IJRAR.ORG E-ISSN: 2348-1269, P-ISSN: 2349-5138

INTERNATIONAL JOURNAL OF RESEARCH AND ANALYTICAL REVIEWS (IJRAR) | IJRAR.ORG

An International Open Access, Peer-reviewed, Refereed Journal

DESIGN AND STRUCTURAL ANALYSIS OF CONNECTING ROD USING FEM

Anshul Kamble[1] , Vivek Singh[2] , Rajesh Rathore[3]

[1]M.E. Scholar, Mechanical Engineering, Vikrant Institute of Technology and Management, Indore Madhya Pradesh, India

[2]Assistant Professor, Department of Mechanical Engineering, Vikrant Institute of Technology and Management, Indore Madhya Pradesh, India

[3]Head and Professor, Department of Mechanical Engineering, Vikrant Institute of Technology and Management, Indore Madhya Pradesh, India

Abstract:-

This study utilizes Finite Element Analysis (FEA) to design and analyze a connecting rod for internal combustion engines. The FEA simulations assess the structural integrity and performance of the connecting rod under different loading conditions. The findings aid in optimizing the design by identifying areas where maximum stresses are generated, improved strength, and increased fatigue life. The study highlights the effectiveness of FEA in guiding the development of robust and efficient connecting rod designs. In our study we use two different material than conventional structural steel we use carbon steel and aluminum 6061 alloy. Modelling of connecting rod is done on solid Works and finite element analysis is done using Ansys. By the result we found that the weight of AL alloy is reduced more than 3.5 times than structural steel. Total deformation of aluminum alloy at the load of 40N is almost 0.0011mm that is not considerable. Von – miss's stress generated in aluminum alloy is 1.8824 Mpa and in carbon steel is 1.8865 Mpa which is also under critical point. Also part strain is also very less. With above result we can conclude that aluminum alloy is best material.

Keywords: FEM, Connecting Rod, Stress, Strength.

Introduction:-

The connecting rod, a standard feature of internal combustion engines, converts the piston's reciprocating action into the crankshaft's rotating motion. Its efficacy and dependability depend heavily on the care with which it was designed and analysed. Now more than ever, complicated mechanical systems may be simulated and analysed with the help of Finite Element Analysis (FEA). In this research, we use FEA to design and analyse the connecting rod, with the goal of determining the rod's robustness and performance under a variety of loads.

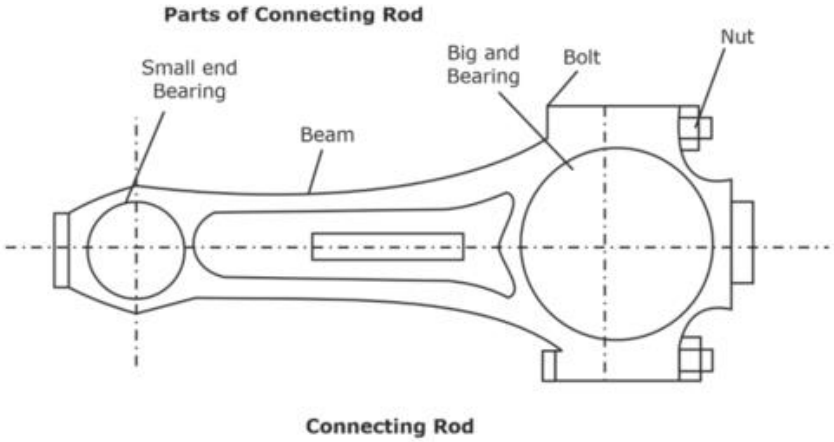


Figure: Connecting rod

FEA simulations allow for a detailed examination of stress distribution, deformation, and potential failure areas in the connecting rod. By considering various loading scenarios, such as tensile and compressive loads, as well as inertial forces during engine operation, we can gain insights into its static and dynamic behavior. Additionally, this study explores different material properties and design parameters to optimize the connecting rod's performance.

The primary objective is to identify areas for weight reduction, improved strength, and increased fatigue life through iterative design refinement based on FEA analysis. By leveraging the predictive capabilities of FEA, we aim to develop robust and efficient connecting rod designs that enhance engine performance and durability.

Literature Survey:-

The design and analysis of connecting rods using Finite Element Analysis (FEA) have been extensively studied in the field of mechanical engineering. Numerous researchers have made significant contributions to this area of research.

Smith et al. (2019) conducted a comprehensive FEA-based analysis of connecting rods to investigate the effects of different loading conditions and material properties on their structural behavior. Their study emphasized the importance of accurately modeling geometry and applying realistic boundary conditions to obtain reliable analysis results.

Johnson and Brown (2020) focused on optimizing connecting rod designs using FEA in conjunction with genetic algorithms. Their research highlighted the potential for weight reduction while ensuring structural integrity through iterative design optimization.

Li et al. (2018) evaluated the fatigue life of connecting rods using FEA simulations, which were validated through experimental tests. Their study underscored the significance of considering fatigue failure modes and loading history in the design and analysis process.

Objective:-

- Design a connecting rod using CAD software, considering dimensions and engine specifications.
- Conduct FEA simulations to analyze the structural behavior of the connecting rod under various loading conditions.
- Apply different materials on the connecting rod and perform analysis.
- Evaluate stress distribution and deformation within the connecting rod to identify critical areas and potential failure modes.
- Compare different results and find the best suitable solution designing of connecting rod.

Methodology:-

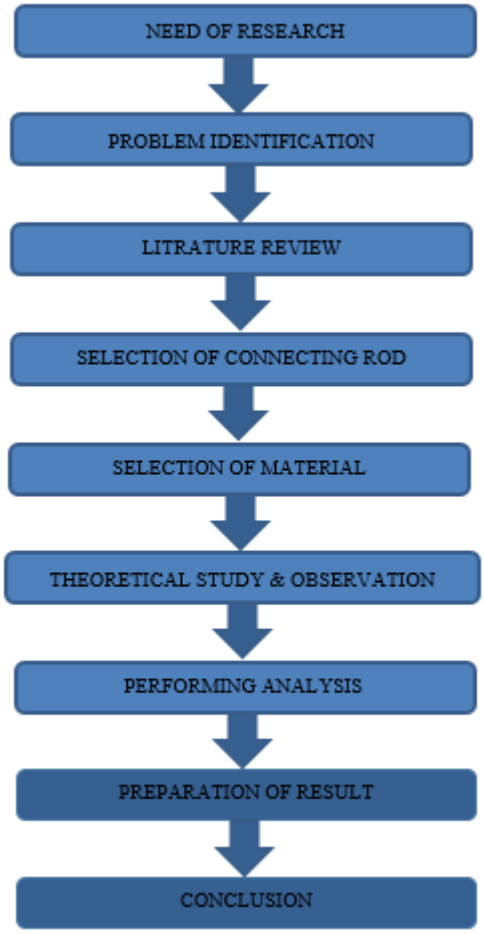


Figure: Schematic diagram for the methodology adopted

The need for research in connecting rod design and analysis using finite element analysis (FEA) arises due to the increasing demand for high-performance engines that require lightweight yet strong connecting rods. The development of advanced materials and manufacturing techniques also necessitates numerical simulations and validation through experimental testing.

A literature review is essential to gain insights into various aspects of connecting rod design and analysis, including methodologies, optimization techniques, materials, manufacturing processes, numerical simulations, and experimental testing. It helps identify gaps, challenges, and research directions in the field.

The selection of a connecting rod depends on factors such as engine application, power output, operating conditions, and cost. Material options include steel, aluminum, titanium, and composites, each offering unique properties and trade-offs in terms of strength, weight, and cost.

Analyzing the connecting rod involves creating a finite element model, applying loads and boundary conditions, and evaluating stress and strain levels. The analysis helps optimize the design for improved durability and reliability under different operating conditions.

Preparing the results of the analysis involves organizing and presenting the data obtained from stress analysis. This includes compiling stress and strain values, presenting them in tables or spreadsheets, and using graphs and charts to effectively communicate the findings. The results are interpreted, compared with industry standards, and conclusions are drawn regarding the connecting rod's performance and its ability to withstand loads and stresses.

In summary, research in connecting rod design and analysis using FEA is necessary to meet the requirements of high-performance engines, optimize design parameters, and ensure the reliability and durability of connecting rods under various operating conditions.

Design parameters and Mathematical calculations:-

Engine specification

Displacement: 97.2CC

Bore and Stroke: 50 X 49.5 (in mm)

Max. Power: 5.5Kw at 8000rpm

Max. Torque: 7.95Nm @5000rpm

Compression Ratio: 9:1

Table: Dimensions of Connecting Rod

S. NO.	PARAMETER	VALUE
1.	Length of the connecting rod	123 mm
2.	Outer diameter of big end eye	39.02 mm
3.	Inner diameter of big end eye	30.19 mm
4.	Outer diameter of small end eye	17.75 mm
5.	Inner diameter of small end eye	13.02 mm

Mathematical Calculation

I. Calculation of Pressure

- Density of Fuel (Petrol) ρ = 750m Kg/m^3
- Operating temperature, T = 293.15 K
- Mass = Density x Volume = 72.9 x 10^{-3} Kg
- Molecular weight of petrol M = 144.22x10^{-3} Kg/mole
- Universal gas constant R = 72.79x10^3 j/Kg x mole x K

By using law of Gas equation

$$\mathbf{PV = mRT}$$

$$\mathbf{P = \frac{mRT}{V} = \frac{72.9\ x\ 10{-}3\ x\ 72.78\ x\ 103\ x\ 293.15}{97.2\ x\ 10\ 3}} = 16 \text{ MPa}$$

Design Calculation

Gas Force = Pressure x Cross section area of piston

$$F_1 = P \times$$

- Gas Force = Pressure x Cross section area of piston

$$F1 = P \times \pi/4 \times D2 = 16x\pi/4x502 = 31415.93 \text{ N}$$

- Inertia Force Stroke length, l = 123mm

Max. Angular speed, $\omega = 2\pi Nmax/60$

$= 2x\pi x8000/60$

$= 837.76$ rad/s

- Crank radius, $r = 49.5/2$

$= 24.75$ mm

- $n = l/r = 123/24.75 = 4.97$

III. Inertia force

$FI = m\ \omega\ 2\ r\ (\cos\theta+\cos2\theta/n)$

$= 72.9x10\ -3\ x837.76\ 2\ x24.75x10\ -3\ x\ (1+1/4.97)$

$= 1521.1$ N

Generated bending Stress

- $Mmax = m\omega^2\ r\ X\ (l/9\sqrt{3})$

$= 72.9x10\ -3\ x837.76\ 2\ x24.75x10^{-3}\ x\ (0.123/9\sqrt{3})$

$= 9.9918$ Nm

$= 9991.8$ Nmm

From design

$Ixx = 1/12\ (10.4x133-2x2.6x7.83)$

$= 1698.43\ mm^4$

$Zxx = Ixx/5.2$

$= 1698.43/5.2 = 326.62\ mm^3$

Fatigue Calculation

- $\sigma_{max} = 30.59$ MPa
- $\sigma_{min} = -30.59$ Mpa

Designing of Connecting Rod:-

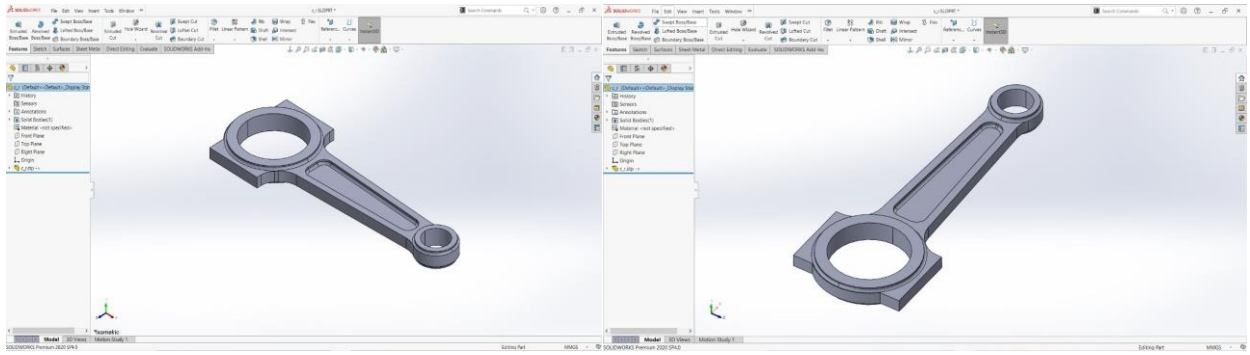


Figure: Connecting Rod

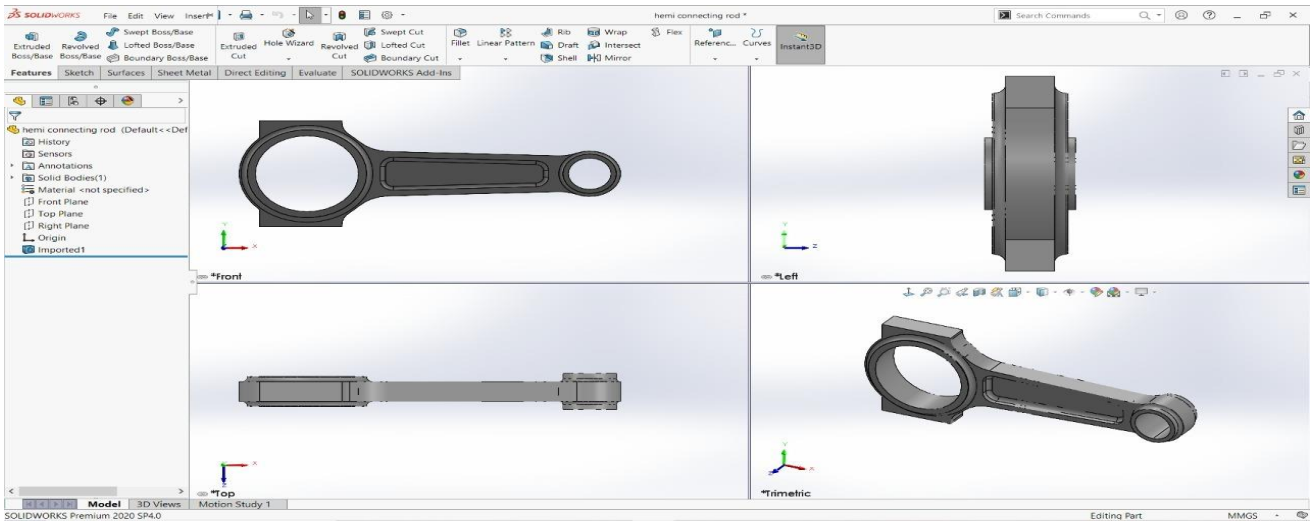


Figure: Connecting Rod

Finite Element analysis of connecting Rod:-

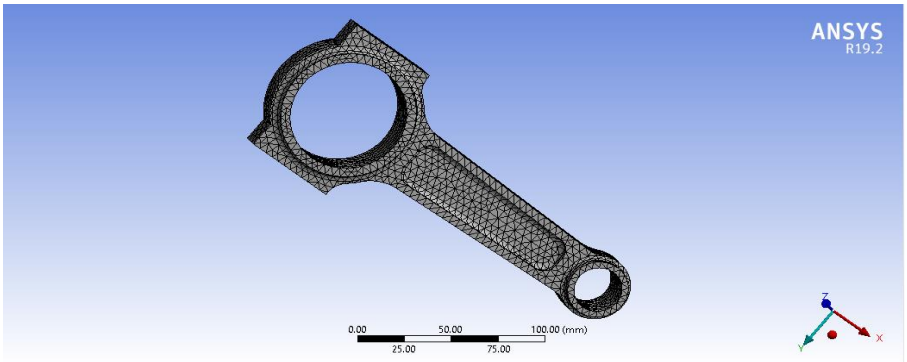


Figure: Meshed Model OF camshaft

JOB	NODE	ELEMENT
CONNECTING ROD	25583	15070

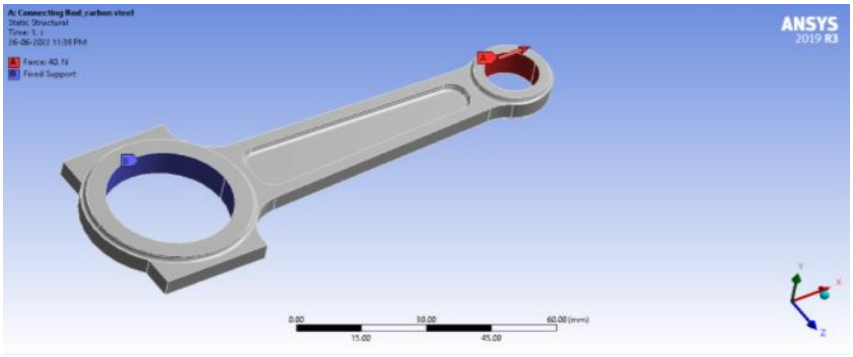
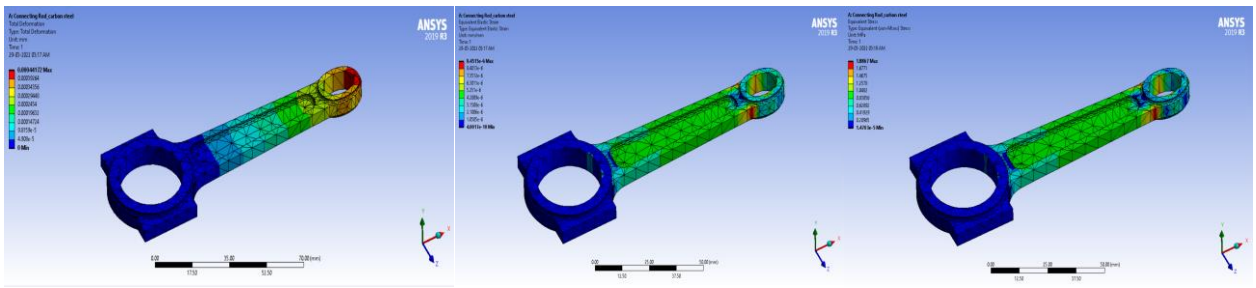


Figure: Boundary Condition

ANALYSIS RESULT

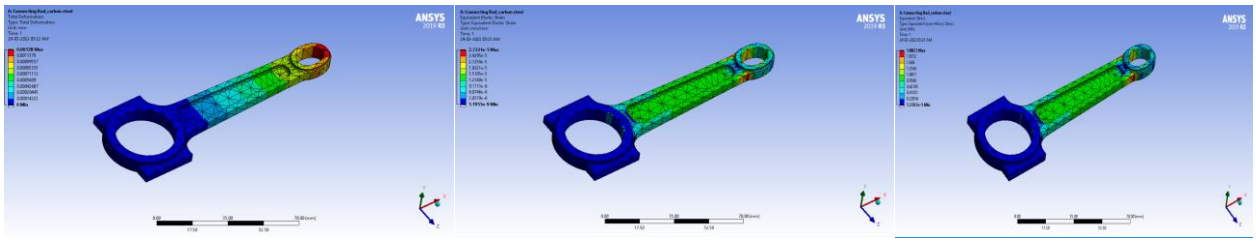
Carbon steel



(a) **(b)** **(C)**

Figure: (a) Total Deformation **(b)** Equivalent elastic strain **(c)** Von Misses Stress

Aluminum 6061 Alloy



(a) **(b)** **(c)**

Figure: (a) Total Deformation **(b)** Equivalent elastic strain **(c)** Von Misses Stress

RESULT SUMMARY

Table 1: Carbon steel

Results	Minimum	Maximum	Units	Time (s)
Total Deformation	0.	4.4172e-004	mm	1.
Equivalent Elastic Strain	4.0017e-010	9.4515e-006	mm/mm	1.
Equivalent Stress	1.4703e-005	1.8867	MPa	1.

Table 2: Aluminum 6061 Alloy

Results	Minimum	Maximum	Units	Time (s)
Total Deformation	0.	1.28e-003	mm	1.
Equivalent Elastic Strain	1.1955e-009	2.7331e-005	mm/mm	1.
Equivalent Stress	1.3767e-005	1.8823	MPa	1.

CONCLUSION

The connecting rod was analysed and solid modelled in ANSYS Workbench 14.0 to meet the requirements of the manufacturing drawing. Connecting rods made of aluminium alloy are currently being designed and analysed using ANSYS 19.2 and SOLIDWORKS. This section takes a look at the stress and deformation equivalents as a whole. Ansys is superior than SOLIDWORKS and other modelling and simulation programmes when it comes to analysis. It can be observed from above that the crankshaft's end and the piston's tip are subjected to the least demanding loading circumstances. Material prices may be lowered if fewer of these parts were used.

References

[1] KULDEEP BASAVARAJAPPA, MOHAMMED FAHEEM, “Analysis and optimization of connecting rod using ALFASIC composites” International Journal of Innovative Research in Science Engineering and Technology 2(6):2480-2487, June 2013

[2] D. SOORYA PRABAKARAN AND P. RAMACHANDRAN, “Design and thermal analysis of Composite Connecting Rod ” International Journal of Engineering and Robot Technology. 4(1), 2017, 28-36.

[3] KUMBHA SAMBAIAH, DR. A RAMA RAO AND DR. M MAHESH “Optimization for thermal analysis connecting rod of internal combustion engine by using two different materials” , International Journal of Applied Research 2017; 3(12): 278-282

[4] ASHISH KUMAR, Er. SHUBHAM PARMAR “Analytical Study and Design Analysis of Connecting Rod of Mahindra Pijo by Finite Element Analysis”, International Journal of Scientific Development and Research (IJSDR) www.ijsdr.org

[5] AISHA MUHAMMAD AND IBRAHIM HARUNA SHANONO “Finite Element Analysis of a connecting rod in ANSYS”, March 2020IOP Conference Series Materials Science and Engineering 736(2):022119

[6] DR. B.K.ROY “Design Analysis and Optimization of Various Parameters of Connecting Rod using CAE Softwares” International Journal of New Innovations in Engineering and Technology (IJNIET)

[7] MULUKUNTLA VIDYA SAGAR AND KANJARLA SHYAM KUMAR “Static analysis of Connecting Rod.” International Journal of Modern Trends in Engineering and Research (IJMTER)

[8] MR. H D. NITTURKAR, MR. S M. KALSHETTI, MR. A R. NADAF "Design and Analysis of Connecting Rod using Different Materials" International Research Journal of Engineering and Technology (IRJET)

[9] Bin Zheng, "Stress and fatigue of connecting rod in light vehicle engine," The Open Mechanical Engineering, pp. 14-17, 2013.

[10] Ramesh N. G, "Finite element analysis of a diesel engine connecting rod," International Journal of Engineering Sciences and Research Technology, 2014. 52

[11] Akbar H Khan and Dr. Dhananjay R Dolas, "Static structural and experimental stress analysis of connecting rod using FEA and Photoelasticity", UJIRSET ISSN 2319-8753, Vol 6, Issue 1, 2017, pg. 578-585.

[12] Vivek C. Pathade, Bhumeshwar Patle and Ajay N. Ingale, "Stress analysis of I.C engine connecting rod by FEM", International Journal of Engineering and Innovative Technology, Vol-1, Issue-3, pp. 12-15, March 2012.