International Journal on Recent and Innovation Trends in Computing and Communication ISSN: 2321-8169
Volume: 3 Issue: 3 1439 - 1443

Analysis of Crank End of Connecting Rod using Finite Element Method

Mohammad Umair Zaki
Faculty of Mechanical Engineering
Noida International University
Greater Noida, India
e-mail-umairzaki@yahoo.com

Prateek Joshi
Department of Mechanical Engineering
Noida International University
Greater Noida, India
e-mail-prateekjoshi72@yahoo.com

***Abstract*—** Connecting Rod is used practically in generally all varieties of automobile engines. Its acts as an intermediate link between the piston end and the crank end of an engine. It is responsible for transmitting the up and down motion of the piston end to the crankshaft of the engine, by converting the reciprocating motion of the piston end to the rotary motion of the crankshaft. Therefore, this study is carried out for the load and stress analysis of the crank end of the connecting rod. Also the results can be used for optimization for weight reduction and design modification of the bigger end of the connecting rod. For this the Pro-E software is used for modeling and analysis is carried out using ANSYS software. The results archived can help us identify the spot or section where chances of failure are high. Also the results obtained can be used to modify the existing designs so that better performance and longer life cycle can be archived.

***Keywords*—** *Connecting Rod, Pro-E, FEA, Ansys Workbench, Crank, Crankshaft, Piston.*

*****

1. Introduction

Connecting Rod is used practically in generally all varieties of automobile engines. Its acts as an intermediate link between the piston end and the crank end of an engine. It is responsible for transmitting the up and down motion of the piston end to the crankshaft of the engine, by converting the reciprocating motion of the piston end to the rotary motion of the crankshaft. While one end or the small end the connecting rod is connecting to the piston of the engine by the piston pin, the other end or the bigger end is connected to the crankshaft with the big end bearing lower half by the two bolts.

Generally connecting rods are made up of cast iron either through the forging or powder metallurgy, as these methods provides high productivity and that too with lower cost. Forces generated by mass and combustion of fuel acts upon connecting rod, which results in bending and axial stresses.

Thus here it order to study the stress concentration and deformation in the connection rod at the crank end, firstly the model of the connecting rod is prepared and then it is analyzed using the Finite Element Method and results thus obtained will provide us the required data. Further study can carried out later on based on dynamic loading conditions of the connecting rod and also improvement in the design can be made for better life cycle and operation condition.

Pro/ENGINEER Wildfire 4.0 software is used for modeling of the connecting rod and ANSYS 12.1 is analysis part. ANSYS is an analysis system which stand foe advanced numerical system simulation. It is CAE software, which has many FAE capabilities, ranging from simple linear static analysis to complex, non-linear, transient state analysis. With solid modeling, the geometric shape of the model is described, and then the ANSYS program automatically meshes the geometry with nodes and elements. The size and shape of the elements that the program creates can be controlled. In order to estimate the results at each and every point meshing is done and also accurate results are obtained. Here the elements formed after meshing are tetrahedral shape. Loading in ANSYS can be applied either by solid model (by points, lines & areas) or by the finite-element model (nodes & elements).

FEM helps tremendously in producing strength and stiffness visualization and also by minimizing weight, material and cost.

2. Finite Element Method

[5]Finite Element Method is a numerical method to find out approximate solution of problem. It subdivides a whole problem into smaller simpler parts, called finite elements, and solve them for the problem. The main feature of FEM is that it can handle complicated boundary (and geometries) with ease.

Steps for the Finite Element Method are:-

- Creating the Model
- Importing the model
- Defining the element type
- Defining the material properties
- Meshing
- Applying boundary condition and constrains
- Applying loads
- Obtaining results and analyzing it.

3. Specification of the Problem

The objective of this work is design and analysis of Crank end of connecting rod, connecting rod specified here is made up of cast iron. Model of connecting rod is created in Pro-E Wildfire 4.0 and imported to ANSYS 12.1 for the analysis work in order to obtain the FEA results.

4. Modeling of the Connecting Rod using Pro-E

[1]Connecting rod of a Light Commercial Vehicle easily available in the, market of selected and its dimensions are measured and noted. According the measured data the model of the connecting rod is developed in the Pro/ENGINEER Wildfire 4.0. Model of the connecting rod and its Big end Bearing Lower half the separately developed of this study as shown in figures below,

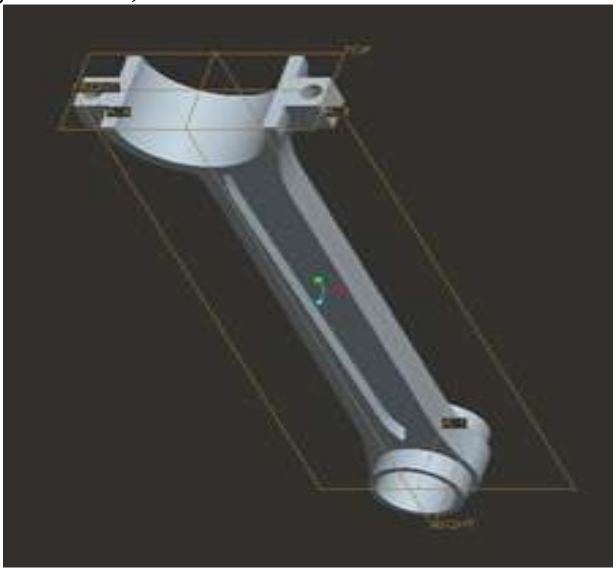

Figure 1: Model of connecting rod in Pro-E

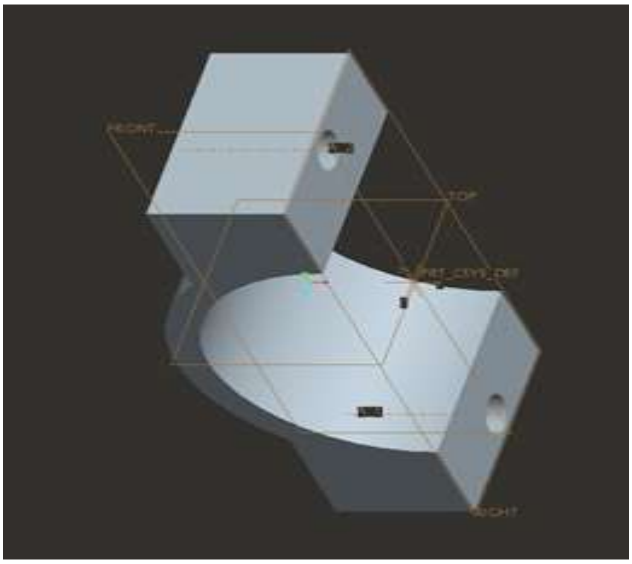

Figure 2: Model of Crank end Bearing Lower half

5. Finite Element Analysis using ANSYS

[2]The analysis of Crank end of connecting rod is carried out by using ANSYS software using Finite Element Method. Firstly the model files prepared in the Pro-E are exported to the ANSYS as an IGES files as shown in figure 3 &4 below;

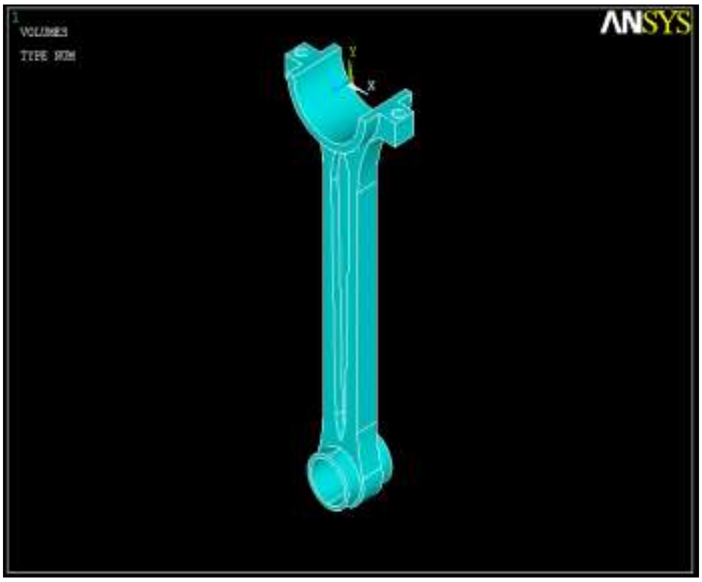

Figure 3: Imported model to ANSYS

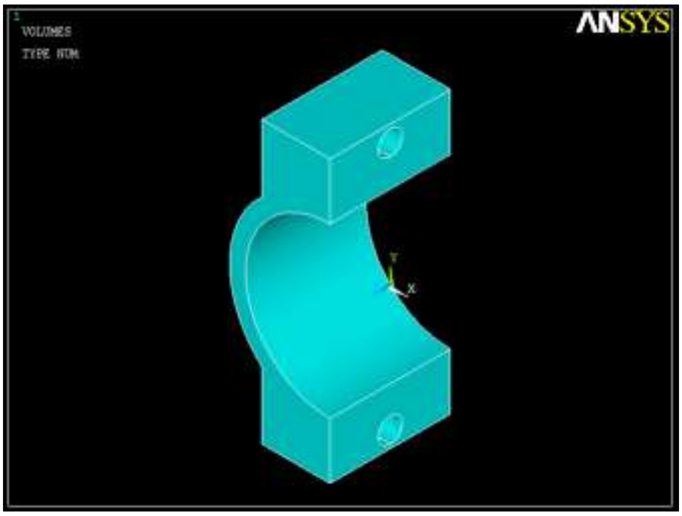

Figure 4: Imported model to ANSYS

After this all the areas are segmented as sometimes importing of model in ANSYS results in some imperfection. If the imported geometry generates any imperfection in geometry, which may results in in accuracy of final analysis results, then the geometry cleanup is done. Once cleanup is done, [3] now the material properties are applied on the model for the material used (here cast iron) as shown below. Once this step is done, the meshing of model is done. The mesh model of the components is as shown in figure 5 &6 below,

Material Properties used for Analysis;

Young's Modulus (E)	-	1.78e+005Mpa
Poisson's Ratio	-	0.3
Density	-	7.197e-006kg/mm^3
Yield Strength	-	250 Mpa

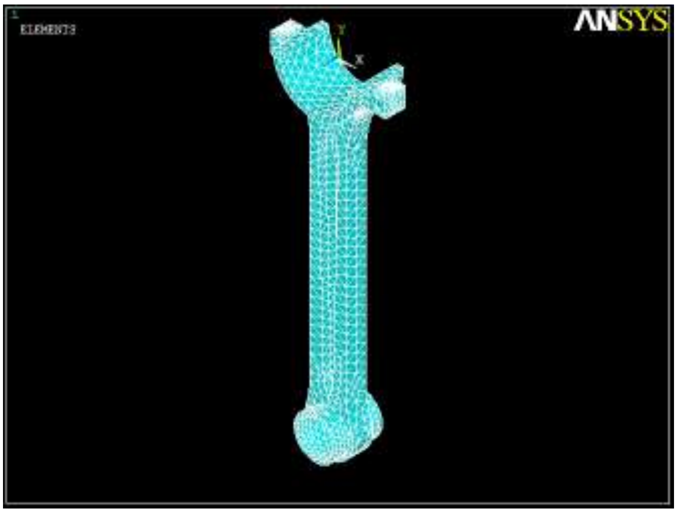

Figure 5: Meshed model of connecting rod

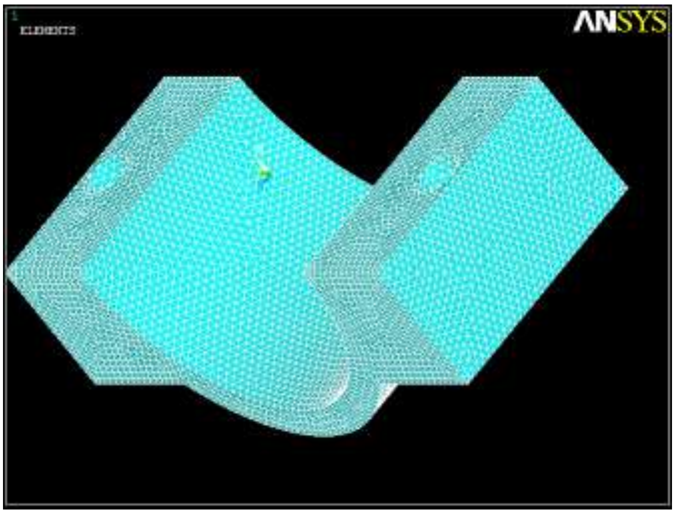

Figure 6: Meshed Model of Crank end Bearing lower half

Once the Meshing is done the boundary condition including DOF constrains, forces, loads are applied on the model. As shown fig 7, the pressure is applied on the big end bearing of the connecting rod, by keeping the small end or piston end fixed. The pressure of 13 Mpa is used.

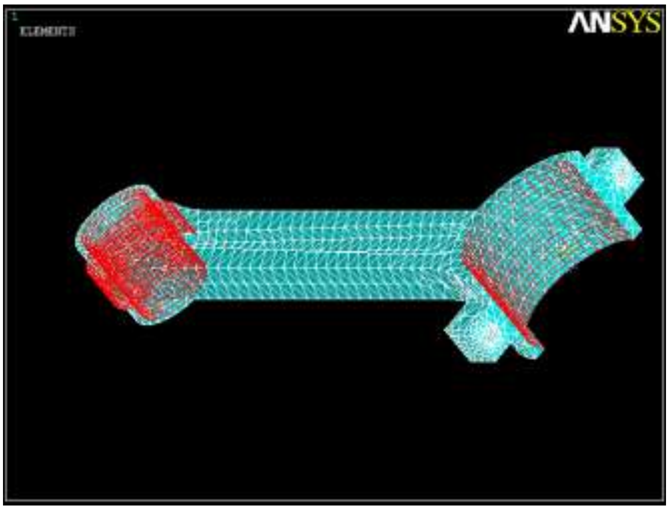

Figure 7: Load and boundary constrained on the connecting rod.

Now the Crank end bearing Lower half is taken and boundary constrained and load is applied on it. As shown in fig 8, the pressure is applied on the bearing face of the part, by keeping the bolt area fixed. The pressure of 13Mpa is used.

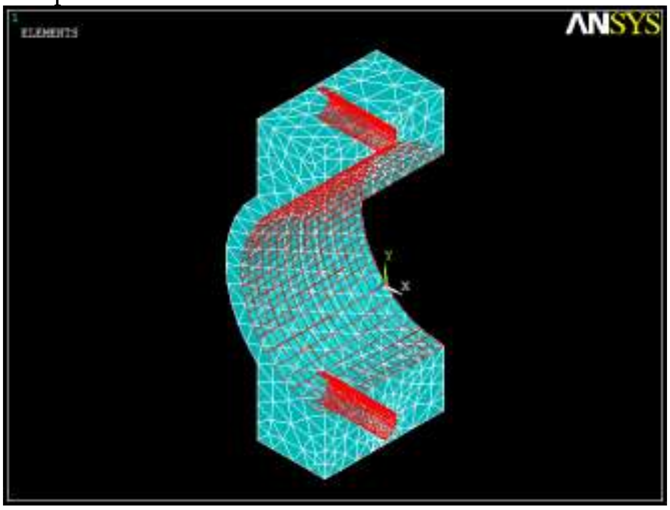

Figure 8: Load and boundary constrained on the Crank end Bearing Lower half.

6. Results of Static Analysis:

The static analysis of connecting rod big end was conducted to identify the fatigue locations on it. The term static implies that the forces do not change with time. [6][7]Results of the static analysis are shown via stress and deformation under the applied load. The FEA results of static analysis of both the components are shown in fig 9, 10, 11 &12 below;

ISSN: 2321-8169
1439 - 1443

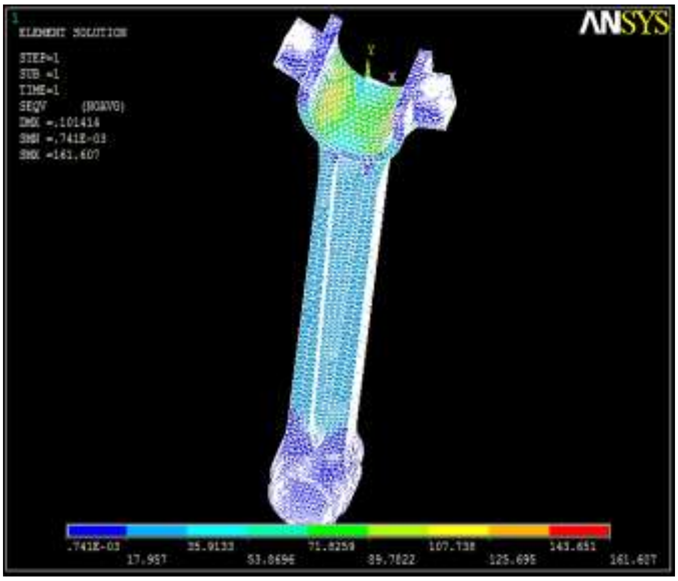

Figure 9: Equivalent Von-misses stress

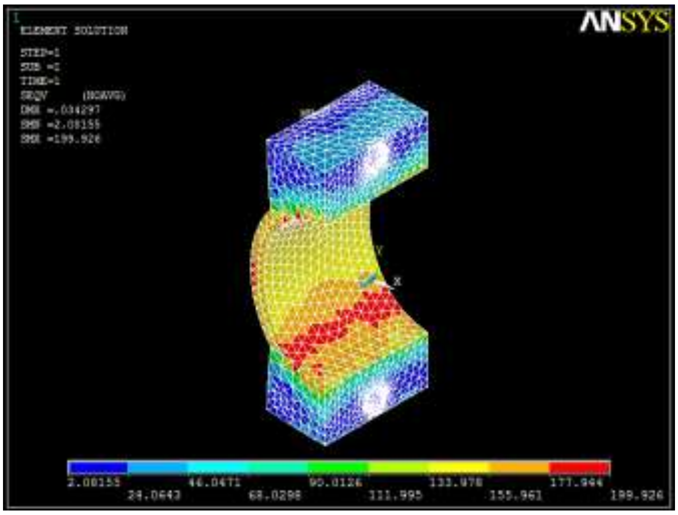

Figure 10: Equivalent Von-misses stress

[4]From the fig 9, the maximum stress occurs at the crank end of connecting rod is 161.40 Mpa and the minimum stress occurs at the crank end of connecting rod is 0.741E-3 Mpa. From the fig 10, the maximum stress occurs at the crank end bearing of lower half is 199.92 Mpa and the minimum stress occurs at the crank end bearing of lower half is 2.08155 Mpa.

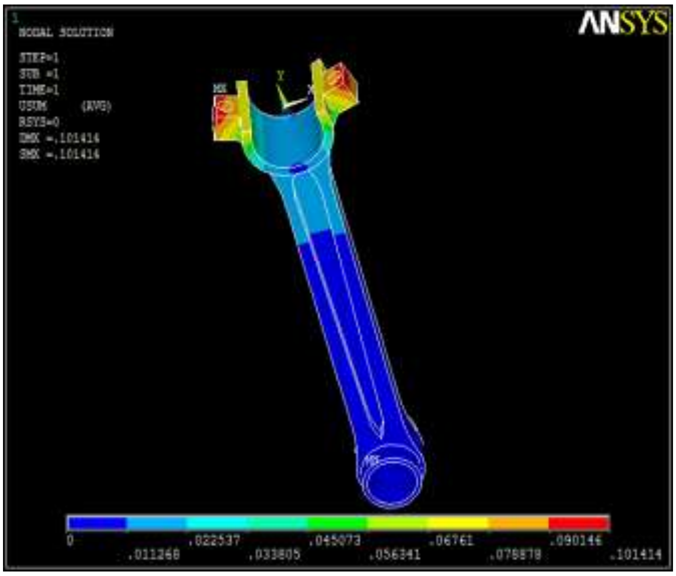

Figure 11: Total Displacement

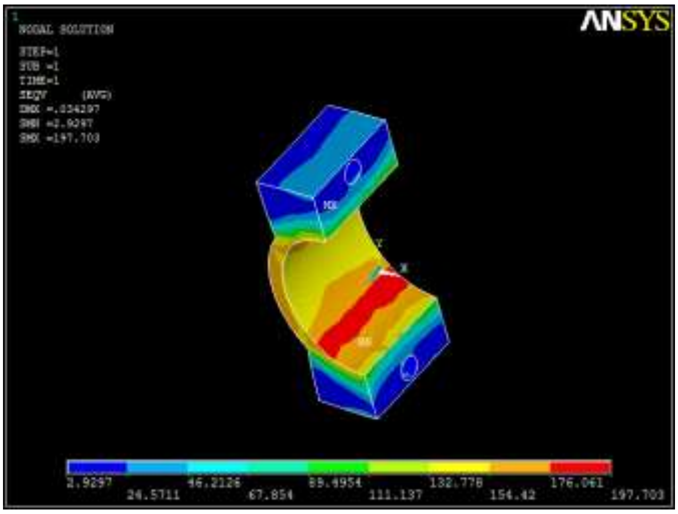

Figure 12: Total Displacement

From the fig 11, the maximum displacement occurs at the crank end of connecting rod is 0.101414 mm. From the fig 12, the maximum stress occurs at the crank end bearing of lower half is 2.9297 mm.

7. Conclusions

The following conclusion can be drawn from this study:-

1. This analysis helps to identify high & low stress region from the input of the material properties, boundary condition and loads.
2. The study shows the portions or areas where maximum stress and maximum deformation develop, as it is more susceptible to the failure.
3. The areas where the stress concentration is more, in order to minimize the stress at that stressed portion material can added.

ISSN: 2321-8169
1439 - 1443

4. Also, the areas where the stress concentration is less, in order to optimize the materials used material can removed formed that portion.
5. Increasing the thickness or diameter of the crank end bearing housing can reduce the deformation.
6. Hence there is provision for weight reduction which help in reducing the resulting inertial and centrifugal force which will improve the static results. Thereby improving the life and manufacturability.

References

[1] Pro/ENGINEER Wildfire 4.0 manual.

[2] ANSYS WORKBENCH 12 manual.

[3] G.M Sayeed Ahmed, Syed Hazed Shareef "DESIGN, FABRICATION AND ANALYSIS OF A CONNECTING ROD WITH ALUMUNIUM ALLOYS AND CARBON FIBER",ISSM:2319-8753,International Jornal of Innovative Research in Science, Vol. 3, Issue 10, October 2014.

[4] KuldeepB, Arun L.R, Mohammed Faheem "ANALYSIS AND OPTIMIZATION OF CONNECTING ROD USING ALFASiC COPMOSITIES", ISSN:2319-8753, International Jornal of Innovative Research in Science, Vol. 2, Issue 6, June 2013

[5] Aniket B.Phatangare, Prof. M.s.Mhaske, Prof. S.B.Belkar, Swapnil Kulkarni "USING FINITE ELEMENT ANALYSIS FOR DETERMINING FATIGUE LIFE OF CONNECTING ROD AS USED INA MOTOR-BIKE", ISSN:2249-8974, International Journal of Advanced Engineering Research and Studies.

[6] FEM analysis of connecting rod by R.Vozenilek, C.Scholz (The Technical University of Liberec, Halkova.

[7] K. Sundershn Kumar, Dr. K. Tirupathi Reddy, Syed Altaf Hussain "MODELLING AND ANALYSIS OF TWO WHEELER CONNECTING ROD",International Journal of Modern Engineering Research, Vol.2, Issue 5, Sep-Oct 2012.