International Journal on Applications in Mechanical and Production Engineering
Volume 2: Issue 3: March 2016, pp 10-13. www.aetsjournal.com ISSN (Online) : 2395-3500

Evaluation and Analysis of static stress on Connecting Rod using CosmosXpress Study on Solid works

D.Balaji , G.Guberan Muthu , E.SathishKumar

Abstract— Now a day's vehicle is must for commute. As Internal combustion engine are the heart of any vehicle, in which connecting rod have the major impact of engine life. In this research paper the static stress subjected by connectedrod made up of Chrome stainless steel for vehicle is analyzed using the special feature called cosmos xpress by Solid works software. By accurate dimensions from the manufacture and actual measurement the model was developed on solid work software.Byfixing the smaller end of connecting rod as rigid one, specific load, which is derived from data available from manufacturer, is applied at bigger end.Post processing work is done by cosmos xpress, such as deformed shape, stress distribution, area below factor of safety is one are derived from the analysis software. Stress at which maximum and point lead to failure are identified and analyzed.

Keywords: AA 6061, CosmosXpress, static Analysis, Analysis of connecting Rod.

I. INTRODUCTION

In automotive engine connecting rod plays a vital role. It links the piston and the crankshaft, conveying the piston thrust to the crankshaft so that the reciprocating signal of piston is converted torotating motion by crankshaft. Material selection is the root for any stress analysis and also for production process and fatigue parameter.

Awidespread literature review was undertaken for the research work. Mohammed et al [1] describe the reason of failure of connecting rod using FEA by using MSC Patran software and performed the study of metallographic buckled connected rod. Pathade et al [2] studied stress analysis at different sections. First connecting rod is modelled in pro/E wildfire 4.0 and analysis is done by ANSYS Workbench 11.0 software. Failure analysis is made up of different material such as carbon steel, brass,mild steel, and aluminum are characterized in the study of Noor et al [3]. By FEM method result has found out verified. Geometric parameters of bigger end of connecting rod is investigated in the study of Chikalthankar et al [4]. Tata motors connecting rod of light commercial vehicle is analyzed in the study of Doshi and Ingole [5]. Designed by machine design approach Connecting rod is analysiedand it is compared with the production drawing. They utilized Pro\E wildfire 4.0 for modeling and ANSYS 11 for analysis at different load. Connecting rod is dynamically analyzed by Ram Bansal [6]. In his paper model of connecting rod of single cylinder four stroke diesel is created in CATIA V5R18by taking the dimensions of the rod and analysis is done on ANSYS. Boundary conditions performed by dynamic simulation.

Knowledge acquired from the above literature review and study, MaximumStress at connecting rod of 885 BC engine is presented in this work. This paper is organized by four sections. The Second section is problem identification, which involvesmodeling of connecting rod in Soliworks by measuring the dimensions, and calculating load applications. Analysis is conducted in third section with help of COSMOSXPRESS and result is analyzed. Conclusion is arrived at last section.

D.Balaji , Assistant Professor, Department of Mechanical Engineering ,Sri RaajaRaajan College of Engineering ,Amaravathipudur, Karaikudi, TN, India (Email: deenabalaji1984@gmail.com)
G.Guberan Muthu , Assistant Professor, Department of Mechanical Engineering ,Sri RaajaRaajan College of Engineering , Amaravathipudur, Karaikudi, TN, India (Email : guberanmuthu07@gmail.com)
E.SathishKumar , Assistant Professor, Department of Mechanical Engineering, Sri RaajaRaajan College of Engineering , Amaravathipudur, Karaikudi, TN, India(Email : es.kumar001@gmail.com)

II. PROBLEM IDENTIFICATION

In connecting rod, bigger end and smaller are the most critical regions for stress failures. The stress for four selected region (crosssection) is calculated and shown on Fig 1. While analysis the load is applied on bigger end of connecting rod and remain fixed the smaller end. Based on engine manufacturer specification, we have selected three different load conditions for our analysis.

Then the required model of connecting rod is created based on the actual dimension using solid works software.

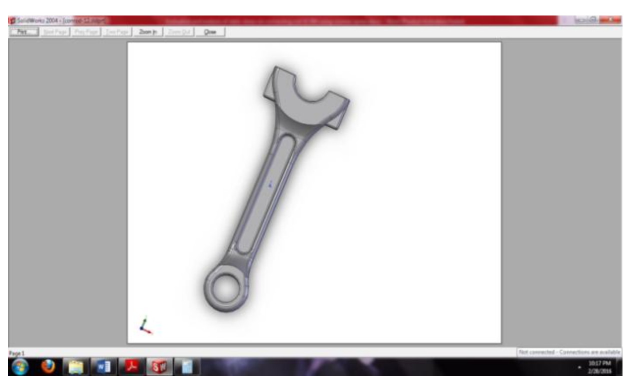


Fig 1. Model of connecting rod in Solid works software.

By using Vernier caliper, steel rule and micro meter the actual dimensions of the connecting rod is measured. The manufacturer specification of the connecting rod is given in Table 1.

Table 1. Specifications of connecting rod and Engine

Engine	4 stroke, 6 cylinder
Cylinder volume	14L
Cylinder Bore	142 x 154 mm
Connecting rod mass with cap	4.4 kg
Connecting rod mass without cap	3.31 kg
power	290 HP @1800rpm, 300 HP @ 2200rpm

Based on the table value the maximum and minimum normal force imparted on connecting rod is calculated using torque and power formula P = T The corresponding values are given in table 2.

Table 2. Specifications of connecting rod and Engine

Power in (HP)	Revaluation in Rpm	Developed Load KN
290	1800	15.74
300	2200	13.89

From the calculations we arrived 15.74 KN for 290 HP at 1800 rpm, and 13.89 KN for 300 HP at 2200 rpm. So we take a mean of these to forces, it is 14.82 KN. All the three forces are taken for static stress analysis consideration.

III. STATIC ANALYSIS OF CONNECTING ROD USING COSMOS XPRESS STUDY

Finite element analysis isdone on connecting rod by using the software facility. First the model is created based on the actual dimensions measured from Verniercaliper, micrometer etc.

Then the material properties such as density, young modulus and poison ratio all are given. We consider the chrome stainless steel as material so the respective properties are given in Table 3.

Table no 3. Material property of connecting rod.

Density	7300 kg/m3
Yield tensile strength	2.412×10^{8} N/M^{2}
Poisson's ratio	0.26
Young's modulus	1.9×10^{11} N/M^{2}

After imparting the material properties, the constrains is asked to given. The smaller end of connecting rod is fixed at piston, so the constrain is given to smaller end inner circumference area. Next is the load applying, at the smaller end is fixed, the load is applied at the bigger end, assuming the force is tension and applied right direction to the face of bigger end of connecting rod.

The three loads, such as 15.74 KN, 14.82 KN, and 13.89 KN is applied for each analysis, and the corresponding stress distribution, deformed shape of connecting rod after load condition, area below the factor of safety as one are derived from the cosmos xpress study feature from solidwork software.

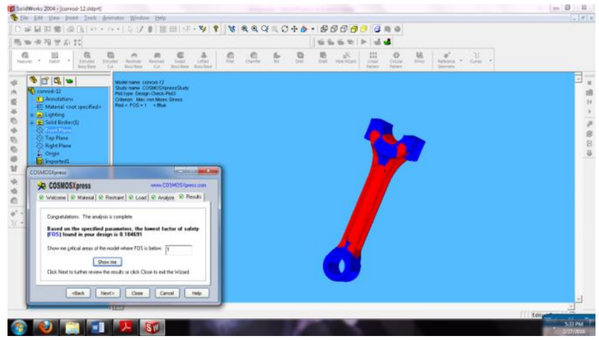


Fig 2. Critical Area which is FOS below one.

In figure 2 shows the red mark which is critical at loading, which are factor of safety below one under 15.74 KN loading. Other two also same results at FOS, difference is negligible.

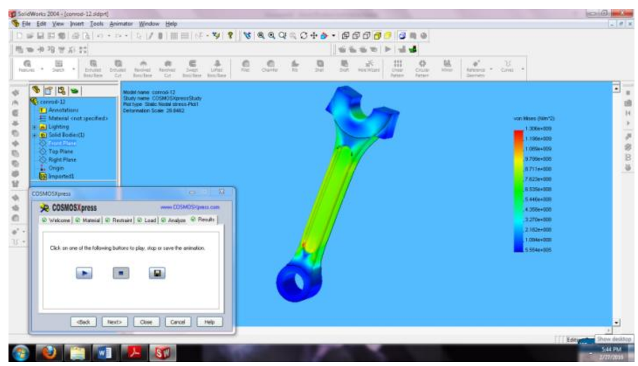


Fig 3. Stress distribution diagram of connecting rod under 15.74 KN loading

Figure 3 shown the von misses stress distribution for the 15.74 KN load of connecting rod. From graphical representation, apart from the smaller and bigger end of connecting rod, all the areas are subjected to heavy stresses.

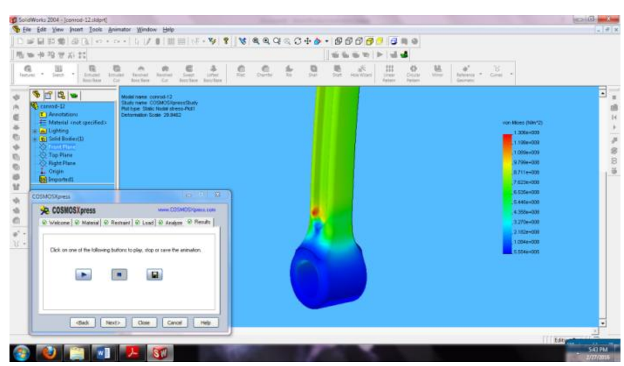


Fig 4. Close up view of stress distribution at 15.74 KN loading

From the three different loading, the maximum stress occurred at the root of the smaller end of connecting rod

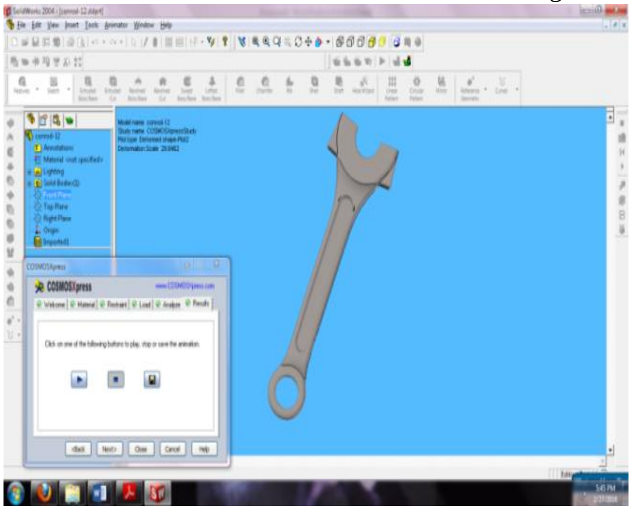


Fig 5. Deformed shape of connecting Rod at 15.74 KN loading

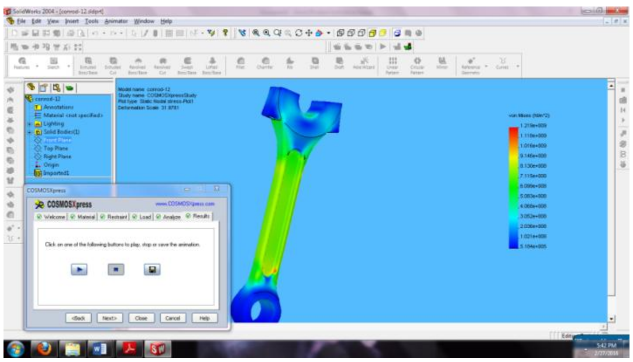


Fig 6. Stress distribution in connecting rod at 14.82 KN loading

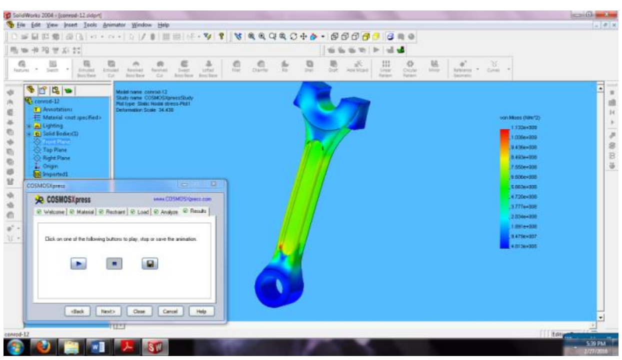


Fig 7. Stress distribution in connecting rod at 13.89 KN loading

Figure 7 clearly shows the stress distribution and deformation of connecting rod at lower load such as 13.89 KN.

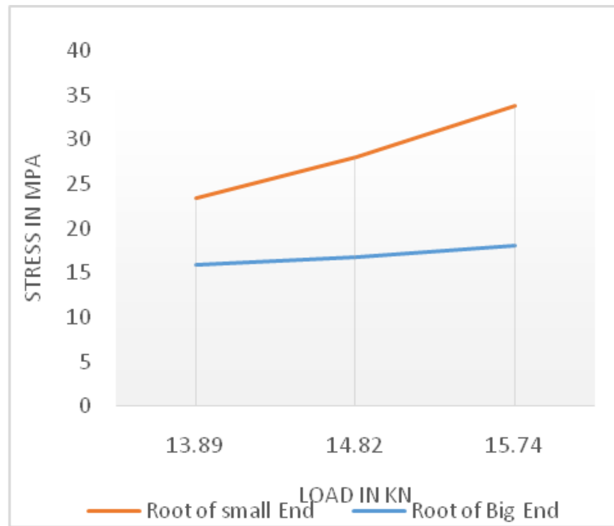
IV. Result and Discussion

From the figures 4, 6, & 7 it is sure, that the maximum stress is occurred at the root of smaller end of connecting rod. So the most significant should be given at the root of connecting rod smaller end.

Table 4. Maximum Stress at Different regions with respect To Load

Applied Load in KN	Maximum stress occurred at different regions			
	At Big End	Root of Big End	Root of small End	At small End
13.89	1.54	15.94	23.41	1.82
14.82	1.67	16.82	27.98	1.64
15.74	2.02	18.11	33.84	1.98

The stress distribution at root of two ends is higher thanthat stress at inner part of smaller & bigger end. Graph 1. Shows the load Vs stress which is plotted for root of big end and smaller end.



Graph 1 Maximum stress in MPaVs Applied Load in KN

V. Conclusion

It is clear that root near by the smaller end is very critical to failure, especially at higher load such as 15.74KN. It might be caused by crushing load at gudgeon pin assembly.at initial stage the void formation occurs leads to fatigue failures of connecting rod.

References

[1] Mohammed et al. 'Failure Analysis of a Fractured Connecting Rod'. Journal of Asian scientific research, 2012, vol. 2 issue 11, pp737-741.
[2] Pathade et al. 'Stress Analysis of I.C Engine Connecting Rod' international journal of engineering and innovative technology (IJEIT), march 2012, Vol. 1,issue 3, pp12-15.
[3] Noor et al. 'Analysis of connecting rod based on finite element approach' Malaysian Technical Universities Conference on Engineering and Technology, March 2008, pp95-98.
[4] Chikalthankar et al.'FEA analysis of geometric parameters of connecting rod big end' international journal of engineering science and technology(IJEST), April 2012 Vol. 4,issue 4, pp1560-67.
[5] Doshi and Ingole 'Analysis of Connecting Rod Using Analytical and Finite Element' International Journal of Modern Engineering Research (IJMER), Jan-Feb. 2013, Vol.3, Issue.1, pp-65-68
[6] Ram Bansal 'Dynamic Simulation of a Connecting Rod Made of Aluminum Alloy Using Finite Element Analysis Approach' Method' IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE), Jan. - Feb. 2013 Vol 5, Issue 2, pp01-05.