together commonly to develop the accuracy of the system while in the meantime keeping computational fundamentals to a support to keep the investigation programming from slamming.

Investigation and analysis of flexible body take same standards and principles from that of analysis of rigid body. Simply their required more work for flexible bodies because it contains more number of nodes. An connecting rod could be changed over into flexible body so that the stresses that it come across amid the real run can be investigate to figure out if or not the connecting rod configuration was satisfactory for the system it was to be utilized as a part of.

II. LITERATURE REVIEW

The connecting rod is subjected to a complex cyclic loading during its operational life. It undergoes cyclic loads of the order of 107 to 108 cycles which is very high that is high bending loads due to gas combustion, inertia loads. So, durability of the connecting rod is of great importance. Due to these factors, the connecting rod is the topic of research for different areas such as production technology, materials, performance simulation, fatigue, etc. For the current study work, it was necessary to investigate finite element modelling methods, optimization methods, progress in production technology, new materials technology, fatigue analysis, and manufacturing cost and weight analysis. This brief literature survey reviews some of these aspects.

Mohammad et al. [1] Performed Stress Analysis of Connecting Rod of Nissan Z24 Engine by FEM. Stress analysis of connecting rod has been carried out also kinetic and kinematic analysis of slider-crank mechanism in engine was performed for max power and torque. Analysis of engine was carried out in MSC/ ADAMS software and forces that act on different portions of crank mechanism were found when simulation of connecting rod was done in Solid Works software, ANSYS software was used for meshing and critical loads were applied on it and hence stress analysis was performed. [1]

Ramanpreet Singh [2] investigated Stress analysis of orthotropic and isotropic connecting rod using FEM. The main objective of this research is to develop a new idea for the use of composites material in connecting rods material. Comparison of the conventional isotropic and the orthotropic Composite Material were done using Finite element method. Linear static analysis was obtained for both isotropic material and orthotropic composite material with mesh TET4. Differences between both the material was done by keeping the boundary and load conditions same for both materials. For future research work, the same static analysis can be done with the mesh TET10 and the same can be compared to obtained results is recommended. [2]

Shang-rouhsieh et al [3] add flexible effects for a connecting rod and transverse vibrations analysis was carried out. Various nonlinear resonances and instabilities of mechanism have been obtained. This work can be directly associated with the work of the elastic performance of rigid or flexible four-bar crank mechanism. A proper research of the 4 bar mechanism with big length ratio and low frequency ratio is also recommended. [3]

A. Mirehei et al. [4] investigated the live problem of industry of connecting rod fatigue failure of universal tractor (U650) through the ANSYS software and calculate approximately its lifetime. This research shows the connecting rod performance affected by fatigue phenomenon due to the continuous cyclic loadings and varying speeds, and research results were adopted for saving more cost and time. [4]

Jung Ho Son et al. [5] Predict the Fretting damage of connecting rod of marine diesel engine. Fretting damage occurs frequently at the contacting surface of a connecting rod because the connecting rod in engine is heavily loaded as well as speedily rotated. The actual forces acting on the connecting rod were calculated by dynamic analysis of connecting rod considering flexibility. [5]

The literature survey suggests that the connecting rod fails during its service life due to different loading situations. Generally stresses are developed at fillet section of big and small ends. That's why it is very much crucial to understand the various loads acting on it, analyse the rod for higher speeds, flexibility effects should be taken into account for analysis, acceleration pressure and crank angle for different speeds should be consider for analysis and also the vibrations produced during conduction of thrust. Multi body dynamic behaviour, harmonic and should be carried out.

Although the convenience provided by simulation software’s, still the errors arises because of material characteristics, design and technical problems and inadequate procedures during the analysis are carried out. Small end of connecting rod that is connected to piston get distorted due to continuous cyclic loading at higher speed above 3500 rpm. Outcome of natural frequency from finite element analysis is not explicitly generated.

2.1 Objectives

The objective of the present work is to design and analyses of connecting rod made of graded stainless steel. Observing the different loads act on the connecting rod. Running non-linear harmonic analysis under typical conditions

2.2 Design of Connecting Rod

A connecting rod is a component of engine which is subjected to axial and bending stresses also alternating direct compressive and tensile forces during its operating life. As maximum principal and von mises stress occurs on it so the compressive forces are of much higher values than that of tensile force, so the cross-section of the connecting rod is designed as a strut as a I-section and Rankine formula is used to find its thickness of flange and web section. A connecting rod subjected to an axial load may clasp with x-axis as neutral axis in the plane of movement of the connecting rod or y-axis is a neutral axis. Thus the connecting rod is considered like both ends hinged for clasping about x-axis and both ends fixed for clasping about y-axis. So connecting rod must be strong enough in clasping for both of the axis.

2.3 Material Selection

Depending on the needs of industry various types of materials are used for manufacturing of the connecting rods. The material for a connecting rod is selected based on the use and purpose of the connecting rod and also on the requirement of the internal combustion engines.Material should have high strength and stiffness to withstand the heavy loads in modern engines, and should have opportunities for downsizing, weight and cost reduction. It must resists to fatigue in torsion and bending.It must possess low vibration during high speeds and must have property of resistance to wear in the bearing areas. Materials that are used in the manufacturing of connecting rod are Cast Iron, Aluminium Alloys, Carbon Steel Stainless Steel, Magnesium, and Titanium.

Generally stainless steel materials are used for the connecting rods and taken into account or avoided for the optimization. So a proper picture of the von mises and max principal stress variation during a loading cycle for typical speeds is essential for analysis point of view and this will require finite element analysis over the entire engine cycle

2.4 Methodology

Finite Element Analysis Finite element method is used to analyze structures by computer simulations and therefore it helps to reduce the time required for prototyping and to avoid numerous test series. The modeling and analysis will be done using Finite element Analysis software. Steps for finite element analysis: FEA is mainly divided into three following stages:

1. Preprocessing
2. Creating the model.
3. Defining the element type
4. Defining material properties
5. Meshing
6. Applying loads
7. Applying boundary conditions
8. Harmonic Analysis
9. Review of results

III. DESIGN AND ANALYSIS

3.1 Specifications of Connecting Rod

Table III 1 Specifications of Connecting Rod

Sr. No.	Nomenclature	Dimensions
1.	Length of Connecting rod	155 mm
2.	Crank radius	38.75 mm
3.	Stroke	77.5 mm
4.	Stroke to bore ratio	1.275
5.	Area of cross-section	222.734 mm^2
6.	Total mass of reciprocating parts	1.9258 kg

3.2 Material Properties

Table III 2 Material Properties

Sr.No.	Nomenclature	
1.	Designation	SS-304
2.	Young's modulus	190000 Mpa
3.	Density	7700 Kg/m^3
4.	Tensile Strength	620 Mpa
5.	Compressive Strength	310 Mpa
6.	Poisson's Ratio	0.275

I-section of connecting rod:

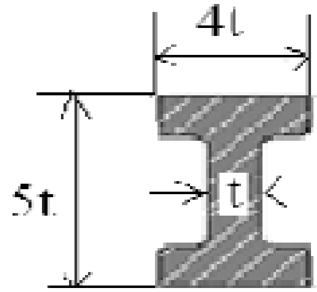


Figure 1I-section of connecting rod

Thickness of flange & web of the section (t) = 4.52mm

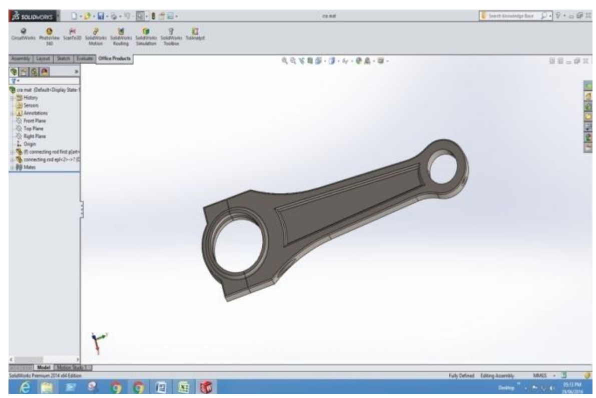


Figure 2 CAD Model of Connecting rod

Results and Discussion

A harmonic analysis is used to determine the response of the structure under a steady-state sinusoidal i.e. harmonic loading at a particular frequency. A harmonic or frequency response considers loading only at single frequency during analysis. Loads may be out-of-phase with one another, but the excitation is at a known frequency. This procedure is not used for an arbitrary transient load. Harmonic analysis is used to find and understand the dynamic characteristics of the model. In this problem we used harmonic analysis because of cyclic load acting on small end of connecting rod.

Results of Harmonic Analysis

At 500 rpm von mises stress and maximum principal stress are founded

1. Von-mises Stress

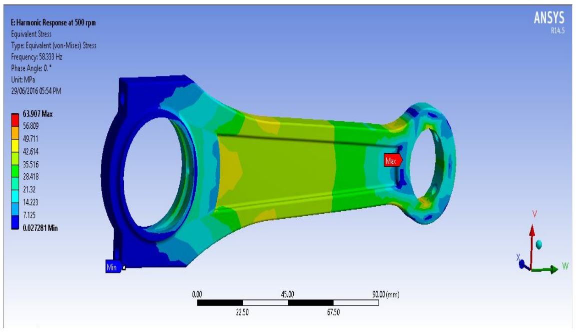


Figure 3 Harmonic Response at 500 rpm for Von Mises Stress

Von Mises stress level of 63.9 MPa suggest that design and analysis is very well safe for safe stress of around 140 MPa with safety factor of around 2.5 which are typical for casted as well as forged steel.

2. Maximum Principal Stress

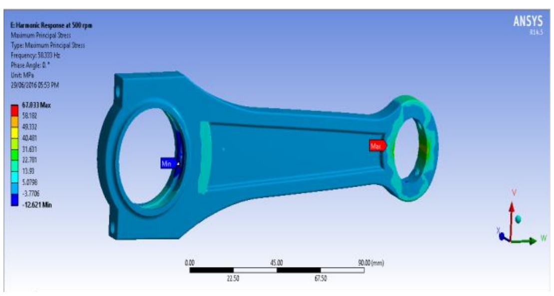


Figure 4Harmonic Response at 500 rpm for Maximum Principal Stress

Max Principal is coming out be more than equivalent stress, which suggests that secondary and tertiary principal stresses are also tensile. Thus failure of connecting rod will be more likely due to sudden tensile loads.

At 1000 rpm von mises stress and maximum principal stress are founded

Von-mises Stress

Situation remains same for higher speeds. Von mises stress is dominant but for most of the portion it is remaining around 40 MPa , which is quite within range as long as material von misses stress limit is to be considered.

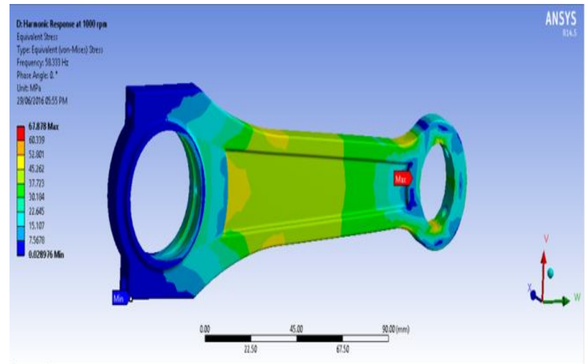


Figure 5 Harmonic Response at 1000 rpm for Von Mises Stress

2. Maximum Principal stress

Maximum principle stress is negative for most of the portion suggesting of moderate compressive loading coming into action at this much speed. But this is not of great concern as it is nearly starting speed of engine. So once transient phase passed away, most of analysis will be steady at higher speeds.

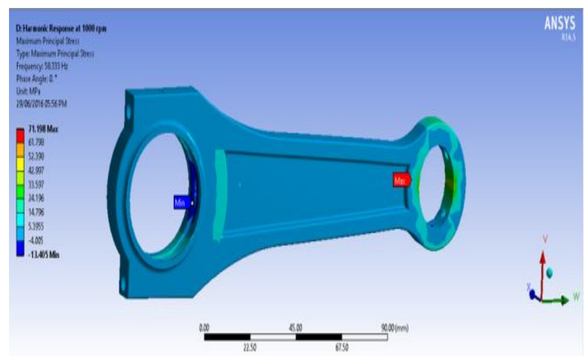


Figure 6 Harmonic Response at 1000 rpm for Maximum Principal Stress

At 2500 rpm von mises stress and maximum principal stress are founded.

1. Von-mises Stress

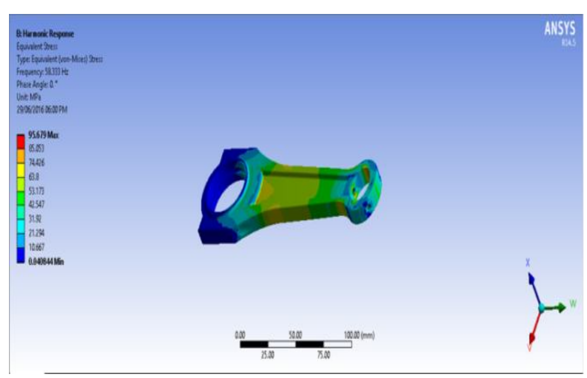


Figure 7Harmonic Response at 2500 rpm for Von Mises Stress

Stressed regions are heavily separated around the circumference but are non-symmetric suggesting that deformations are tighter around these points. So commencements of oval shapes are at odd positions.

2. Maximum Principal stress

Again maximum principle stress has been overshoots to negative maximum. Thus it can be suggested that, maximum principle stress is fluctuating more which is indicating that odd loads and flexibility effects are becoming dominant at some times while at some other time they are remaining inactive.

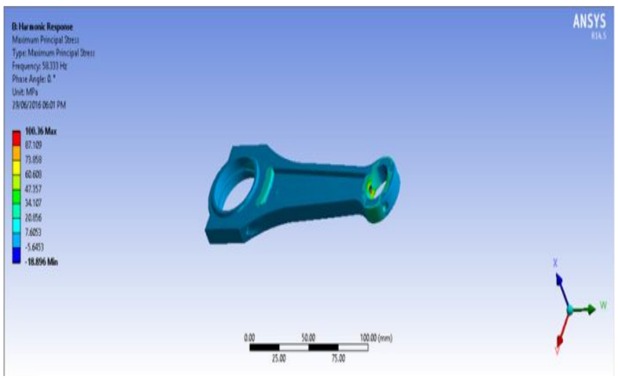


Figure 8Harmonic Response at 2500 rpm for Maximum Principal Stress

At 3500 rpm von mises and maximum principal stress are founded.

1. Von-mises Stress

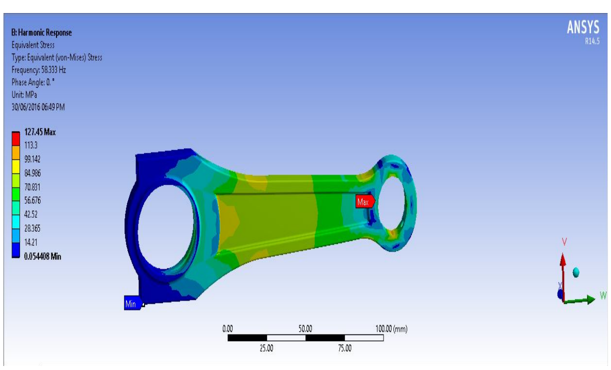


Figure 9 Harmonic Response at 3500 rpm for Von Mises Stress

Further most of things are suggesting that secondary and tertiary principle stresses are of low magnitude may be around zero.

2. Maximum Principal Stress

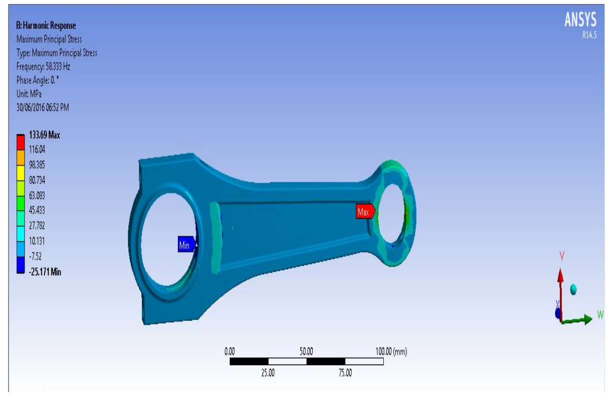
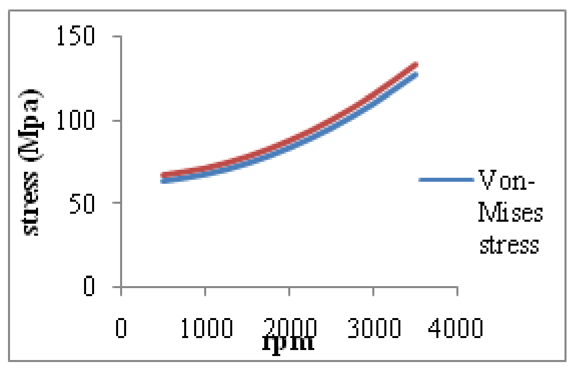


Figure 10 Harmonic Response at 3500 rpm for Maximum Principal Stress

Maximum stress is reported to be inside of small end along length direction which is of tensile in nature .If it exceeds the yield strength; it will cause small end distortions.

It can be seen that Von-mises stress increases with increase in RPM of crankshaft. Secondary effects are suppressed in the fig above. Thus stress is working on the lines of typical accelerations and no damping is changing the physics of problem .Same is story with maximum principle stress as has been shown below.

Graph of von mises and maximum principal stress versus rpm



Graph 1von mises and maximum principal stress vs rpm

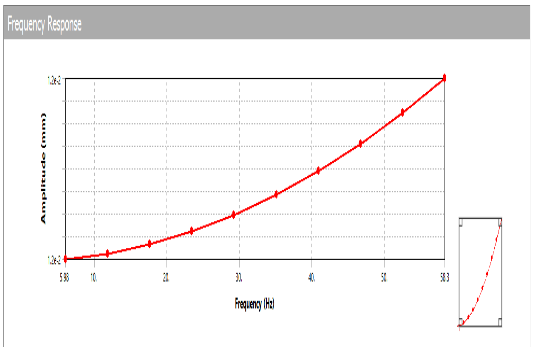
Both Von-mises Stress and Maximum principal stress increases with increase in RPM. Maximum principal stress is higher in magnitude compare to von-mises stress.

Frequency Response

The frequency response was carried out on the connecting rod to establish its resonant frequency. The resonant frequency is the frequency at which any excitation produces an overstated response. It is essential to identify it because excitation close to a system's resonant frequency will frequently produce unfavourable defects. It normally involves extreme vibration which leads to the fatigue failure, it results in break or crack to the mere parts of the body or in severe cases the total structure can be failed.

At 2000 rpm for different frequency range deformation and stress are founded.

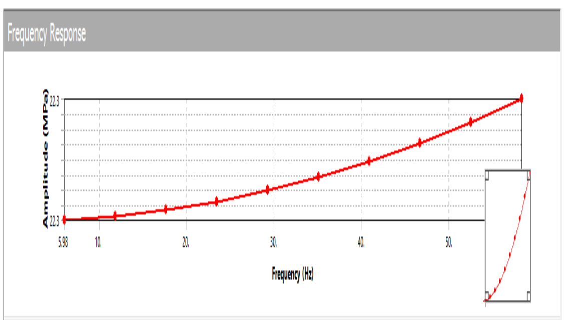
1. Deformation



Graph 2 Deformation at 2000 rpm

Trends for deformations are similar to stress suggesting that Young's moduli are time independent.

2. Stress

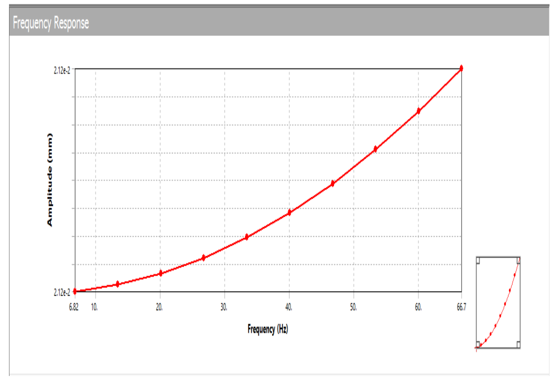


Graph 3 Stress at 2000 rpm

Trends are similar at higher speeds indicating that material orthotropy is not causing more damage as long as frequency response is considered.

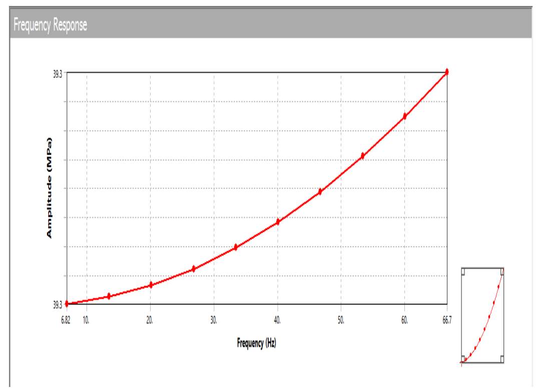
At 4000 rpm for different frequency range deformation and stress are founded.

1. Deformation



Graph 4 Deformation at 4000 rpm

2. Stress



Graph 5 Stress at 4000 rpm

Deformation rise is consistent though. Thus material is behaving such that stress is changing unevenly while deformations are changing consistently. Thus property of connecting rod can be termed as graded. This can be concluded from stress plot shown below as well.

ANSYS results

Table 1 ANSYS Results

Nomenclature	**ANSYS (MPa)**
Normal stress in X direction	128.88
Normal stress in Y direction	43.14
Shear stress at X axis and in Y direction	24.014
Von-mises stress	127.45
Maximum principal stress	133.69

V. CONCLUSION

Harmonic analysis with flexibility effects captures dynamics of connecting rod. Natural frequencies can be eyed at without going into vibration details. Von mises stress levels start from peak goes down at middle, and then comes up. This clearly suggests that stiffness of material is changing continuously due to flexibility effects which are novelty of current work. Company is getting new conservative design with optimised weight of connecting rod, without going into more practical details. Harmonic frequency which is 58.33 Hz corresponds to speed of 3500 rpm. This is clearly suggesting that analysis was restricted up to this speed which is sufficient to capture real time behaviour of connecting rod.

REFERENCES

[1] Mohammad Ranjbarkohan, Mohammad Reza Asadi, and Behnam Nilforooshan Dardashti, "Stress Analysis of Connecting Rod of Nissan Z24 Engine by the Finite Elements Method," vol. 5(12, no. 2084-2089, 2011.

[2] Ramanpreet Singh, "Stress Analysis of Orthotropic And Isotropic Connecting Rod Using Finite Element Analysis," vol. Vol. 2, No. 2, no. ISSN 2278 – 0149, 2013.

[3] SHANG-ROU HSIEH and STEVEN W. SHAW , "The Dynamic Stability and Nonlinear Resonance of a Flexible," 1992.

[4] A. Mirehei, M. HedayatiZadeh A. Jafari, and M. Omid;, "Fatigue analysis of connecting rod of universal tractor through finite element method (ANSYS)," vol. V.4(2), no. 21-27, 2008.

[5] Jung HoSon and Jong GugBae Man Yeong Ha Sung Chan Ahn , "Fretting damage prediction of connecting rod of marine diesel engine," vol. 25 (2), no. 441~447 /content/1738-494x, 2011.

[6] Dongkai Jia , KeWu, Shi Wu, Yuntong Jia, and Chao Liang, "The Structural analysis and Optimization of Diesel Engine Connecting Rod," International Conference on Electronic & Mechanical Engineering and Information Technology, 2011.