International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

Modelling and Performance of Engine Connecting Rod Using Ansys

Anup Dubey[1], Dr. Mayank Ladha[2], Dr. Rahul Joshi[3]

[1]Research Scholar, Dept. of Mechanical Engineering, Swami Vivekanand College of engineering, Indore (M.P.)
[2]Asscociate prof., Dept. of Mechanical Engineering, Swami Vivekanand College of engineering, Indore (M.P.)
[3] Asscociate prof., Dept. of Mechanical Engineering, Swami Vivekanand College of engineering, Indore (M.P.)

ABSTRACT

This study focuses on the modeling and performance analysis of an engine connecting rod using Ansys, a powerful engineering simulation software. The connecting rod plays a crucial role in transmitting the reciprocating motion from the piston to the crankshaft, and its structural integrity is essential for efficient engine operation.To assess the performance of the connecting rod, a detailed three-dimensional model is developed using Ansys, considering material properties, geometrical dimensions, and loading conditions. The Finite Element Method (FEM) is employed to simulate the behaviour of the connecting rod under various operational conditions, such as varying loads, rotational speeds, and thermal effects. The objectives of this study include evaluating the stress distribution, deformation, and fatigue life of the connecting rod. By analyzing these factors, potential areas of weakness or failure can be identified, leading to design improvements and enhanced performance. Moreover, the effect of different materials and design modifications on the connecting rod's performance is also investigated.

INTRODUCTION

The connecting rod is a critical component in an internal combustion engine, playing a vital role in converting the reciprocating motion of the piston into rotational motion at the crankshaft. It is subjected to high loads, dynamic forces, and thermal variations during engine operation. Therefore, ensuring the structural integrity and performance of the connecting rod is crucial for the overall efficiency, reliability, and durability of the engine. Traditionally, the design and analysis of connecting rods have relied on empirical approaches, prototype testing, and costly trial-and-error methods. However, with the advancements in computer-aided engineering (CAE) tools, such as Ansys, engineers now have the capability to accurately model and simulate the behaviour of the connecting rod under various operating conditions. Ansys is a widely used software package that employs the Finite Element Method (FEM) to analyze and predict the structural response of complex mechanical systems. By creating a virtual model of the connecting rod and applying appropriate material properties, geometrical dimensions, and boundary conditions, Ansys enables engineers to evaluate the stress distribution, deformation, and fatigue life of the connecting rod accurately. The objective of this study is to utilize Ansys to model and analyze the performance of an engine connecting rod. By conducting virtual simulations, we can gain valuable insights into the structural behavior and performance characteristics of the connecting rod. These insights can aid in identifying potential areas of weakness or failure and guide design improvements to enhance the overall performance and reliability of the connecting rod. Through this research, we aim to contribute to the field of engine design and optimization by providing a comprehensive understanding of the behavior and performance of the connecting rod. The findings can support engineers in making informed decisions regarding material selection, design modifications, and operational conditions to maximize the efficiency and durability of the engine.

LITERATURE REVIEW

Aditya A. Lotakeet. Al. (2019) In this demonstration, the two-wheeler rod is created in CATIA v5 R21. Following connecting rod modeling, it is made available for ANSYS analysis. The connecting rod is studied statically in ANSYS Workbench. Cast iron, cotton alloy, anisotropic silicon, structural steel, and titanium alloy are among the materials analyzed. The foregoing analysis demonstrates that titanium alloy is the optimal construction material for a connecting rod that features two rollers. A static study of connecting rods for two-wheeled vehicles is performed for a variety of materials, including cast iron, copper alloy, silicone, structural steel, and titanium alloy. Based on the results of the above analysis, it can be concluded that the titanium alloy generates significantly less stress (1.4631e8 Pa) than the aforementioned materials. Therefore, the titanium alloy is ideal for the bicycle's connecting rod.

Arun Kumar et. al. (2019) The project's objective is to compare and contrast connecting rods made from metal and composites. The connected rods are frequently employed in internal combustion engines, where they endure a wide variety of stress cycles that eventually lead to fatigue failure. In contrast, this technique involves carving a sculpture out of a rod made of either conventional material (Al7175-T66,C70S6,AISI4140, and TI-6AL-4V) or a composite material (LM 25 aluminum MMC, KELER CNT MMC, CVI-C / SIC, and 25Si3N4-MG MMC). CATIA V5 computer code is used most often for designing rods, and NX NASTRAN is used most often for simulating rods made from any typical composite material. When it comes to stress, strain, and displacement, we provide the most accessible material possible.

Tara Mohan et. al. (2019) In this research, an analysis technique was used to develop the IC rod's design. On this basis, a physical model is created in CATIA V5. The connecting wire's structural study was performed using FEA. FEA ANSYS WORKBENCH Software 14.5 is modified by applying various stresses to account for optimal loading conditions. Research into the mechanical and thermal properties of several materials, including C70 steel and ALSIC. The proper material for roll contact was determined after analyzing the data (stresses, shear stresses, total deformation, strain, distribution of heat, heat flux) and comparing them to various performances.

BogaSudhaet. al. (2018) In this investigation, carbon steel and aluminum alloy are contrasted to determine which is superior. In 3D modeling software, the connecting rod is represented by a Solid object. These thermal analysis designs then follow. Ansys is used to conduct this kind of analysis. The heat flow values are obtained using thermal analysis, allowing us to select the optimal connection rod material. The heat flux of two materials is compared when they are utilized in a connecting rod analysis at different temperatures. According to the data shown above, carbon steel is superior to aluminum alloy in terms of strength.

Nageswara Rao et. al. (2018) The primary focus is on bettering gas/gasoline engine parts. In the first study, a Finite Element Analysis (Structural) was performed after a 3D model of the piston, connecting rod, and crankshaft was created in solid works. The Yamaha FZ-16's optimal installation of engine components was matched by the results of a structural FEA. Finally, our weight studies showed that we could minimize the optimal assembly of engine parts by 13.76 percent.

Achyut Chauhan et. al. (2017) Research is being conducted to determine the most effective material for rod attachment, with the goal of improving connection rod performance in terms of reduced weight, stress, strain, displacement, and capacity. The connecting rod can be dissected for structural analysis and operational efficacy evaluation. Multiple ANSYS simulations showed how much pressure builds up in a connecting rod under stress; researchers also discovered that the mass of connecting rods can be decreased and that they have a place in modern machinery.

Bodige Mahesh et. al. (2017) The purpose of this research is to analyze the load, strain, and stress on the crank end of the connecting rod using multiple materials. High-resistance Carbon Fiber connecting wire will be paired with a connector rod fabricated from stainless steel and aluminum alloy. The findings can be used to improve the connecting rod's design and further slim it down in terms of weight. For modeling and analysis, ANSYS employs Pro-E software. The saved data can also show us places where failure is more likely to occur as a result of pressure. The findings can also be utilized to improve upon and extend the useful life of already existing designs.

METHODOLOGY

A wide variety of engineering problems can be numerically solved using the finite element approach. The method's inherent flexibility makes it applicable to a wide variety of complicated geometries and materials, as well as a wide variety of boundary and loading circumstances. Since closed form solutions of governing equilibrium equations are not typically available, the finite element approach is well suited to the analysis needs of modern complex engineering systems and designs. It is also an effective design tool for performing parametric design studies, where designers analyze multiple design situations (alternative shapes, materials, loads, etc.) before deciding on the best design.

This technique was developed in the aerospace industry to analyze strain in intricate aircraft structures. It develops from the so-called matrix analysis technique previously employed in aeronautical engineering. There has been a rise in the method's popularity among academics and professionals alike. The idea behind the finite element method is that larger bodies or structures can be broken down into smaller, more manageable pieces. Assembling these parts together at a finite number of sites they call nodes or nodal points, we can think about the original body or structure.

General procedure of finite element method

The finite element method is a piecewise approximation technique in which the structure or body is divided into small elements of finite dimensions called finite elements and then the original body or structure is considered as an assemblage of these elements connected at a finite number of joints called nodal points or nodes. The variation of a field variable within a finite element can be estimated by a simple function because the real variation of field variables such as displacement, stress, temperature, pressure, or velocity inside the continuum is unknown. The values of the nodes' field variables are used to define the approximation functions, which are known as interpolation models. By solving the field equations, which typically take the form of matrices, we may determine the values of the field variable at each node.

Once the nodal values are known, the field variable is defined globally across the ensemble by the approximation functions.

The finite element approach provides a systematic, repeatable procedure for solving general continuum issues. Following is a detailed description of how to solve a static structural problem:

Step 1:- Description of Structure (Domain). In the finite element approach, the first step is to break the solution region's structure up into smaller pieces, called "elements".

Step 2:- Choosing the right interpolation model. Assuming some suitable solution, within an element, to approximate the unknown solution is necessary because the displacement (field variable) solution of a complex structure under any specified load conditions cannot be predicted exactly. The solution assumed should be straightforward and meet specific convergence criteria.

Step 3:- The characteristic matrices and load vectors for an element are derived. It is expected that equilibrium conditions or a suitable Variation principle will be used to obtain the stiffness matrix [K(e)] and the load vector P(e) of element 'e' from the anticipated displacement model.

Step 4:- Assemblage of element equations to obtain the equilibrium equations.

Since the structure is made up of numerous finite elements, the stiffness matrices and load vectors for each element must be built in an appropriate fashion, and the overall equilibrium equation must be expressed as follows:-

$[K]\varphi = P$

Where [K] is called assembled stiffness matrix, Φ is called the vector of nodal displacement and P is the vector or nodal force for the complete structure.

Step 5:- Finding the displacement (the field variable) at the nodes requires solving a system equation. Boundary circumstances necessitate adjustments to the overall equilibrium equations. When boundary conditions are included, the equilibrium equations become,

$[K]\varphi = P$

For linear problems, the vector 'φ' can be solved very easily. But for non-linear problems, the solution has to be obtained in a sequence of steps, each step involving the modification of the stiffness matrix [K] and 'φ' or the load vector P.

Step 6:- Strain and stress elements are computed. Using the appropriate equations of solid or structural mechanics, the element strains and stresses can be calculated from the known nodal displacements. Words in parentheses above indicate the overall FEM step-by-step approach that is being implemented.

RESULTS AD DISCUSSION

The FEA model is subjected to a wide range of boundary conditions and loads. The entire heat flux field, or temperature field, can be measured. Stress and strain in the connecting rod were analyzed statically as well. Connecting rod thermal analysis in ANSYS is performed by assigning the Material characteristics of model to one of four different materials: aluminum alloy, structural steel, magnesium alloy, or titanium alloy. One way to determine how a material's physical properties change with temperature is through thermal analysis. Methods that calculate the mass or energy changes of a material model are the most widely employed. Figures illustrating the effects of boundary conditions and temperature gradients across connecting rod materials at steady state are provided. Top on the scale for heat. Total thermal stress and equivalent stress in static structural analysis are shown along contours once the solution is processed. In thermal analysis, the two most important variables are temperature and total heat flux. These findings are achieved for each material studied in order to optimize connecting rod failure during the process by means of structural and thermal analysis.

Analysis of Connecting Rod with variations of materials

Piston and crank are connected through connecting rod. It converts the piston's reciprocating motion into the crank's rotary action by transmitting the thrust from the piston pin to the crank pin. While carbon steel has traditionally been employed, aluminum alloys have become increasingly used in the production of connecting rods in recent years.

The Connecting rod model being studied is broken down into a grid of discrete, measurable building blocks. Basic polynomial profile capacity and nodal Temperature are assumed to be used to calculate the difference in displacement inside each segment. Strain and stress conditions are established all the way to the unknown nodal temperature. The resulting grid-like collection of equilibrium conditions is highly flexible. Boundary conditions and steady-state temperature differences for several connecting-rod materials are depicted in the images below. The connecting rod is subjected to the combustion process's maximum temperature. In thermal analysis, we compared the temperatures and total heat flux of the processed solution to those of Al 360, structural steel, magnesium alloy, and titanium alloy. All conditions, including differences in material and loading on the big end and the small end of the connecting rod, are accounted for in these findings from the structural and thermal study.

Thermal analysis in ANSYS of Connecting rod

Time-dependent temperature and other thermal quantities can be calculated via thermal analysis. In quenching analysis for heat treatment, for example, understanding how temperatures fluctuate over time is crucial. The thermal stresses that can lead to failure as a result of temperature dispersion are also of interest.

- **Temperature distribution analysis of Connecting Rod Models**

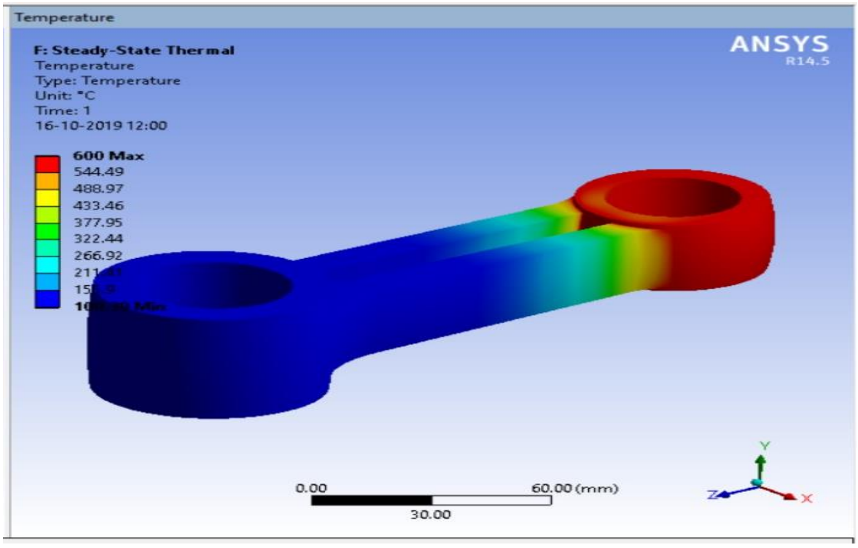

Figure 1: Temperature Distribution of Connecting rod with Aluminium Al 360

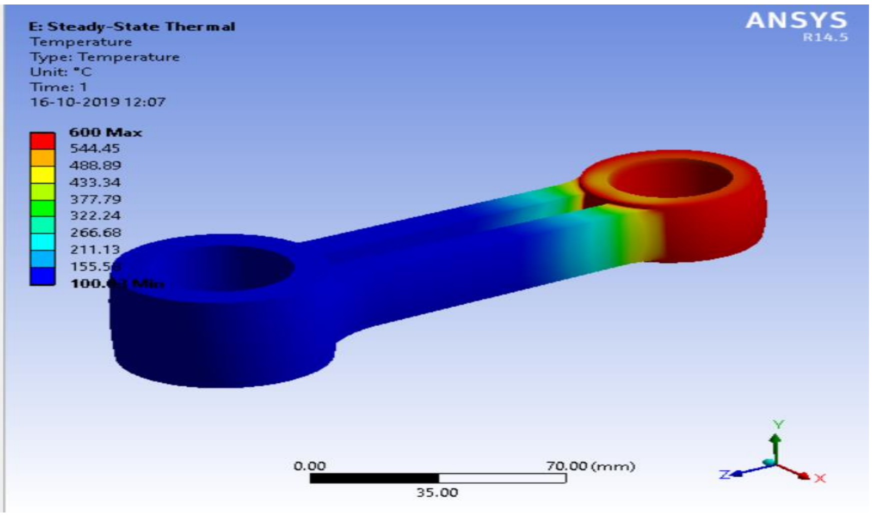

Figure 2: Temperature Distribution of Connecting rod with Structural Steel material

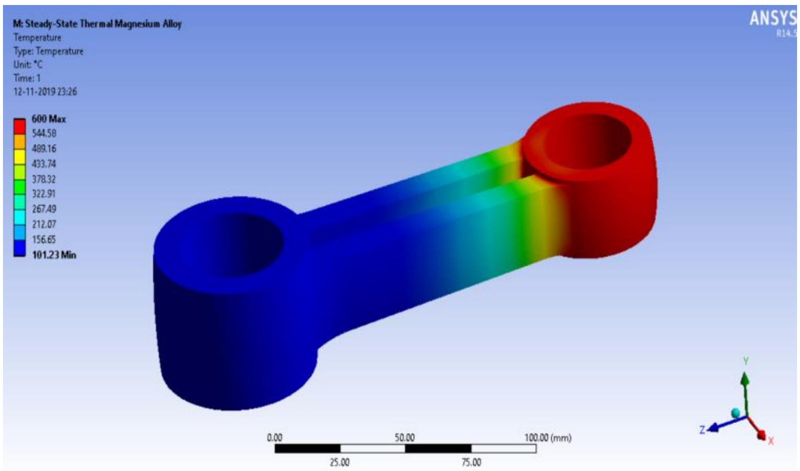

Figure 3: Temperature Distribution of Connecting rod with Magnesium alloy Material

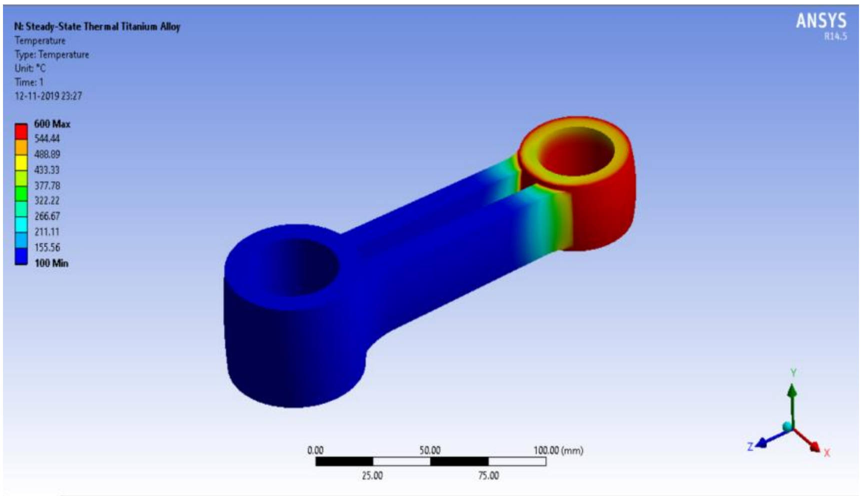

Figure 4: Temperature Distribution of Connecting rod with Titanium alloy Material

- **Heat flux distribution analysis of Connecting Rod Models**

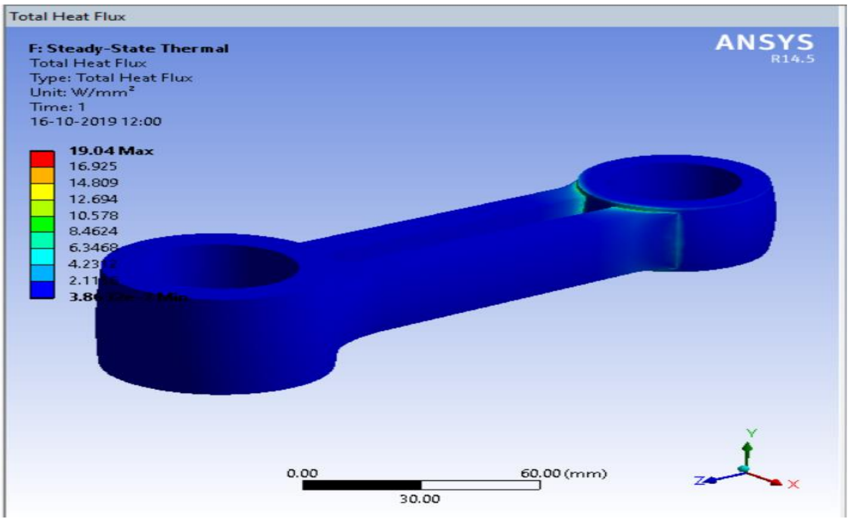

Figure 5: Temperature Distribution of Connecting rod with Aluminium Al 360

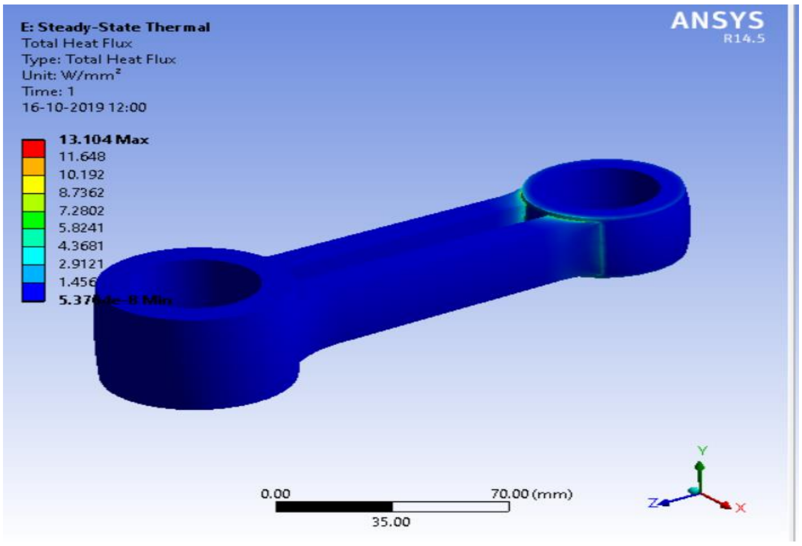

Figure 6: Temperature Distribution of Connecting rod with Structural Steel material

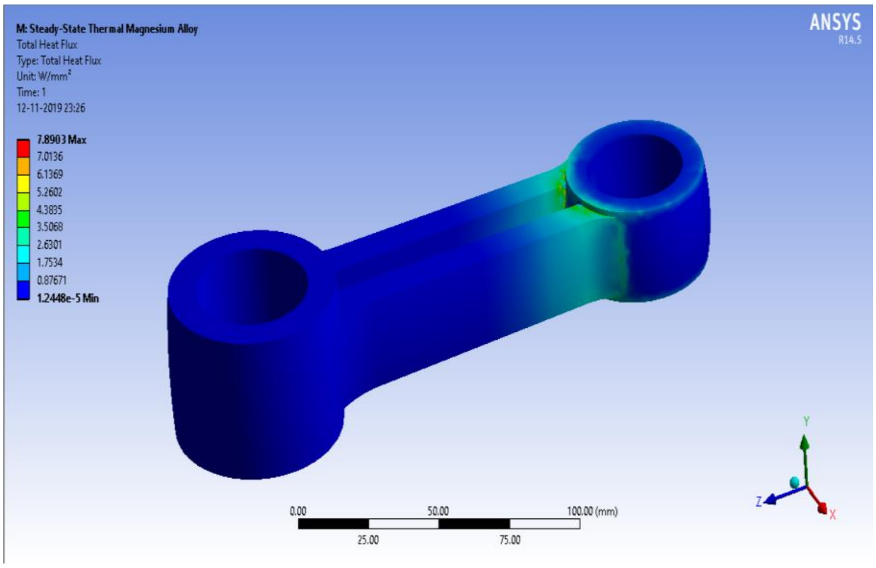

Figure 7: Temperature Distribution of Connecting rod with magnesium alloy Material

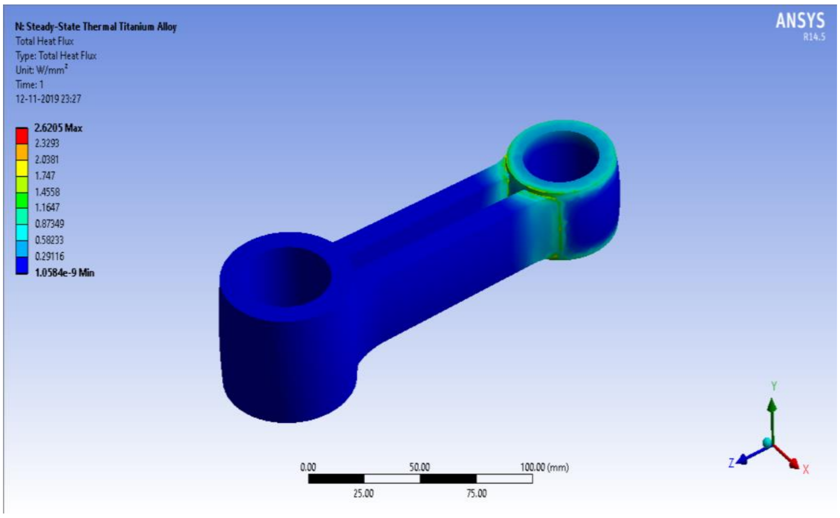

Figure 8: Temperature Distribution of Connecting rod with Titanium alloy Material

➢ **Structural analysis of Connecting Rod Models**

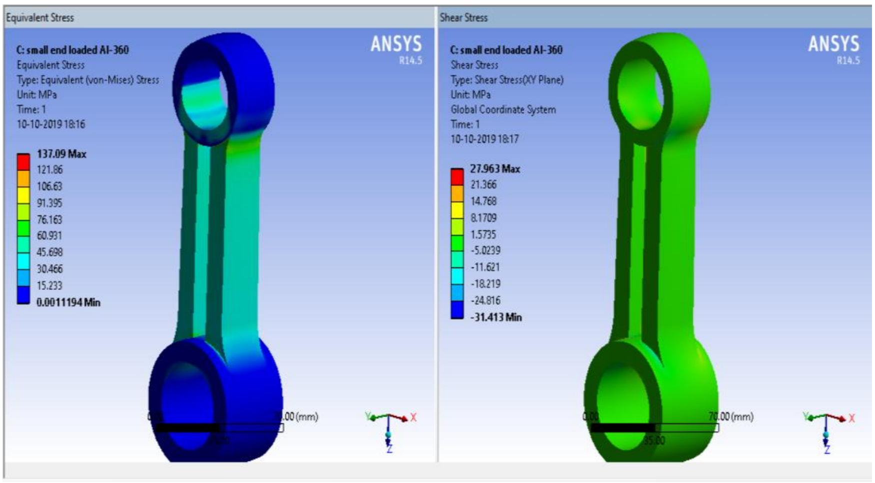

Figure 9: Maximum Stress found on Connecting rod small end of Aluminum Al360

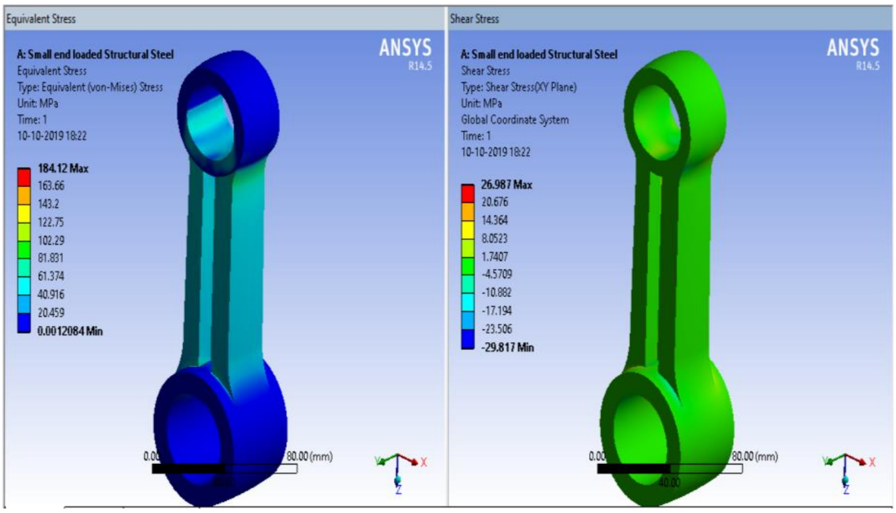

Figure 10: Maximum Stress found on Connecting rod small end of Structural Steel

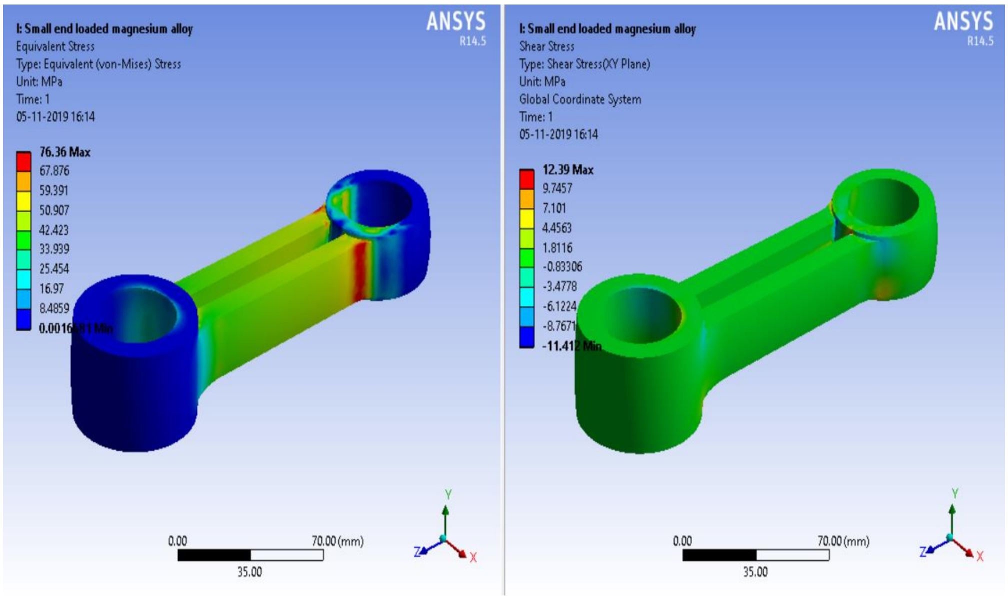

Figure 11: Maximum Stress found on Connecting rod small end of Magnesium alloy

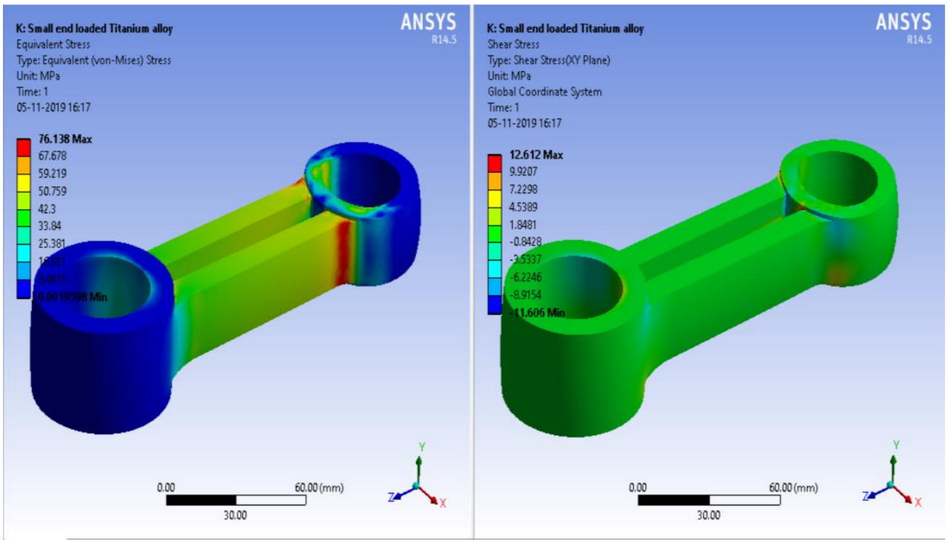

Figure 12: Maximum Stress found on Connecting rod small end of Titanium alloy

CONCLUSIONS

The current research took into account the possibility of building an engine without first modelling its thermal environment, specifically the Connecting rod and its support.The model for the planned model has been completed using Solidwork. We used Ansys to do a steady-state thermal analysis under both closed and open conditions, with a connecting rod made of a single metal. Thermal analysis has also been performed by us.Design and study of connecting rods made from various materials, due to their crucial significance in the I. C. engine. Solidwork was used to design the connecting rods used in this project. Ansys workbench was used to simulate the stresses, shear stresses, temperature distributions, and heat fluxes of an IC engine connecting rod made from a variety of materials, including structural steel, magnesium alloy, Titanium alloy, and Al360, for a 180cc engine. Alloy of magnesium. Therefore, our analysis concludes that connecting rods made from magnesium alloy have superior thermal characteristics. Connecting rods made from structural steel, on the other hand, display reduced heat flux and higher strains during the testing procedure, leading to engine failure. ANSYS 14, a finite element analysis (FEM) program, is used for the overall analysis. This allows for the use of cutting-edge materials and a variety of design and analysis methods in future studies. The following inferences are drawn from the aforementioned body of research:

- Connecting rod thermal analysis by varying material type, operating temperature, and load has been performed.
- As per above comparison graph it is concluded that connecting rod with structural steel having lower heat flux rate that is 4.49 w/m^2 and connecting rod with magnesium alloy having maximum heat flux 7.89 w/m^2.
- When compared to other materials, the magnesium alloy used in connecting rods is noticeably lighter.
- Maximum stress found in connecting rod with material Structural steel is 184.12 MPa.
- Using an engine cylinder with a connecting rod made of magnesium alloy is preferable to using a connecting rod made of structural steel or other specified materials, as shown by the findings of the analysis.

REFERENCES

1. Aditya A. Lotake, Shakir M. Mulani, Sohel M. Mulani, "Comparative Stress Analysis of Connecting rod using ANSYS for Different Materials", International Journal of New Technology and Research, Volume-5, Issue-4, April 2019.
2. Arunkumar, Rama krishna, Maboobbasha, "modelling and simulation of connecting rod by conventional and composite (mmc) materials", International Journal of Technical Innovation in Modern Engineering & Science (IJTIMES), Volume 5, Issue 07, July-2019.
3. M Frăţiţa, F Popescu, K Uzuneanu, "About Structural and Thermal Analysis of Diesel Engine Piston Using Ansys Software", National Conference on Thermodynamics with International Participation, Materials Science and Engineering, 595, 2019.
4. Tara Mohan, G. Veeresh Kumar, B.RajaNarender, "Structural and Thermal Analysis of IC Engine Connecting Rod Using Ansys", International Journal of Modern Engineering and Research Technology, Volume 6, Issue 1, January 2019.
5. BogaSudha, I Satyanarayana, Sivakandhan, "Design and Thermal Analysis of IC Engine's Connecting Rod for Different Heat Conditions", International Journal of Research in Advent Technology, Vol.6, No.11, November 2018.
6. Nageswara Rao, K. Vasudev Rao, V. Ananta Rahul, "Design, Analysis and Optimization of Gas/petrol Engine Components", International Journal of Latest Technology in Engineering, Management & Applied Science, Volume VII, Issue III, March 2018.
7. Achyut Chauhan, HevalSaxena, Niraj Mehta, "A Review on FEA of Connecting Rod of I.C.Engine", IJIRSET, 3712-3716.
8. Bodige Mahesh, NareshBabu, "Thermal Analysis and Performance of a Connecting Rod with Different Materials", International Journal of Applied Research in Science and Engineering, Volume 2, Issue 2, June-2017.
9. D. SooryaPrabakaran, P. Ramachandran, "Design and Thermal Analysis of Composite Connecting Rod", International Journal of Engineering and Robot Technology, 4(1), 2017, 28-36.
10. G Gopala, Dr L Suresh Kumarb, K VijayaBahskar Reddy, "Analysis of Piston, Connecting rod and Crank shaft assembly", Materials Today: Proceedings, Elsevier, 4, 2017, 7810–7819.
11. KumbhaSambaiah, A Rama Rao, 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.
12. M. Zgoul, j. Yamen, Eng. Yazan Yousef, "Analysis of Mechanical and Thermal Stresses on Connecting Rod/Piston of A Variable Compression Ratio Engine Powered by Biodiesel", International Journal of Mechanical and Production Engineering, Volume- 5, Issue-10, Oct.-2017.
13. Raj kumar, NishantSaxsena, Arvind Singh, "Finite Element Analysis of Connecting rod for Internal Combustion Engines", International Research Journal of Engineering and Technology (IRJET), Volume: 04, Issue: 02, Feb -2017.