Static Finite Element Analysis and Optimization of Two Wheeler Connecting Rod

Ganesha Ram[1], Dr. P. K. S. Nain[2], Mr. Pramod Kumar[3]
1- M.Tech [Automobile Engineering] Student, Galgotias University, Greater Noida, India
2- Professor at Galgotias University, Greater Noida, India
3- Assistance Professor at Galgotias University, Greater Noida, India

***Abstract:* - This paper deals with static analysis and optimization of a alloy steel connecting rod. Optimization is important as less time required to produce the alloy steel connecting rod which is stronger, lighter with minimum cost. The design and weight of the connecting rod control the engine performance. Hence optimization is to carried out. Descriptions of connecting rod have been evaluated to calculate the loads acting on it. Static analysis is carried out on the piston pin end and crank pin end of connecting rod then further study was show to explore weight reduction possibility for a production of connecting rod. The component is to be optimized for weight subject to constraint of allowable stress and factor of safety. A CAD model and FEA analysis of connecting rod is carried out to determine the maximum Von-Misses stresses for the given loading conditions using software SOLIDWORKS. The percentage weight reduction obtained was 13% by Optimization.**

Keywords: Connecting rod, Finite Element Analysis (FEA), Finite Element Method (FEM), Optimization, CAD, SOLIDWORKS.

1. INTRODUCTION

The Connecting rods are widely used in automobile/stationary engine. The function of connecting rod is to transmit the thrust of the piston to the crankshaft, and as the result the reciprocating motion of piston is converted into rotational motion of crankshaft. It consists of a pin-end, a shank section, and a crank-end. Pin end and crank-end pinholes are machined to permit accurate fitting of bearings. One end of connecting rod is connected to the piston by the piston pin. The other end revolves with the crankshaft and connected to the crank pin end.

Webster *et. al.* [1] was explained the loading of connecting rod in diesel engine. The Tension and compression loadings were used based on experimental results. It is highest stress occurred at four location of connecting rod. The upper area of cap end on the axis of symmetry, the transition region of bolt section and the lower rib, the transition region of the lower rib and connecting rod's bolt head.

Pranav *et. al*. [2] was carried out the FEA and optimization of connecting rod using ANSYS Workbench. The study two type of analysis, static analysis and fatigue analysis. The main objective of this study was explore the weight of connecting rod. The weight reduction of achieved by 9.24% under static loading conditions of existing connecting rod.

Pravardhan *et. al.* [3] was presented the FEA procedure for optimization for connecting rod weight and cost reduction. Weight reduction forged steel connecting rod by iterative procedure. This study result was in an optimized connecting rod 10% lighter and 25% less expensive as compared to existing connecting rod.

Vasile *et.al.* [4] Was presented a method used to verify the stress and deformation of connecting rod using FEM with ANSYS. The obtained results by this method to compared results obtained by classic calculation, in similar conditions of application, and after wards conclusion were drawn.

2. STEPS IN MODELING OF CONNECTING ROD

Connecting rod was modeled with the help of SOLIDWORKS Software. The Orthographic and solid Model of the connecting rod is shown in figure below.

The following is the list of steps that are use to create the required model:

- The base feature is created on the three orthogonal datum planes.
- Creating two circular entities on either sides of rod crank and piston pin end.
- Filling the material between the crank and piston pin end (with the help of EXTRUDE option).
- The second feature is also created on datum planes.
- A cut-feature is created on the second feature.
- Creation of plane perpendicular to axis for first hole.
- Creating the first hole at the piston end.
- Creation of plane perpendicular to axis for second hole.
- Creating the second hole at the piston end.

The Weight difference between the measured and Software calculated value was found to be less than 1%. This is an indication of the Software model accuracy.

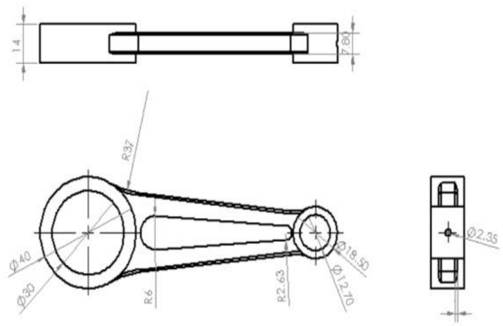


Figure1: Orthographic model of connecting rod.

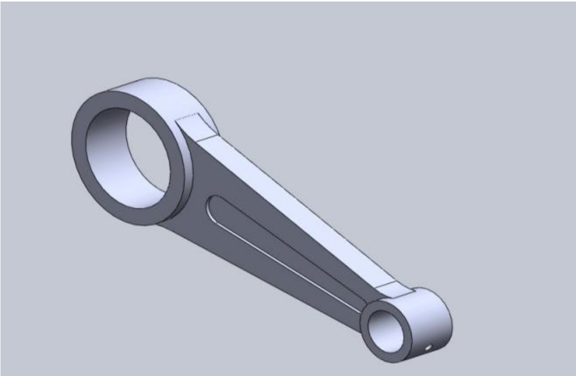


Figure2: Solid model of connecting rod.

3. FINITE ELEMENT ANALYSIS

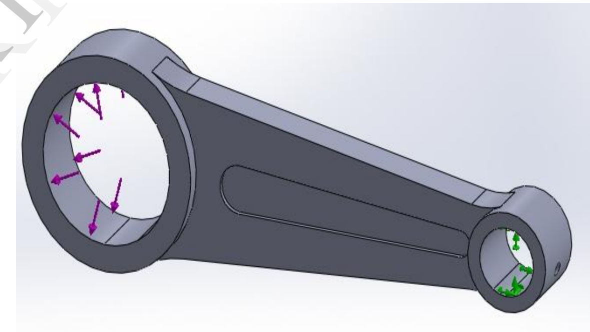
The Objective of FEA was to investigate stresses and problem area experienced by the connecting rod. From Stress contours, the state of stress as well as stress concentration factors can be obtained and consequently used for life predications. Alloy steel connecting rod was selected for the FEA, since this connecting rod was also used at optimization. Linear elastic analysis was used to connecting rod is designed for long life where stresses are mainly elastic. Therefore material properties are show in table below:

Table1: Material properties of connecting rod

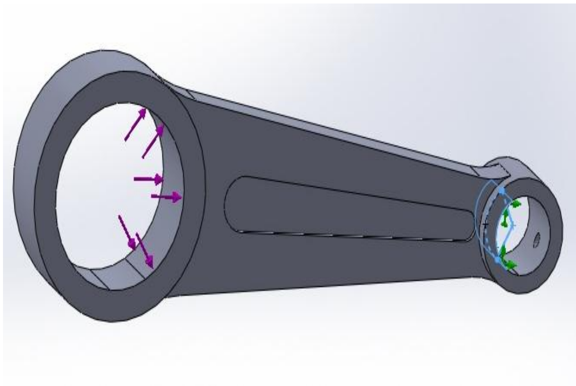
Material Properties	unit	scalar
Young's modulus (E)	GPa	200
Poisson's ratio	Unitless	0.30
Mass density	Kg/m^3	7820

A parabolic tetrahedral element was used for the solid mesh. Sensitivity analysis was performed to obtain the optimum element size. This analysis was performed iteratively at different element lengths until the solution obtained proper accuracy. Convergence of stresses was observed, as the mesh size was successively refined. The element size of 1.5 mm was finally considered. Total number of elements 43367 and Total number of nodes 67038were generated at 1.5 mm element length.

Tension and compression loads were applied as pressure on the bearing surfaces of the connecting rod. Webster et al. (1983) found the actual service conditions, the pin end tension by the piston pin causing distribution of pressure along the upper half of the inner diameter (over 180^o of the contact area), which is approximated by the cosine function. In compression, the piston pin compresses the bearings against the pin end inner diameter (over 120^o contact arc), causing uniform distribution of pressure. The pressure acts as normal to the contact surface area in a symmetric manner. Therefore, four cases were analyzed, two for tension loading (cosine pressure distribution over 180^o) and two for compression loading (uniform pressure distribution over 120^o), each with end, or loading the crank end while constraining the crank end, or loading the crank end while constraining the pin end. The constraints were applied for all six degrees of freedom over 180^o for tension loading cases, and over 120^o for the compression loading cases. Figure3 shows the boundary conditions used for the finite element analysis for the cases of tension as well as compression loading of the crank end, while the pin end is constrained.



(a)



(b)

Figure 3: Loading and constraint used for FEA. (a) Tension loading at the crank end. (b) Compression loading at the crank end.

4. RESULTS OF FINITE ELEMENT ANALYSIS

Figure 4 shows the von misses stress contours for the cases of tension loading at the crank end, while constraining the pin end. Figure 5 shows the von Misses stress contours for the cases of compression loading at the piston end, while constraining the crank end.

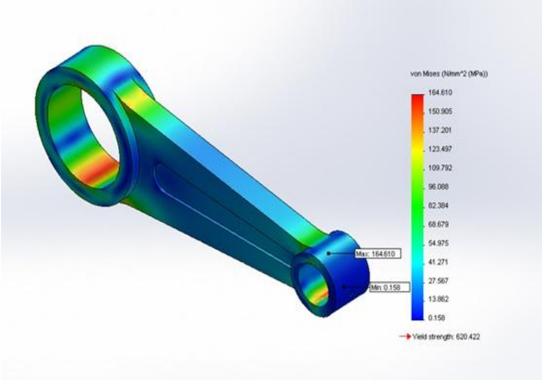


Figure 4: Von-Misses Stress Contours for tension loading at the crank end.

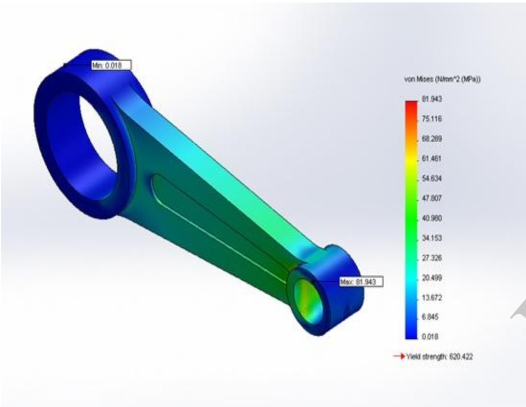


Figure 5: Von-Misses Stress counters for compressive loading at piston end.

5. OPTIMIZATION STATEMENT

The Objective of optimization task was to minimize mass of connecting rod under the effect of a load range comprising the two extreme loads, the peak compressive gas load, such that the maximum, minimum, and equivalent stress amplitude are within the limits of allowable stresses. The connecting rod production cost was minimized. The buckling load factor under the peak gas load has to be permissible.

Mathematically stated, the optimization statement would appear as follows:

Objective: - Minimize Mass and Cost.

Subject to:-

- Compressive load = peak compressive gas load.
- Maximum Stress < Allowable Stress.
- Side constraints (component dimension).
- Manufacturing constraints.
- Buckling load > Factor of safety X the maximum gas load (Recommended FOS, 3 to 6).

The optimized model of connecting rod shows in Figure 6.

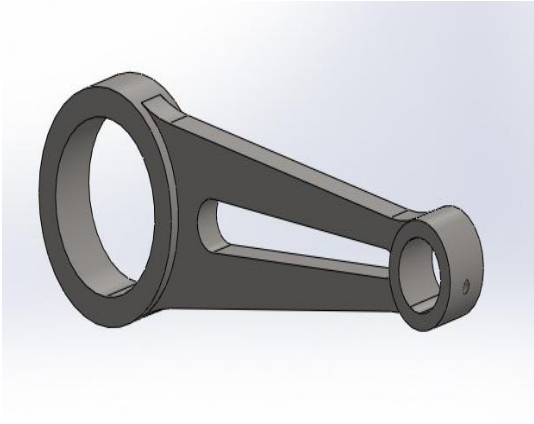


Figure 6: Optimized solid Model of Connecting Rod.

Original Model:

Mass = 170 gram

Factor of safety = 3.78

Optimized Model:

Mass = 148 gram

Factor of safety = 3.75

Following Table 2 gives the comparison between equivalent stress and deformation at different loading conditions for original model as well as optimized model.

Table 2: Maximum Von-Misses Stresses and Deformation of Original and Optimized Connecting Rod

model		original	optimized	original	optimized
Types of loading	Applied pressure	Max Von Misses Stress- MPa		Deformation	
Tensile Loading	Crank end	164.610	165.962	$4.725E^{-2}$	$5.116E^{-2}$
	Pin end	231.590	221.047	$1.539E^{-2}$	$1.654E^{-2}$
Compressive Loading	Crank end	59.933	80.550	$3.194E^{-2}$	$2.335E^{-2}$
	Pin end	81.943	101.081	$1.224E^{-2}$	$3.390E^{-2}$

Percentage Weight Reduction

= (Original Mass - Optimized Mass)/ (Original Mass) ×100

= (170-148)/170×100

= 13%

6. CONCLUSION

- 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. Also forces at pin end are lower in comparison to the forces in crank end so strength of pin end should ideally be lower in comparison to the strength of crank region.
- In tensile loading with pressure applied at the crank end and pin end restrained maximum value of Von-Misses stress was observed for both original as well as optimized model compared to other loading conditions this is critical loading condition used for optimization study.
- Factor of safety was greater than 3.7 in both tensile as well as compressive loading cases for both original as well as optimized model.
- Percentage weight reduction was about 13% which will save material directly to reduce the manufacturing cost with increased engine efficiency.

7. REFERENCES

[1] 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.
[2] Pranav G. Charkha and Dr. Santosh B. Jaju "Analysis and optimization of a connecting Rod" Second International Conference on Emerging Trends in Engineering and Technology, ICETET-09.
[3] Pravardhan S. Shenoy and Ali Fatemi "Connecting Rod Optimization for Weight and Cost Reduction" 2005, SAE Technical paper series, paper No.-2005-01-0987.
[4] Vasile George Ciotia, Imre kiss "Computer aided design of the connecting rod"
[5] SOLIDWORKS Manual for design and simulation.