Dynamic Analysis of Single Cylinder Petrol Engine

Dr. K.H. Jatkar, Mr. Sunil S. Dhanwe

***Abstract** -In this study a dynamic analysis of single cylinder petrol engine was conducted. Finite element analysis was performed to obtain the variation of the stress magnitude at critical locations of connecting rod and crankshaft. The dynamic analysis resulted in the development of the load on piston. This load is calculated from MATLAB. This load was then applied to the FE model and boundary conditions were applied according to the engine assembly. It is observed that maximum stress is developed at crank pin of crank shaft. The maximum stresses are developed at the fillet section of the big and the small end of connecting rod. Hence, the project deals with the stress analysis of connecting rod and crankshaft by Finite Element Method using ANSYS WORKBENCH 11.0 Software. Also Results obtained from the analysis were then compared with analytical method.*

***Index Terms**—Crankshaft, connecting rod and Ansys workbench 11.0*

I. INTRODUCTION

The Internal combustion petrol engine are those that burn their fuel which is mixture of air and petrol from carburetor inside the cylinder. These engine convert the chemical energy stored in their fuel into heat energy during the power stroke of piston. The heat energy produced from burning of fuel is used for motion of piston. During Operation of four stroke of piston various parts engine is acted by different stress. Also some parts are undergone Deformation. All these stresses and deformation must be studied so that Petrol engine can be designed by Optimum. The magnitudes, variations and exposure times for the stresses and temperatures experienced are the major factors controlling the lives engine components in service. The role of stress analysis is to calculate these quantities so that component service lives can be predicted.

Currently the dynamic analysis of the internal combustion engine for a given configuration and specifications is most widely conducted using a physical test rig. Though the analysis of in-process stress is very much difficult owing to the complications of the system and the temperature of the piston and cylinder assembly..All these process are manually operated therefore itis time consuming.

Dynamic analysis of the IC engine can also be performed using dynamic simulation software like MSC ADAMS, MATLAB Sim-Mechanics, ANSYS and the results of the dynamic analysis can then be imported into a FEA software which can solve for structural solutions like stress, deformation.

II. LITERATURE REVIEW

Work done by various researchers in the areas of defined problem is focused as below

H. D. Desai [1] explained that the reciprocating engine mechanism is often analyzed, since it serves all the demands required for the convenient utilization of natural sources of energy, such as steam, gaseous and liquid fuels, for generation of power. Further, it is widely employed as suitable mechanism for pumps and compressors. In this paper the complete kinematic and combined static and inertia force analysis of a horizontal, single – cylinder, four stroke internal combustion engine is discussed. The analytical approach is used as it is more accurate and is less time consuming if it is programmed for the computer solution. The data for the analysis of the engine has been taken from the available literature. The present investigation furnishes the complete kinematic history of the driven links and the bearing loads for the complete working cycle of the engine mechanism. The complete force analysis of the engine is simplified by a summation of the static forces and inertia forces ignoring the friction forces which makes the analysis linear. The computer program is prepared in fortran language for both kinematic and dynamic analysis of the engine at the crank interval of 15° Vivek. C. Pathade et.al [2] recognized that, the automobile engine connecting rod is a high volume production critical component. Every vehicle that uses an internal combustion engine requires at least one connecting rod .From the viewpoint of functionality, connecting rods must have the highest possible rigidity at the lowest weight. The major stress induced in the connecting rod is a combination of axial and bending stresses in operation. The axial stresses are produced due to cylinder gas pressure (compressive only) and the inertia force arising in account of reciprocating action (both tensile as well as compressive), where as bending stresses are caused due to the centrifugal effects. The result of which is, the maximum stresses are developed at the fillet section of the big and the small end. Hence, the paper deals with the stress analysis of connecting rod by Finite Element Method using Pro/E Wildfire 4.0 and ANSYS WORKBENCH 11.0 Software.

Momin Muhammad Zia Muhammad [3] presented that the crankshaft is an important component of an engine. This paper presents results of strength analysis done on crankshaft of a single cylinder two stroke petrol engine, using PRO/E and

ANSYS software. The three dimensional model of crankshaft was developed in PRO/E and imported to ANSYS for strength analysis. This work includes, in analysis, torsion stress which is generally ignored. A calculation method is used to validate the model. The paper also proposes a design modification in the crankshaft to reduce its mass. The analysis of modified design is also done.

Anil Kumar et.al [4] explained that Aim of this work is to optimize weight and reduce inertia forces on the existing connecting rod, which is obtained by changing such design variables in the existing connecting rod design. The model was developed in Pro/E wildfire 5.0 and then imported as parasolid (xt) form in ANSYS workbench. In this work finite element analysis of the single cylinder four stroke petrol engine connecting rod is considered as case study. The Von Mises stress, strain and total deformation determined for the same loading conditions and compared with the existing results. Based on the observation of static FEA and the load analysis result, the load for the optimization study was selected same as on existing connecting rod. The current work consists of static structural analysis. The static analysis was carried out under axial and buckling load. The model is also selected for fatigue analysis to determine the fatigue strength.

Gunter Knoll et.al [5] studied that A software package was developed to simulate the dynamics of a flexible crankshaft and a flexible engine block, coupled by nonlinear, distributed, elastohydrodynamic fluid film bearing reaction forces. The Kernel of the developed code is a step size controlled integration of Newton's equations of motion in the time domain. Large rigid body motions and small elastic deformations are separated using special matrix operations. No additional set of equations for the rigid body motion is required. Structural properties of the elastic bodies are represented by stiffness and mass matrices obtained from commercial finite element programs. A combined static (Guyan) and modal reduction scheme is used to limit the number of degrees of freedom, at which gyroscopic, centrifugal and coriolis effects are considered by suited *reduced* vectors and matrices. An arbitrary number of fluid film bearings can be defined to couple the finite element structures. Various (elasto-) hydrodynamic calculation models can be selected depending on the needs of accuracy and computational speed. In the elastohydrodynamic solution a sub structuring method is employed and Reynolds' equation, including mass conserving cavitation, is solved on the bearing surface of the substructure using finite element methods. The computed results, nodal displacements, velocities, accelerations and forces, lubricant pressure and density distributions, journal orbits etc., indicate the system behavior and support the system optimization. In an exemple study computed results are compared to measurements. Test rig runs were done with a four cylinder four stroke 1.8 gasoline engine.

Amit Solanki et.al [6] explained that the performance of any automobile largely depends on its size and working in dynamic conditions. The design of the crankshaft considers the dynamic loading and the optimization can lead to a shaft diameter satisfying the requirements of automobile specifications with cost and size effectiveness. The review of existing literature on crankshaft design and optimization is presented. The materials, manufacturing process, failure analysis, design consideration etc. of the crankshaft are reviewed here.

Farzin H. Montazersadghand Ali Fatemi [7] presented that a dynamic simulation was conducted on a crankshaft from a single cylinder four stroke engine. Finite element analysis was performed to obtain the variation of stress magnitude at critical locations. The pressure-volume diagram was used to calculate the load boundary condition in dynamic simulation model, and other simulation inputs were taken from the engine specification chart. The dynamic analysis was done analytically and was verified by simulation in ADAMS which resulted in the load spectrum applied to crank pin bearing. This load was applied to the FE model in ABAQUS, and boundary conditions were applied according to the engine mounting conditions. The analysis was done for different engine speeds and as a result critical engine speed and critical region on the crankshaft were obtained. Stress variation over the engine cycle and the effect of torsional load in the analysis were investigated. Results from FE analysis were verified by strain gages attached to several locations on the crankshaft. Results achieved from aforementioned analysis can be used in fatigue life calculation and optimization of this component.

III. Analytical Vector Approach

The main objective of the analytical analysis is to determine the magnitude and direction of the loads that act on the bearing between connecting rod and crankshaft, which was then used in the FEA over an entire cycle. An analytical approach was used on the basis of a single degree of freedom slider crank mechanism. MATLAB programming was used to solve the resulting equations.

The analytical approach is discussed in detail in this section. The slider-crank mechanism with a single degree of freedom considered for solving the equations of motion is as shown in figure 3.1 below. The following procedure was performed to obtain different dynamic properties of moving components.

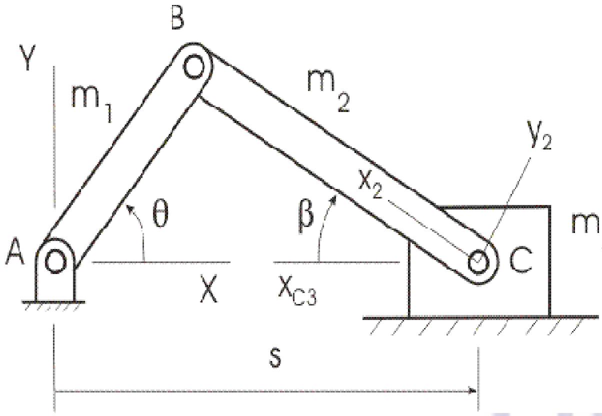


Fig1slider-crank mechanism

The angle θ shown in Figure1 represents the crankshaft angle, which is used as the generalized degree of freedom in the mechanism; therefore every other dynamic property in this mechanism would be a function of this angle. The equations where used in MATLAB and provided the values of angular velocity and angular acceleration of the connecting rod, linear acceleration of center of gravity of the connecting rod, and forces at the connecting rod-piston bearing and connecting rod-crankshaft bearing. The advantage of using MATLAB programming is that any changes in the input could be made very easily and solution quickly obtained, whereas using commercial programs such as Cosmos Motion requires much more time editing the input data. This advantage comes into consideration when optimization is to be performed on a component, since during optimization mass and/or some dimensions change and making these changes in the commercial software is time consuming. The complete MATLAB program used in this analysis is also given

The engine configuration from which the crankshaft was taken is shown in Table I below.. The four link mechanism was then solved by MATLAB programming to obtain the volume of the cylinder as a function of the crank angle.

IV. Analytical analysis of Slider crank mechanism

The analysis was based on simulation of the simple slider-crank mechanism which is shown 3.1 in figure above. As can be seen in the figure 3.1, link AB is the crankshaft

radius, link BC is the connecting rod length, and the slider is the piston assembly. The various parameter of geometry of engine is listed in Table I which shows Configuration of the engine to which the crankshaft belongs.

Table-I

Crankshaft radius	37 mm
Piston diameter	89 mm
Mass of connecting rod	0.283 kg
Mass of piston assembly	0.417 kg
Connecting rod length	120.78 mm
IZZ of connecting rod about CG	0.663 X 10-3 kg/m2
Distance of C.G. of connecting rod from crank end center	28.6 mm
Maximum gas pressure	35 Bar

Some analytical formulae for piston connecting rod and crankshaft used for dynamic analytical analysis of petrol engine.

I. Piston Effort:It is the net force acting on the piston. It is denoted by F_p

$$F_{p}= F_{L}+ F_{I}$$

Where F_L=Net load on piston=Pressure X $\frac{\pi}{4}$ X D^2

F_I=Inertia force of reciprocating parts

II. Forces acting along connecting rod $F_Q = \frac{Fp}{\cos\beta}$

III. Crank pin effort: $F_T= F_Q \times \sin(ø + \beta)$

IV. Torque on the crankshaft: $T= F_T \times r$

Where r =Radius of crank

V. Shear stress of crankshaft: $f_s = \frac{T.16}{\pi D^3}$

Minimum crossectional area of connecting rod is 1500 mm^2.

Using above formulae dynamic analytical analysis of petrol engine can be done. stress on connecting rod and Shear stress of crankshaft can be determined

Table II Shows stress on connecting rod and Shear stress of crankshaft for crank angle 0°, 45°,90 ° up to 360°.

Table II

Crank Angle	Stress in connecting rod (N/mm^2)	Shear stress of crank shaft (N/mm^2)
0	-0.174	0
45	1.160	0.6829
90	1.67	1.109
135	1.814	0.678
180	2.178	0
225	2.299	0.860
270	2.760	1.833
315	6.470	3.809
360	13.926	0

VI. STRESS ANALYSIS BY FEA

In this chapter, Material properties, Meshing Details, Boundary condition and modeling of engine assembly are focused.

Steps of Finite Element analysis of Petrol engine.

I. Defining Element Types: The element library of analysis software contains more than 100 different element types. Each element type has a unique number that defines the element category. I have selected the Element SOLID 186 because I have to analyze 3D model of Engine assembly. Number of element is 867610.

II. Defining the Material Properties: I have selected Structural Steel as a material for 3D solid model of Engine assembly. The value of properties corresponding to Structural steel material like young's modulus , Poisson's ratio, Density or yield strength as shown in table III

Table III Material Properties used for simulation

Material	Structural Steel
Property	Value
Density	7850 kg/m3
Young's Modulus	200 GPa
Poisson's ratio	0.3
Bulk Modulus	166 GPa
Shear Modulus	76 GPa
Tensile yield Strength	250 MPa
Compressive yield strength	250 MPa
Tensile Ultimate strength	460 MPa

III. Geometry (Build the Model): First of all I have made a 3D model of Engine assembly on ANSYS workbench Design modular. In building a model we have used various modeling tools.

IV. Meshing of Model: We discritized the whole solid model into small elements. Depending upon the requirement of the accuracy of results the fineness of meshing varies. More finer is the meshing more we are closer to the actual results. as shown in figure 4.2and 4.3 Number of nodes is 2309632

V. Boundary Conditions: Following boundary conditions are applied on the rectangular flange

VI. Force

VII. Fixed Support

VIII. Cylindrical Support

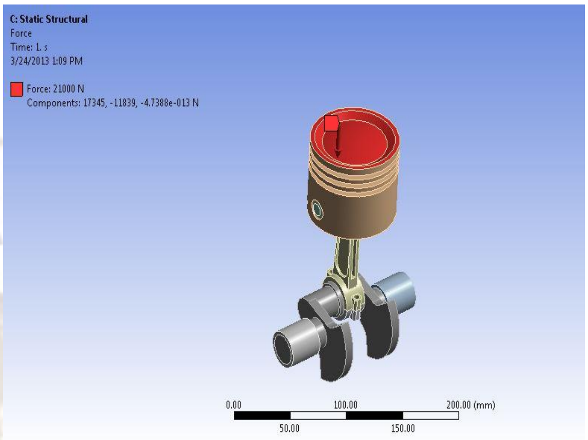
IX. Loading:

From the dynamic analysis carried out in Cosmos Motion & Matlab, the maximum force induced between piston and connecting rod is taken for static analysis in Ansys.

Maximum Force applied from the dynamic analysis: - 21000N i.e. 21kN

Force :A force of 21KN is applied on the top surface of piston in the downward direction as shown in the Figure 2

Fig. 2 Force diagram



VII. RESULTS AND DISCUSSION

In this chapter, analysis of piston assembly, connecting rod and crankshaft for stress and deformation by using Ansys has been done.

i) Deformation of piston assembly: Figure 3 Shows deformation of piston assembly

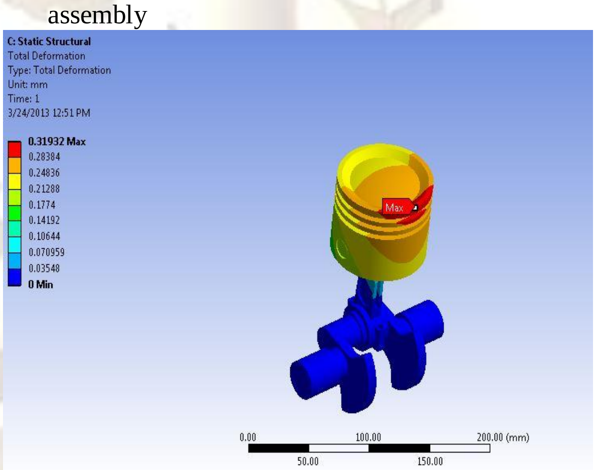


Fig 3 deformation of piston assembly

ii) Stress of piston assembly: Figure 4 Shows Stresses On Piston Cylinder Assembly

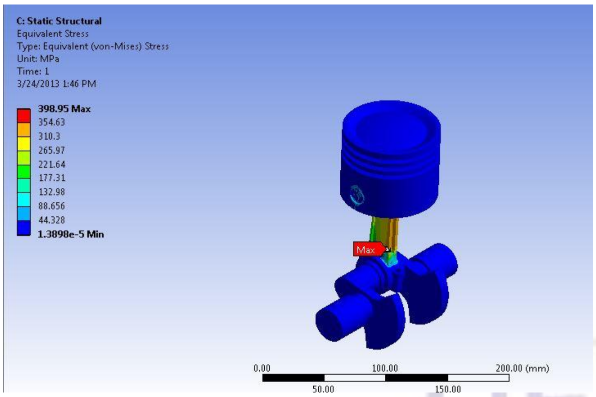


Fig 4 Stresees On Piston Cylinder Assembly

Table IV result of piston cylinder assembly

Results	Maximum	Minimum
STRESS	398.95MPa	1.26e-005 MPa
DEFORMATION	0.31932 mm	0mm

Table IV shows result of piston cylinder assembly

These results values are within the permissible limits for material at working temperature. Maximum stress induced is 391.07MPa

Results	Maximum	Minimum
STRESS	0.0057567	1.4853 e-6
DEFORMATION	1.6142E-6	0

at a location near to Stress concentration area in connecting rod. This value of stress can be minimized by optimizing the connecting rod geometry.

In the rest of the assembly maximum stress induced is 221.64MPa which is within the limit of material.

iii) Analysis of connecting rod

Stress:

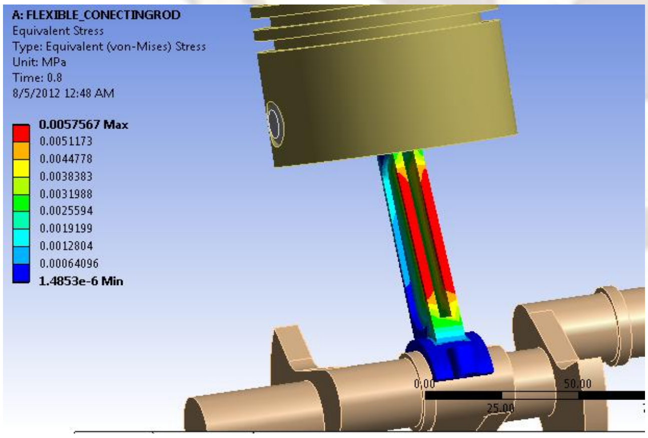


Fig 5. Stresses On Connecting Rod

The above Figure 5 shows Von Mises Stress acting on Connecting Rod. From the above fig. the red portion shows the maximum stress. So it is clear that the maximum stress acting on Connecting Rod is at the fillet section Because of Stress concentration.

Deformation

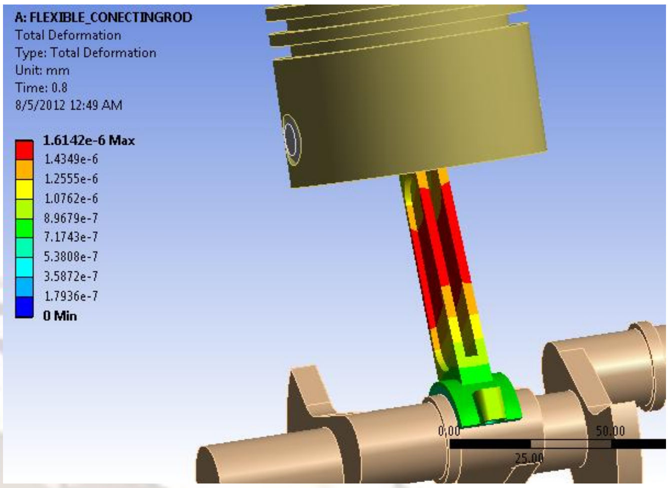


Fig. 6 Deformation Of Connecting Rod

The above fig. 6 shows deformation of the Connecting Rod it is clear that the load imparts a compressive stress to Connecting Rod and causes bending. The red and yellow portion shows the max. Deformation. Table IV shows stress and deformation Of Connecting Rod

Table IV Results Of Connecting Rod

Results	Maximum	Minimum
STRESS	0.57895	0.00071689
DEFORMATION	0.00016631	0

iv) Analysis of Crank Shaft

Stress:

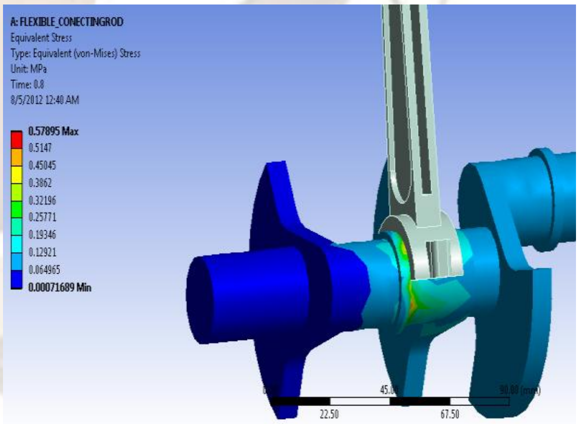


Fig 7 Stresses On Crankshaft

The above Figure 7 shows Von Mises Stress acting on Crankshaft. From the above figure the red portion shows the maximum stress. So it is clear that the maximum stress acting on Crankshaft is at crank pin due to Stress concentration.

Deformation:

The following figure 8 shows deformation of the Crankshaft. it is clear that the load imparts a bending stress to Crankshaft. and causes torsion. The red and yellow portion shows the max. Deformation.

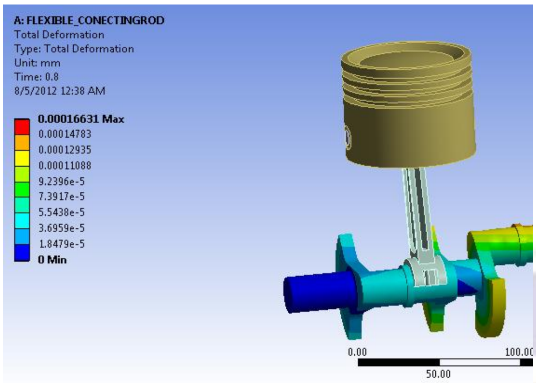


Fig 8 deformation of the Crankshaft

Table V shows maximum and minimum value of stress and deformation of crank shaft

Table V result of crankshaft

VIII. COMPARISON

The analysis results and analytical result are compared here as shown in Table VI and VII

i) Comparison between Analytical Vs Ansys stress for Connecting Rod

Table VI

Crank angle	Ansys stress	Analytical stress
0	-0.180	-0.174
45	1.191	1.160
90	1.75	1.67
135	1.9	1.814
180	2.24	2.178
225	2.30	2.299
270	2.792	2.760
315	6.512	6.470
360	14.12	13.926

ii) Comparison between Analytical Vs Ansys stress for crankshaft

Table VII

Crank angle	Ansys stress	Analytical stress
0	0.105	0
45	0.712	0.6829
90	1.198	1.109
135	0.685	0.678
180	0.045	0
225	0.897	0.860
270	1.916	1.833
315	3.897	3.809
360	0.021	0

From the above table VI and VII it is clear that software results are in good correlation with the analytical results.

IX. CONCLUSION

I. Static analysis of a connecting rod that is typically performed can yield unrealistic stresses, where dynamic analysis provides more accurate results better suited for fatigue design and optimization analysis of this high volume production component.

II. The stresses induced in the small end of the connecting rod are greater than the stresses induced at the big end. Therefore, the chances of failure of the connecting rod may be at fillet section of both ends.

III. The bending stress produced as a result of dynamic loading is significant and bending stiffness in the shank should be considered as an important design factor.

IV. Kinematic analysis of the engine has been carried out by analytical method Using MATLAB method as this method is more accurate than graphical method and can give results for all the phases of the mechanism.

REFERENCES

[1] H. D. Desai "Computer Aided Kinematic and Dynamic Analysis of a Horizontal Slider Crank Mechanism Used For Single-Cylinder Four Stroke Internal Combustion Engine"London, U.K.

[2] Vivek. C. Pathade, BhumeshwarPatle, Ajay N. Ingale 'Stress Analysis of I.C.Engine Connecting Rod by FEM.' Department of mechanical engg, Wardha.

[3] Momin Muhammad Zia Muhammad Idris 'Crankshaft Strength Analysis UsingFinite Element Method' PIIT, New Panvel, India.

[4] Anil kumar, Kamaldeep Grover Balvinder Budania 'Optimization of Connecting Rod Parameters using CAE Tools' ,International Journal of Latest Trends in Engineering and Technology (IJLTET)

[5] Gunter Knoll .Richard Schönen. Klaus Wilhelm, 'Full dynamic analysis of crankshaft and engine block with special respect to elastohydrodynamic bearing coupling' Germany

[6] Amit Solanki, Ketan Tamboli, M.J.Zinjuwadia' Crankshaft Design andOptimization- A Review' National Conference on Recent Trends in Engineering & Technology

[7] Farzin H. Montazersadgh and Ali Fatemi 'Dynamic Load and Stress Analysis of a Crankshaft' The University of Toledo SAE International 2007-01-025

[8]. Pravardhan S. Shenoy, "Dynamic Load Analysis and Optimization of Connecting Rod"Department of Mechanical Engineering, The University of Toledo. A Thesis, May 2004

[9] M. F. Spotts, "Design of Machine Elements", Prentice Hall of India Pvt. Ltd, New delhi,

[10] Reddy J.N. ."An Introduction to Finite Element Method", Tata McGraw-Hill Publication, Fifth Edition.'

[11] V.B. Bhandari, "Design of Machine Elements", Tata McGraw Hill Publishing Co .Ltd, New Delhi, 2005.

[12] Joseph Edward Shigley, Charles R. Mischke*"Mechanical Engineering Design",*McGraw-Hill Book Company, New York, 2000, Sixth Edition.

[13] Ansys User Guide- 2010

AUTHOR BIOGRAPHY

Dr.K.H.Jatkar presently working as HOD in Walchand Institute of technology, Solapur, Mechnical Engg. Dept

Mr. Sunil Shivaji Dhanwe Pursuing M.E. (Design)II from Walchand Institute of technology, Solapur. Also, Working as Sr.Lecturer in Brahmdevdada Mane polytechnic, Solapur.

Membership- I.E.I. Membership No. : - AM0994688