Optimizing Connecting Rod Design Using Fillet Radius Modifications in ANSYS

Swet Chandan¹, Abhishek Singh²

¹Galgotias University, Yamuna Expressway Greater Noida 201307, India
swetchandan@gmail.com
²Galgotias University, Yamuna Expressway Greater Noida 201307, India
abhisheksingh10051993@gmail.com

Abstract.
In this paper we have investigated a connecting rod design of a 2wheeler. We have first measured the design parameters of the existing connecting rod and established the material used and cause of failure. We found that surface roughness in the big end is the main area for failure. Then a solid model of the same connecting rod was made using solidworks software. ANSYS software was employed to analyze and examine the effect of stress in different parts of the connecting rod. For the analysis we have started with the existing design parameters and then incrementally changed the fillet radius at big end with keeping the small end dimensions constant and then changing the fillet radius of the small end with keeping the fillet radius of the big end intact. Finally optimum parameters were selected for the modified design for the minimum stress conditions.

Keywords: Connecting rod, Fillet radius, stress analysis, Optimization, Finite element analysis.

1 Introduction

Connecting rod is one of the most important parts of any internal combustion engine which experiences cyclic loading. Mostly it consists of three parts: small end or pin end, the main body and the big end or the crank end. Aftzal et al (2003) has analyzed in the detail connecting rod dynamic loading and has shown how time dependent load and stress varies in a given connecting rod. Mirehei et al. (2008) has employed ANSYS software to investigate the various stress acting on the connecting rod used in the universal tractor. Webster et al. (1983) have extensively modelled a connecting rod used in the high speed diesel engine. They have employed iso-parametric 20-node, 15-node and beam element and used symmetry to reduce the computation time. Calculation of pressure distribution on the big end and small end due to force exerted by the piston is still used by most of the finite element analysis. Athavale and Sajanpawar (1991) has used modeled the connected rod using finite element and employed inertia load for the analysis. Khare et al.(2012) has done investigation on the connecting rod of a two wheeler and found that the fatigue load is the main culprit of the early failure of the given connecting rod. They have used fillet radius and contact stresses to analyze the problem. Lee et al.(2006) has used big end and small end fillet radius as a design parameters to analyze the connecting rod.

2 Model Description:
2.1 Sample Selection:

We have selected the sample of a two wheeler connecting rod after the user complains about the vibration and non-smooth driving condition by the driver. After investigation we have found that connecting rod of the vehicle is damaged due to spalling. Dimensions of the connecting rod were measured and with the help of laboratory tests composition of the material was established.

![Actual model of sample of Yamaha Crux connecting rod](image)

Fig. 1: Actual model of sample of Yamaha Crux connecting rod

2.2 SOLIDWORKS (3D) model of sample:

After the sample selection I made the 3D model of sample in SOLIDWORKS software with the same dimension of sample. For making the model I started with following steps.

2.2.1 Design Criteria:

Connecting rod converts reciprocating motion of piston into rotary motion of the crankshaft. It consists of two parts the upper part is forked section which fits on the crosshead bearing and lower part fit on the crankpin bearing. Different dimensions of internal combustion engine connecting rod are given below. For making the model of connecting rod we have used following configuration.

- Connecting rod length=12.2 cm
- Piston diameter=1.8 cm
- Crank radius=1.35 cm
- Stroke length=2*Crank radius=3.7 cm
- Mass of connecting rod=0.52894 kg
- Volume of connecting rod=6738.1 mm³
With the help of above parameters 3 D model was constructed for the connecting rod in the SOLIDWORKS 2016 software. The SOLIDWORK model of the connecting rod is shown in figure 2.

Fig. 2: SOLIDWORKS (3D) model of sample connecting rod

| S No | Material Property          | SAE-4140 |
|------|---------------------------|----------|
| 1.   | Young’s modulus (MPa)     | 71000    |
| 2.   | Density                   | 7.85g/cm³ |
| 3.   | Poisson’s ratio           | 0.27     |
| 4.   | Bulk Modulus (MPa)        | 140e+003 |
| 5.   | Shear Modulus (MPa)       | 80e+003  |

Table 1: Mechanical properties of materials (From Spectro Analysis)

3. Finite Element Analysis under the compression loading condition:
3.1 Boundary conditions and Load:
Distributed load (Webster, 1983) was applied at the small end (Pin end) internal surface while at the big end (crank end) surface displacement (zero) boundary was applied.

3.2 Changing the shapes and selection of design:
By changing the shapes like fillet radius, thickness and other parameters to find out for which design the stress is minimum. I have changed the fillet radius in CRANK SIDE (I SECTION) dimensions by using SOLIDWORKS software.

When the modification of design is completed then we have choose the best dimension which satisfied all the constrained and reduces the stress.

3.2.1 Effect of stress in connecting rod while changing the fillet radius in CRANK SIDE (I SECTION):
The initial analysis using ANSYS was performed with distribution of loads at big end. There were fixed displacement boundary conditions at pin end.

Finite Element Analysis with ANSYS:
The initial analysis was performed by using ANSYS software with distribution of load at big end and fixed support at small end. Figure 6 shows the modeling and analysis results of...
connecting rod used to determine the variation of stresses though change the fillet radius in CRANK SIDE (I SECTION)

Fig. 4: Modeling and Analysis result with ANSYS
Fig 5: Stress analysis result at big end (I section)

**ANSYS result examination:**

After the running the simulation in the ANSYS workbench we have found that stress value peaks in the fillet area zone. Further analysis has shown that fillet radius of 1.7mm has proven to be the minimum stress (212MPa) at the big end. Further for similar investigation has shown that for the small end the minimum stress (226MPa) occurs at the 1.6mm.

**Fig 6 Analysis results**

4. Conclusion:
We have changed the fillet radius at crank end (I section), pin end (I section) in the model. And also we have analyzed the stresses. We have found that the value of stress is less for fillet radius 2.5 mm in small end and 3.0 mm fillet radius at crank end.

The following conclusions can be obtained from this study.
- Change in the fillet radius greatly affects the maximum stress value inside the connecting rod.
- Change in the Design parameters of the big end in the connecting rod affects more than the small end.

5. Future Work:

The effect of change in the material and use of different types of coating can be employed to study the performance of the connecting rod.

6 REFERENCES:

1. Afzal.A and Shenoy.P 2003 Dynamic Load Analysis and Fatigue Behavior of Forged Steel vs Powder Metal Connecting Rods American Iron and Steel Institute
2. Mirehei,A, Hedayati.M, Zadeh, Jafari.A, Omid A.,2008 Fatigue analysis of connecting rod of universal tractor through finite element method (ANSYS) Journal of Agricultural Technology, vol-4(2), pp. 21-27
3. Webster, W. D., Coffell R., and Alfaro D., 1983 A Three Dimensional Finite Element Analysis of a High Speed Diesel Engine Connecting Rod SAE Technical Paper Series Paper No. 831322 .
4. Athavale, S. and Sajanpawar, P. R., 1991 Studies on Some Modelling Aspects in the Finite Element Analysis of Small Gasoline Engine Components Small Engine Technology Conference Proceedings, Society of Automotive Engineers of Japan, Tokyo PP. 379-389
5. Khare.S., Singh O.P., Bapanna K., Dora, C. Sasun, 2012 Spalling investigation of connecting rod TVS Motor Company Ltd. Research and Development, Hosur, Tamilnadu, India Engineering Failure Analysis vol.(19) pp. 77–86.
6. Lee. S, et.al. Failure Analysis of Connecting Rod at Big End Trans Tech Publications, Switzerland, vol(306-308) pp. 345-350 (2006).