Analysis and Optimization of Connecting Rod

[1]Altayeb Mahfouth, *Dept of Mechanical Engineering, Sabratha U*, *Altayeb_M85@yahoo.com*
[2]Muhand Murchid, Dept of Mechanical Engineering, Sabratha U, miral.muhand@gmail.com

ABSTRACT

In this study, a Finite Element Analysis and design optimization study were carried out on a connecting rod using Cad software Solidworks. First, the connecting rod was modeled in Solidworks. Then, two finite element models for tensile and compressive loads were analyzed. The sensitivity analysis was conducted; a convergence has been achieved with 1.5 mm uniform element length. The peak stresses mostly occurred in the transition area between pin end, crank end, and shank region. The obtained results from the stress analysis were used in the optimization study. The optimization study was carried out to investigate weight reduction opportunities by modifying some connecting rod dimensions with constraining allowable stresses and factor of safety. The percentage weight reduction obtained was 11.88% by optimization with a minimum factor of safety having a value of 2.2 and maximum Von Mises stress having a value of 127 MPa. The maximum stress occurred in the static structural analysis are less than the yield strength of the material.

KEYWORDS

Connecting Rod, Solidworks, Modeling, Finite Element Analysis, Optimization

Introduction

The connecting rod in the engine of the vehicle is one of the important parts that connect the crankshaft with the piston, and this link is subjected to compressive loads and high tensile stress during the process of rotation of the engine, so consider when the design of this connecting rod to overcome this stress. However, to achieve this purpose, they use some easily stress analysis technologies such as computer aided design (CAD), computer aided manufacturing (CAM), and computer aided engineering (CAE) during the product cycle and analyzing it by using FEA. Numerous researches about have been done on connecting rod but still a lot to be done. here some of these researches.

The different grades of auxiliary steel, titanium, and aluminum can be manufactured the connecting rod. The most widely produced of the connecting rod is made from steel rods. due to their high strength and long fatigue life is considered the best applications used for daily drivers[1].

Both tensile and compressive loads are acting on connecting rod but compressive load are much more prominent than tensile load, hence the designed had been done due to compressive forces. Since, connecting rod is pivoted at both closes by cylinder stick and wrench stick and encounters compressive strengths; hence, we will say that it will carry on like a strut[2]. The Pro/E Wildfire 4.0 and ANSYS V12 was used to model the rod and analysis it respectively to evaluation and optimization purpose of connecting rod. The weight was decreased by 0.477g in this study and little end was noted under maximum stresses. Also the study resulted the change in design parameters can achieve better results and increase in material in the stressed area can diminish stresses. The design and optimization was the important parameter for fatigue strength [3].

The weight of connecting rod was reduced by 43.48% by changing the material from aluminum 360 to aluminum 6061-9% SiC-15% fly ash, and comparatively much stiffer than the former [4].

The changing the material lead to reduce the weight of the current connecting rod. Also the geometry, which optimized 20% lighter than the old connecting rod [5]. Von misses stress distribution on the connecting rod was the greatest on the web zone and the smallest on the crank end zone [6].

The area which prone to failure it was near to root of the smaller end, might be because to higher crushing load due to gudgeon pin assembly [7].

In this paper, connecting rod of Hero Honda splendor is chosen as a model for study whose dimensions data belongs to P. G. Charkha & S. B. Jaju's research paper. This work considers two computer aided design studies using Solidworks. The first study is intended to conduct static load analysis and validation of the connecting rod (connecting rod of four stroke single cylinder petrol engine). The second study is to explore weight reduction opportunities for a connecting rod.

Objectives

The aim of this study is to create three dimensional models in the Solid program, then generate simulation studies in Solidworks under various boundary and load conditions, and carry out mesh sensitivity analyses for the finite elements methods used in Solidworks. after that validate the numerically obtained results " solidworks simulation results" by comparing it with reported results which obtained from ANSYS Simulation in similar conditions of applications; the ANSYS simulation results are belong to S. Chorghe and A.D. Dhale's research paper. finally Conduct an optimization study in Solidworks to explore weight reduction of a connecting rod by identify design objective, specifying design variables and stating design constraint.

Modeling of connecting rod

The connecting rod has been modeled with the solidworks software where the final model of the connecting rod is shown in figure (1) below.

The flowing steps has been used to create the required model of the connecting rod: -

Step 1: - A connecting rod consists of a long shank, a small end and big end. The classification of the connecting rod is made by the cross sectional point of view (cross section of the shank). This cross section can be I-section, H-section, Tabular section, and circular section as can be seen in figure 1. In this paper an I-section connecting rod of single four stroke petrol engines was created with the help of solidworks software. The base sketch of the connecting rod is drawn on the front plane as shown in figure (2).

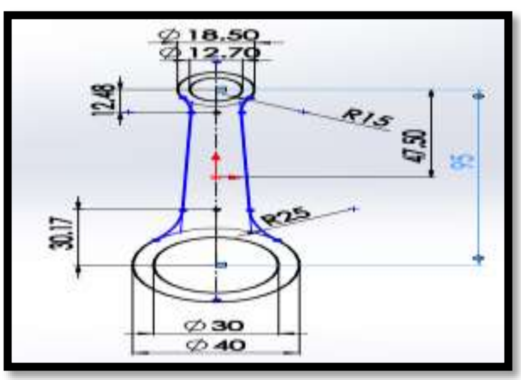

Figure 1 The base sketch of the connecting rod

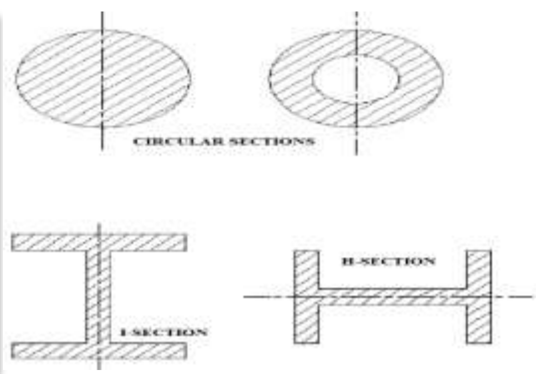

Figure 2 Schematic illustration of types of cross sectional

Step 2: - Then the Extruded Boss/ Base command was used to create the 3D creation of the model.

Step 3: - Design simulation setup, a static analysis study on a connecting rod has been run to calculate displacements, strains, stresses, and factor of safety distribution.

Step 4: - Defining the material of the model, in modern internal combustion engines, the connecting rods are mostly made of steel for production engines, but for lightness and the ability to absorb high impact at the expense of durability, connecting rods can be made of aluminum alloys. Also titanium and cast iron are used as materials for the connecting rod. In this work, cast carbon steel is selected as the connecting rod's material and all properties of the connecting rod's material as seen in table (1) below.

Table 1 Properties of the connecting rod's material

Material	**1023 Carbon Steel Sheet**
Density [Kg/m^3]	**7858**
Young's Modulus [MPa]	**2.049×10^5**
Poisson's Ratio	**0.29**
Tensile yield strength [MPa]	**282685049**
Compressive yield strength [MPa]	**282685049**

Step 5: - Setting boundary conditions, there are various forces acting on the connecting rod. In this study the first two forces have been considered as the boundary conditions which are force acting on the piston pin due to gas pressure and the inertia bending force.

Step 6:- Applying fixtures, various fixtures can be applied to a created model in Solidworks. In this study, a Fixed Geometry fixture was applied to the piston pin end over 180^o as part of the boundary condition to fix the model body.

Step 7: - Applying external loads, two finite element studies were analyzed. FEA for both tensile and compressive loads were conducted, for compressive loading of the connecting rod, the piston pin end was restrained and the crank end is assumed to have a uniformly distributed loading having a value of (4319 N).

Step 8: - Meshing and running the simulation study, this feature in Solid works subdivides the model into small pieces of simple shapes called elements connected at common points called nodes. The mesh size was refined. The element size was 1.28046 mm. Total number of nodes 90769 and total number of elements 59390 were generated at 1.28 mm element length.

Results of FEA

The two main results in FEA will be focused in this work, which are: -

1- Results of finite element analysis for compressive loading at the crank end

The obtained results of max and min of Von Misses stress and max and min shear stress as seen in the table (2) below.

Table 2 The result of compressive loading at the crank end

compressive load	Type	maximum	minimum	Place of occurrence
1	**Von misses stress**	**69.95Mpa**	$6.025 * 10^{-2}$ **Mpa**	**The max at cross section of shank and min at the crank end**
2	**Shear stress**	**23.96Mpa**	**-21.83Mpa**	**the crank end**
3	**Displacement**	$1.7 * 10^{-2}$	10^{-30}.	**the crank end**
4	**Strain**	$2.901 * 10^{-4}$	$2.305 * 10^{-7}$	**the crank end**

In addition, the figures (3), (4) below show the Von Misses stress and shear stress on the connecting rod.

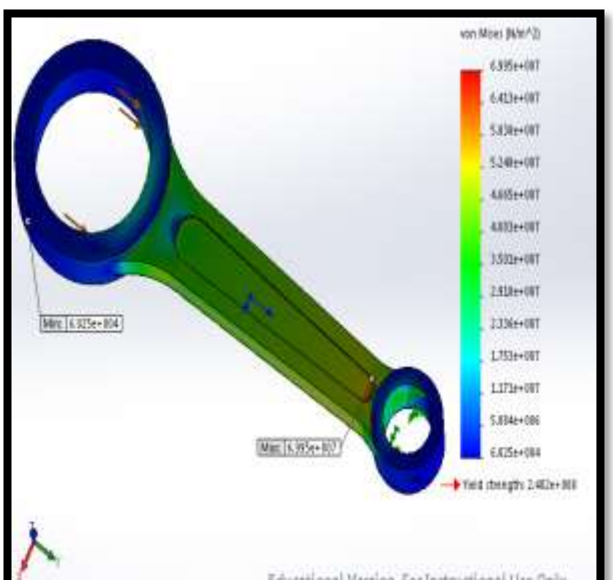

Figure 3 von mises stress distribution with static compressive load of 4319 N at the crank end

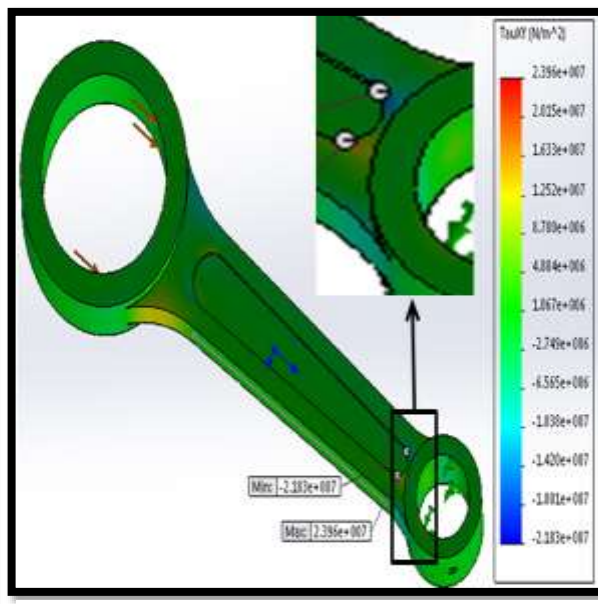

Figure 4Shear stress distribution with static compressive load of 4319 N at the crank end

2-Result of finite element analysis for tensile loading at the crank end

A tensile load having value of 4319N was distributed over 180º on the crank end while the piston pin end was fixed as seen in figure (5) below.

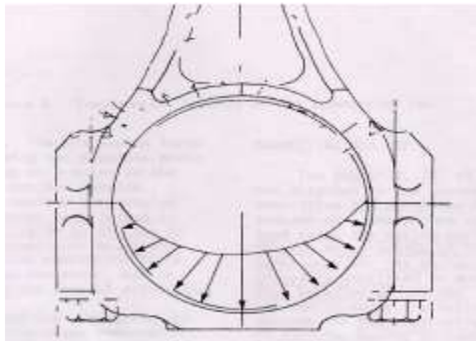

Figure 3Tensile loading of the connecting rod

The results of von misses stress and shear stress with tensile load at the crank end, while piston pin end is restrained under the value of load 4319N schedule in table (3) below.

Table 3 Result of finite element analysis for tensile loading at the crank end

compressive load	Type	maximum	minimum
1	Von misses stress	$2.173 * 10^{8} Mpa$	$1.390 * 10^{5}$ Mpa
2	Shear stress	$9.1710 * 10^{7} Mpa$	$-9.386 * 10^{7} Mpa$
3	Displacement	$8.451 * 10^{-2} mm$	$1.00 * 10^{-30} mm.$
4	Strain	$8.588 * 10^{-4}$	$3.632 * 10^{-6}$

Comparing the Von Mises stress results of static compressive and tensile loads

To record data resulted from running a simulation in Solidworks sensors are to be installed in designated positions within the connecting rod. In this work, nine points (From 1 to 9) as seen in figure (6) below.

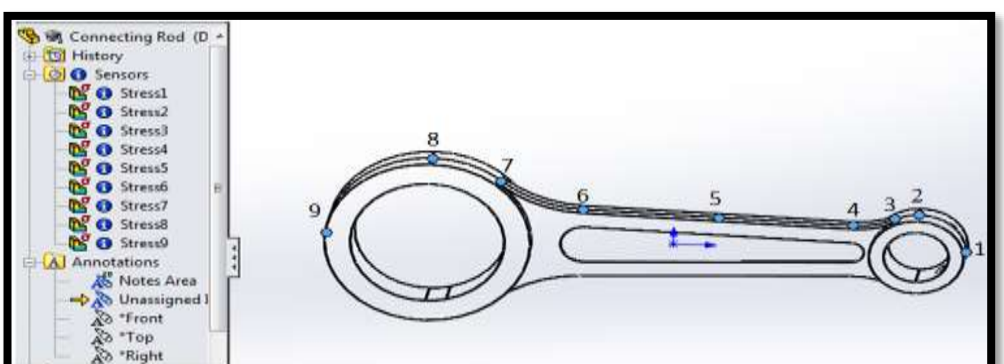

Figure 4 sensors to record Von Mises stress data at a few discrete locations on the mid plane labeled on the connecting rod, along the length

According to the sensors, which have been installed, one can notice that the Peak stresses mostly occurred in the transition area between pin end, crank end and shank region. The value of stress at the middle of shank region is well below allowable limit as shown in figure (7).

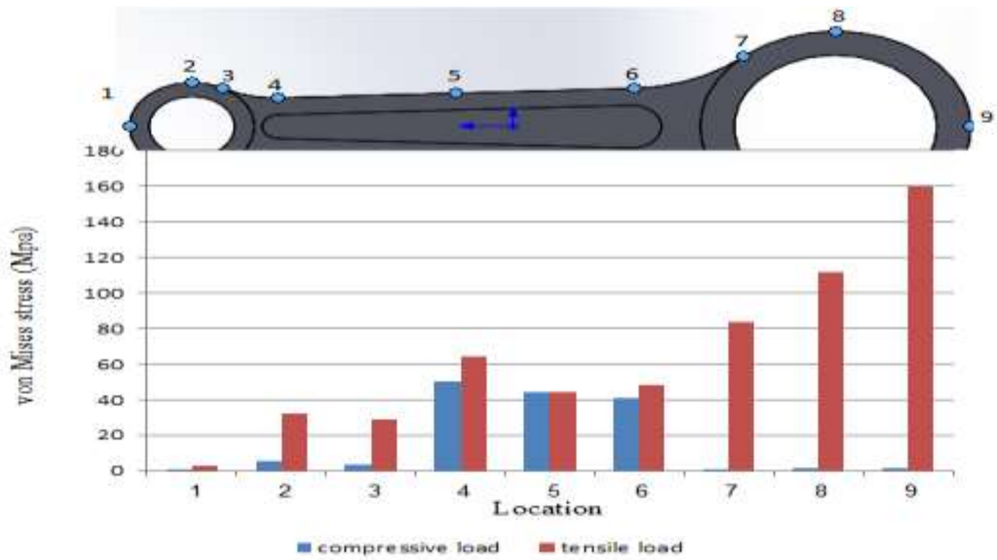

Figure 5 Von Mises stress at a few locations on the mid plane labeled on the connecting rod, along the length, for tensile and compressive loads

Mesh sensitivity analysis

For Mesh sensitivity and convergence the Von Mises stress was checked at nine locations, with various element lengths of 4 mm, 3.5mm, 3mm, 2.5mm, 2mm, 1.5mm, and 1mm. With each element length, the boundary conditions and the applied loads were all kept constant. The results of the recorded Von Mises stresses at all locations for various elements sizes are contained in table (4).

Table 4 The Von Mises stress data for various element sizes

Element Length (mm)	von Mises stress (MPa)								
	Sensor 1	Sensor 2	Sensor 3	Sensor 4	Sensor 5	Sensor 6	Sensor 7	Sensor 8	Sensor 9
4	0.500914	3.69242	13.4404	48.023	43.9734	40.505	1.1359	1.65416	1.88414
3.5	0.50664	2.70827	11.6512	49.0158	43.9778	40.4655	1.24588	1.59038	1.91566
3	0.526018	3.29496	11.419	48.549	43.9637	40.8573	1.32466	1.57987	1.83906
2.5	0.502262	2.81692	12.0709	48.7364	43.9721	41.0181	1.04887	1.54157	1.82686
2	0.515467	2.7485	11.8877	49.1177	43.9942	40.802	0.783886	1.54125	1.79797
1.5	0.520996	2.63117	11.8747	49.5682	43.9615	40.8768	0.707832	1.53149	1.77306
1	0.506793	2.72519	11.6241	49.6335	43.9578	40.8253	0.744147	1.51926	1.77472

The optimization of connecting rod

The main aim of the optimization to reduce the weight of the connecting rod under the effect of a compressive load having value of 4319 N due to the peak compressive gas load. Change in weight lead to change the dimensions of the connecting rod on the other hand there some dimensions cannot be changed, in this optimization study, the design variables were chosen to be some of the dimensions of the cross section of the shank (the thickness T2 & the web T1) as can be seen in figure (8) and (9).

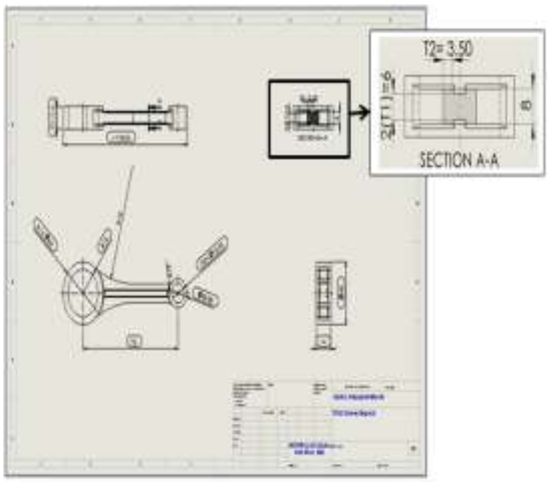

Figure 8Drawing of the connecting rod showing few of the dimensions that are design variables and dimensions that cannot be changed

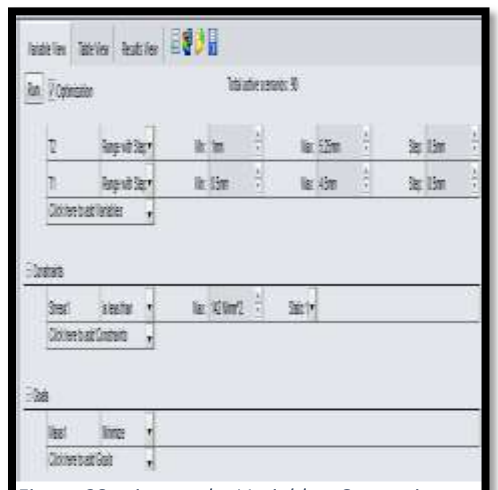

Figure 9Setting up the Variables, Constraints, and Goals for the optimization process

The best result was achieved of running the optimization, whereas the web, T1, was decreased by 1mm to be 2mm and the thickness, T2, was 1mm. At the same time, the mass was minimized to be just 116.261 g. The percentage of weight reduction was 11.88 %. In this case also the stress was maintained to be less than 142 MPa. Figure 12 shows the optimal design for the model. Thus, it was considered in this study that the values of 2mm for T1 and 1 mm for T2 are optimal values. Figure (10) shows the geometry of the optimized connecting rod.

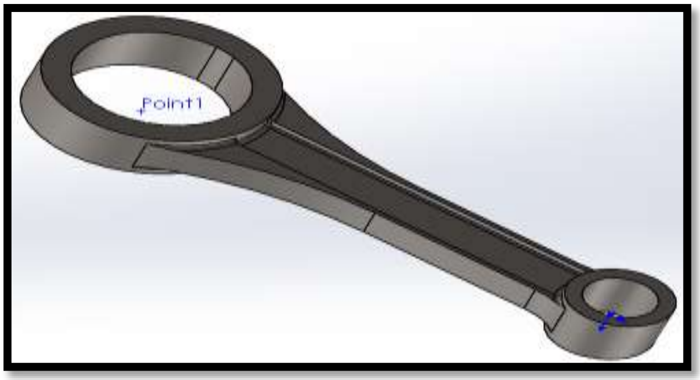

Figure 10 The geometry of the optimized connecting rod

Conclusion

A Finite Element Analysis with validation approach and a design optimization study were carried out on a connecting rod using Solidworks. While conducting these studies, some remarkable features have been used in Solidworks, in addition, a sensitivity analysis was conducted; a convergence has been achieved with 1.5 mm uniform element length. After the FEA have been run again with a uniform global element length of 1.5 mm. In this works also, an optimization study was carried out to investigate weight reduction opportunities by modify some connecting rod dimensions with constrain allowable stresses and factor of safety. The obtained results from the stress analysis were used in the optimization study. The percentage weight reduction obtained was 11.88% by optimization with minimum factor of safety having value of 2.2 and maximum von Mises stress having value of 127 MPa. Then, another FEA was done to the optimized connecting rod. The maximum stress occurred in static structural analysis are less than the yield strength of material. Hence the optimized design is safe.

References

[1]Ramani, M., Shukla, P., & Dr. Pushpen. (2014). FE-Analysis of Connecting Rod of I.C.Engine by Using Ansys for Material Optimization. *Int. Journal of Engineering Research and Applications*, 216-220.

[2]Singh, P., Pramanik, D., & Singh, R. V. (2015). Fatigue and Structural Analysis of Connecting Rod's Material Due to (C.I). *International Journal of Automotive*, 246-253.

[3]Pal , S., & kumar, S. (2012). Design Evaluation and Optimization of Connecting Rod Parameters Using FEM . *International Journal of Engineering and Management Research*, 21-25.

[4]Kuldeep, B., Arun , L., & Mohammed , F. (2013). ANALYSIS AND OPTIMIZATION OF CONNECTING ROD USING ALFASiC COMPOSITES. *International Journal of Innovative Research in Science, Engineering and Technology*, 2480-2487.

[5]Mr. Pranav , C. G., & Jaju, D. B. (2009). Analysis & Optimization of Connecting Rod. *Second International Conference on Emerging Trends in Engineering and Technology, ICETET-09* (pp. 86-90). IEEE.

[6]Gotiwale, D. G., & Ambekar, S. D. (2014). Design of Connecting Rod For Weight Reduction Using C70S6 Material. *International Journal of Scientific & Engineering Research*, 1253-1260.

[7]Gautam , A., & Ajit, K. P. (2013). Static Stress Analysis of Connecting Rod Using Finite Element Approach. *IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)*, 47-51.