© January 2024| IJIRT | Volume 10 Issue 8 | ISSN: 2349-6002

Structural and Thermal Analysis of Piston, Connecting rod and Crank shaft assembly using Catia and Ansys Software

Ravi Ranjan[1], Prof. D. S. Rawat[2] and Dr. Sumit Rai[2]
[1]*ME IVth sem student, Department of Mechanical Engineering, Jabalpur Engineering College Jabalpur (M P), India*
[2]*Assistant professor, Department of Mechanical Engineering, Jabalpur Engineering College Jabalpur (M P), India*

Abstract- The primary role of the piston is to change the volume of the combustion chamber in an internal combustion engine. The piston is in contact with the hot gases that come out burning. In addition to thermal stress, depending on the operating mode of the engine, the piston is required to change the incoming force. Therefore, the iron piston passes because of the high inertial force, and the lightweight aluminum pistons cannot work at high temperatures. Therefore, the analysis should include a rigid body analysis and a dynamic body analysis. So the power applied to the parts when the engine is moving should be calculated and this power is used to calculate the dynamic stress on the profit part i.e. connecting rod. It is proposed to modify two new sets of materials for building parts of the assembly and investigate the parameters by performing a dynamic, dynamic and thermal analysis, thermal analysis involves the analysis of heat flow, temperature distribution through the temperature of the surface of the piston. By looking at the results of both analyses, we can determine whether our designed piston is safe under the applied load conditions. So finally a comparative study is done by showing the different results of the analysis of pistons using different materials.

Keywords- Piston, Connecting rod, Crank shaft, Catia Ansys Software.

1. INTRODUCTION

The engine's electronic computer unit can process a large number of sensors. Turbo-charged petrol engine manufacturers have started using additional sensors. These are temperature, pressure and actuator sensors for precise engine control. To avoid reaching a high temperature in the piston crown that causes melting or damage, engine manufacturers install an exhaust gas temperature (EGT) sensor. When the temperature rises to 800°C, the electronic computer unit reduces the amount of fuel injected by limiting the maximum power of the vehicle. The sensor can detect engine malfunctions when the injectors are not working properly and the fuel doses are too high, to prevent accidental melting of the piston. The research work deals with the study of the assembly of piston, connecting rod and crank shaft of four-wheel gasoline engine. The parts of the assembly must be strong and the assembly must move as one machine. Therefore, the analysis should include a rigid body analysis and a dynamic body analysis. So the force acting on the parts when the engine is revving should be calculated and this force is used to calculate the dynamic stress on the part of interest i.e. the connecting rod.

It is proposed to replace two new sets of materials for assembly parts and check the parameters by performing static, dynamic and thermal analysis. A piston is a part of an internal combustion engine. Build a machine with the help of a connecting rod and a crankshaft called a crank mechanism. The main function of the piston is to convert the pressure produced by the burning air-oil mixture into force acting on the crankshaft. The piston contributes to the loss of heat produced during combustion from the cylinder walls. It ensures the sealing of the combustion chamber, prevents the leakage of gas from it and the entry of oil into the combustion chamber and directs the movement of the connecting rod. The piston also ensures continuous exchange of gases in the combustion chamber and thus produces a variable volume in the combustion chamber. The piston temperature fields for petrol and DME combustion separately are calculated using finite element analysis tools. The result shows that the temperature of the DME petrol engine decreases from top to bottom along the piston axis. The piston temperature of an engine burning DME fuel is increased compared to petrol burning.

ANSYS finite element code is used to complete the modeling process to determine the bond stress. It is made in two models with three dimensions. The first one is used to check the temperature distribution in the piston volume, and the second one is used to check the temperature. Stress distribution due to temperature and other factors. The result shows that the upper limit of the temperature is 4.3 °C and the thermal conductivity of the material increases with increasing temperature. Thermal stress is concentrated on the edges of the piston and depends on the type of material. The analysis is done to minimize the stress concentration at the end of the piston ie (piston head/crown and piston skirt and sleeve). The structural model of the piston will be developed using Computer Aided Design NX/CATIA software. In addition, finite element analysis was performed using ANSYS computer-aided simulation software.

The main parts of the assembly ie engine piston, connecting rod and crankshaft are modeled and assembled as per the given design. Also Finite Element Analysis is done in Ansys. Meshing is done in Hypermesh. Computer-aid design (CAD), also known as computer-aid design and drafting (CADD), is the use of computer technology in design and drafting. Ashish Kumar, Shubham Parmar, The main objective of this study is to analyze and optimize the Mahindra Piezo connecting rod. This study shows that the performance of the connecting rod depends fundamentally on the optimization of the shape and the choice of the material. The thickness of the existing connecting rod is measured with the help of a vernier calliper and a micrometer. The present study was carried out using ANSYS software to analyze the parameters of the connecting rod. The main purpose of this study is to investigate the stress caused by the connecting rod. This can be achieved by changing such design parameters in the existing design of a single cylinder 4 stroke gasoline engine by using FEA (Finite Element Analysis) in the study. During the analysis of the connecting rod parameters, it can be seen that there are several stresses acting during the loading condition of the rod.

Regarding Structural and Thermal Analysis of Petrol Engine Piston Using Ansys Software The primary function of a piston is to change the volume of the combustion chamber in an internal combustion engine. Therefore, steel pistons suffer from high inertial forces, and lightweight aluminum pistons cannot operate at high temperatures. The creative design of the bimetal piston was chosen to combine the advantages of two types of materials. Structural analysis involves applying pressure to the piston and analyzing its stress, strain and deformation. In thermal flux analysis, the temperature distribution is analyzed using the temperature on the surface of the piston. By looking at the results of both analyses, we can determine whether our designed piston is safe under the applied load conditions.

2. LITERATURE REVIEW

Singh et al. to analyze the tension load stress, total strain and factor of safety of the piston end of connecting rod of different materials. They replace the existing materials with beryllium alloy, magnesium alloy. FEA analysis is performed on 5 materials Al360, forged steel, beryllium alloy (alloy 25), titanium alloy, ti-13v-11cr-3al and magnesium alloy. In this they design a 3D model on SOLIDWORKS 2016 and the analysis is done by ANSYS 16.2 software. After analysis, they concluded that the designed and optimized connecting rod is used to replace the existing connecting rod because it is almost lighter 15%.

G Naga et al. Genetics explain the weight optimization in the connecting rods of IC engines by different materials such as steel, aluminium, titanium and cast iron. The model connecting rod is created on Pro-E and the analysis is performed on ANSYS. They perform various load analysis in static and tension analysis of connecting rod. Design optimization for suitable materials to minimize deflection. The load acting on the connecting rod as a function of time is obtained. The load yield relation for the connecting rod at a given constant speed of the crankshaft is also determined. They discovered that the connecting rod could be designed and optimized under tensile load as one peak load corresponding to 3600 crank angles at maximum engine speed and crank pressure as a second peak load resulting in cost reduction and weight reduction. There is a decrease. The bending stresses are calculated to be approximately 266.86333 N/mm^2 tensile bending stresses and it was also found that the connecting rod made of genetic steel shows less distortion and stress as compared to titanium, cast iron and aluminium.

G Shailaja and S Irfan Sadak investigated about static and modal analysis of connecting rods. They replace the material of the connecting rod with carbon steel and the aluminum alloy with beryllium alloy. They analyze the connecting rod to determine the dynamic behavior of the connecting rod by considering deformation, stress and strain. These parameters help to determine/identify a section of failure due to stress induced in the connecting rod. The model of the connecting rod is designed on SOLIDWORKS and the analysis is performed on FEA. They concluded that the section closer to the short end was more likely to fail due to the greater crushing load caused by the gudgeon pin assembly. Beryllium alloy connecting rods have the least maximum von-Mies stress, strain and maximum displacement, which is why beryllium connecting rods have a long life.

Sujal et al. Describe the design evaluation and optimization of connecting rod parameters. They change the design parameters in the existing design to obtain a suitable design for the connecting rod. They take single cylinder 4-stroke petrol engine. They perform structural analysis of the connecting rod. The connecting rod is designed by PRO-E Wildfire and analyzed on FEA software. Static analysis is performed to determine. The results of the analysis are used to determine fatigue strength, fatigue life, damage, factor of safety, stress biaxiality indication. He concluded that the weight of the connecting rod is reduced by 0.477 grams and the inertia force is reduced and the stress at the end of the piston is reduced.

Nikhil et al. In their study the material of the connecting rod has been replaced by aluminum (Al 360) based composite material reinforced with silicon carbide and fly ash and they have also described the model and analysis of the connecting rod. FEA analysis was performed on two materials of 180 cc engine connecting rods. They take parameters like von Mises stress and deformation which were obtained from ANSYS software. The new material was found to have lower weight and better stiffness than existing materials. This resulted in a 39.48% reduction in weight, as well as a 64.23% reduction in displacement. The optimized connecting rod is comparatively much stiffer than the existing connecting rod.

3. ASSEMBLY OF VARIOUS COMPONENTS

3.1. PISTON

Piston is an important part of an engine that works and gives results. The piston forms a guide and bearing at the short end of the connecting rod. It transmits the power of the cylinder to the crankshaft through the connecting rod. The piston transfers the combustion force to the crankshaft and causes the crankshaft to rotate. They also act as a powerful plug that holds the gas in place to keep burning in the cylinder. This is done to remove heat from the heat.

3.2. CONNECTING ROD

The connecting rod connects the piston to the crankshaft. The upper part has a hole for the piston wrist pin and the lower part (large end) is connected to the crankshaft. It is subjected to high stress from the reciprocating load of the piston. With each rotation, it expands and compresses and the load rises to the third power as the engine speed increases.

3.4. CRANKSHAFT

The crankshaft converts the up and down (reciprocating) motion of the piston into reciprocating (rotational) motion. It gives speed to the wheels to rotate. The crankshaft is connected to the piston by a connecting rod. Crankshafts are usually made of alloy steel or cast iron.

4. THEORETICAL CALCULATIONS OF CONNECTING ROD

Engine Type: Petrol engine, 4 cylinders (SDI) Standard, Bore: 90 mm

Stroke: 83mm

Displacement: $2112 cm^3$

Maximum Power: 62 Hp/4500rpm

Maximum Torque: 121Nm/2000rpm

Temperature of Petrol : 22˚C = 295.15K

Density of Petrol at 22˚C = 900 Kg/m^3 = $900*10^{-9} Kg/mm^3$

Mass of Petrol = Density*Volume

$= 900*10^{-9}*2112*10^3$

=1.9008Kg

Molecular weight

= 168.312 = 0.168312 Kg/mole

R (gas constant) = 8.3143

From gas equation PV = m*$R_{specific}$*T

$R_{specific}$= 8.3143/ 0.168312
= 49.3981415J/KgK

PV= Rspecific*Temperature
P = Mass X $R_{specific}$ X Temperature/ Diplacement
P = (1.9008 X 49.3981 X 295.15)/2112
P = 27713.38/2112
P = 13.122Mpa

5. METHODOLOGY

It is an effective method in modeling and structural analysis. In this presentation, weight and structural analysis of the connecting rod by finite element method using ANSYS was performed and presented together. The processes were applied sequentially to a structural steel connection element under a static force loading of 1000N. ANSYS is a general purpose finite element analysis (FEA) software package. Finite element analysis is a numerical technique of dividing a complex system into very small pieces (of user-specified size) called elements. The software uses equations that govern the behavior of these elements and solves them all. These results can be presented in tabular or graphical form. A dynamic, dynamic and thermal analysis is performed on the piston, connecting rod and crankshaft assembly. For the crankshaft the aluminum alloy is 6061 and titanium along with EN308 and high alloy steel.

Finite element analysis is a numerical method described by differential equations to investigate problems and solve them to reach accurate solutions. Solving engineering problems involving complex structures is a great feature of Catia. ANSYS is a CATIA software package that generates calculations that solve and control the behavior of materials. The geometry is defined first depending on the type of analysis to be performed.

A 3D model can be presented in ANSYS by saving it as Initial Graphics Interchange Specification (IGES) format and importing it into ANSYS Workbench, or by creating an entire structure in ANSYS Workbench. In this paper, the analysis is done by importing the geometry from CATIA in IGES format into the system.

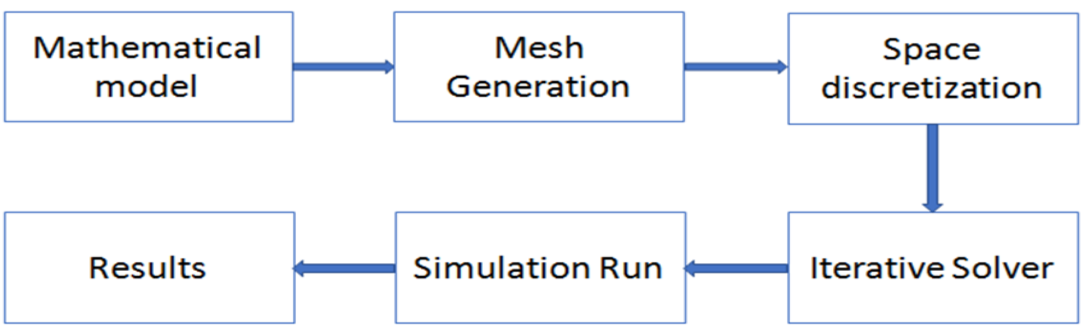

Figure .1. ANSYS Methodology

6. MODAL VALIDATION

The result of the base paper is validated for the following parameters

Table 1 Parameters and Magnitudes

Parameters	Magnitudes		
	Past study	Present study	Percentage error
Total deformation	3.02851 X 10^{-7}	3.19951 X 10^{-7}	0.32
Equivalent Stress	2.1204 X 10^{-8}	2.8704 X 10^{-8}	0.36
Equivalent elastic strain	2.162 X10^{-6}	2.862 X10^{-6}	0.33
Temperature Increments	911.09 °C	951.09 °C	0.05
Directional heat flux	2.1854X 10^{6}	3.1224X 10^{6}	0.43
Total heat flux	6.3594 X 10^{6}	7.5834 X 10^{6}	0.19

7. RESULT AND DISCUSSION

7.1.STRUCTURAL ANALYSIS

Total deformation in transient structural analysis refers to the cumulative displacement and distortion experienced by a structure or component as it responds to changing loads and conditions. Transient structural analysis involves studying how the structure changes over time due to variable or time-dependent loads, such as impact, vibration, temperature changes, and any other various mechanical forces. The total deformation takes into account both the rapid elastic response of the material to applied loads and any plastic deformation that may occur due to material withdrawal. The sum of all the small displacements and deformations that occur during each time step of the analysis. To calculate the total deformation in a short-term structural analysis, you can usually do the following steps:

Discretion

Divide the structure into smaller elements to make a finite element mesh. The behavior of each element is predicted based on its physical properties and geometry.

Time integration

Since transient analysis involves changes over time, the simulation is performed step by step, moving sequentially through time. At each time step, loads and boundary conditions are applied, and the equations of motion are solved to determine the displacement and strain at that particular time.

Cumulative distortion

The displacement and strain obtained at each time step are accumulated to obtain the total strain. This includes both translational and rotational displacements.

Matrial behavior

Depending on the properties of the material and the applied load, the material can undergo elastic deformation (recoverable deformation) and plastic deformation (permanent deformation). This is taken into account when calculating the total deformation.

Convergence and accuracy

It is important to ensure the accuracy and stability of the numerical solution. Iterative solvers and adaptive time-step techniques can be used to obtain accurate results.

Total deformation in transient structural analysis is essential to understand how a structure responds to changing conditions and loads over time. It provides insight into potential failure points, stress concentrations and the overall behavior of the structure under dynamic conditions. Different types of transient loads, such as impact, thermal cycling, and seismic events, can lead to different types and magnitudes of total deformation, making transient structural analysis a valuable tool in engineering and design processes.

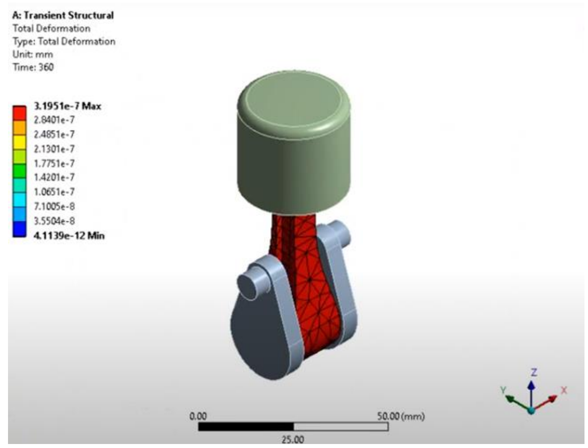

Figure.2. Total deformation

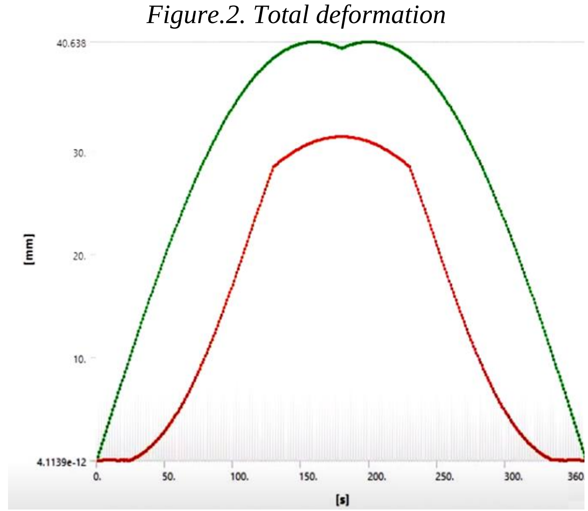

Figure.3. Curve of Total deformation

"Total deformation" in the context of structural analysis refers to the complete displacement and deformation of a structure under the influence of applied loads and boundary conditions. It includes all types of displacements, rotations and distortions occurring in a structure due to external forces, thermal effects or any other relevant factors.

Total deformation takes into account both elastic and plastic deformations, as well as any permanent changes in shape that may cause yielding or failure. It is the sum of all incremental displacements and strains occurring at each time step of the analysis.

In structural analysis, whether for static or dynamic loading, total deformation is important to understand the behavior of a structure under different conditions. It helps engineers and designers assess factors such as stress concentrations, potential failure points, and the overall structural integrity of the system.

To help engineers and designers understand how a structure reacts to load and how it evolves over time, total deformation can be visualized through animations or diagrams. Software tools such as finite element analysis (FEA) are often used to simulate and analyze total deformations in complex structures and systems.

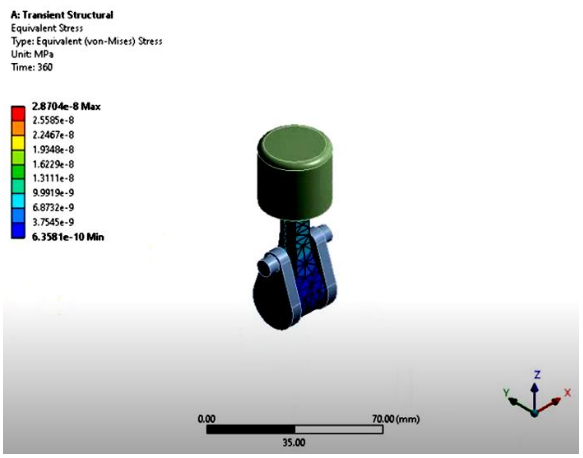

Figure.4. Equivalent Stress

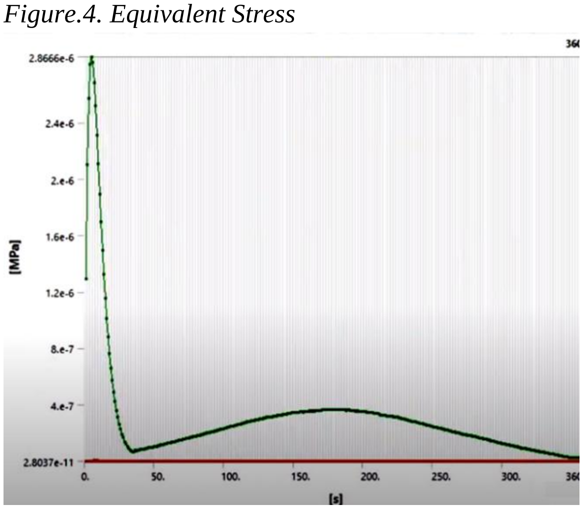

Figure.5. Equivalent volumes stress.

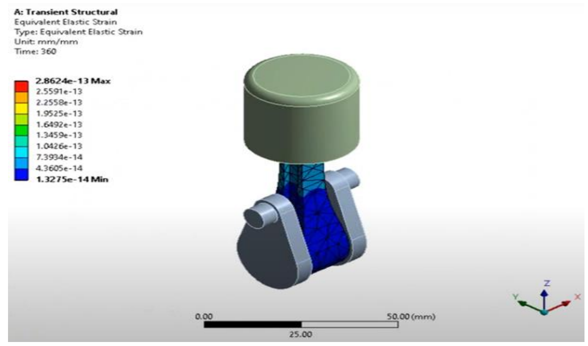

Figure.6. Equivalent elastic strain

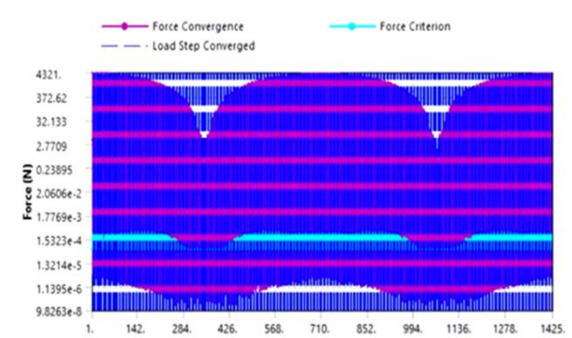

Figure.7. Force based Iteration process

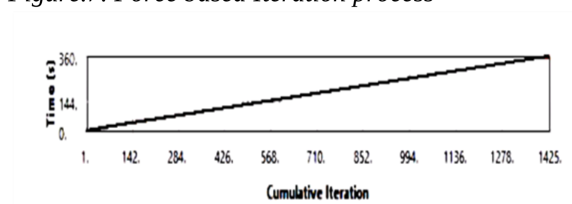

Figure.8.Time based Iteration process

7.2 THERMAL ANALYSIS

Temperature analysis for pistons involves studying the distribution of temperature within the piston as it operates within the cylinder of an engine. This analysis is essential to understanding the piston's thermal behavior, optimizing its design, and ensuring that it can withstand the thermal stresses and conditions experienced during engine operation. Here are the key aspects of temperature analysis for pistons:

Heat generation

Heat is generated by the combustion of fuel inside the engine cylinder. This heat is transferred to the piston through direct contact with the hot gases as well as through radiation and convection from the surrounding components. Understanding the rate of heat generation is important for accurate temperature forecasting.

Thermal properties

Different parts of the piston, such as the crown, skirt, and rings, can have different thermal conductivity, heat capacity, and physical properties. These properties affect how heat is conducted and distributed within the piston.

Contact and clearance

The piston makes contact with the cylinder wall, and this contact can result in heat transfer between the piston and the cylinder. Additionally, there are small gaps between the piston and the cylinder to accommodate expansion. These factors affect the heat transfer mechanism and temperature distribution.

Thermal boundary conditions

The cylinder wall temperature, the cooling medium (eg, coolant or oil), and the combustion gases are important boundary conditions that affect piston temperature. Accurate modeling and consideration of these conditions is critical for realistic temperature analysis.

Cooling Channel and Oil Jet

Some modern engines have cooling channels or oil jets built into the piston design to control temperature. These characteristics affect heat transfer and can significantly affect the temperature distribution within the piston.

Transient vs Steady-State Analysis

Depending on the operating conditions of the engine, a transient or steady-state temperature analysis may be appropriate. Transient analysis considers temperature changes over time, while steady-state analysis assumes a constant operating condition.

Thermal stress analysis

High temperatures can lead to thermal expansion of the piston material. This expansion, combined with mechanical loading from engine operation, can result in thermal stresses that can affect the structural integrity of the piston.

Thermal management

Temperature analysis can help design effective cooling strategies, such as optimizing cooling channel layout or adjusting cooling fluid flow rate, to prevent overheating and ensure consistent piston performance.

Material selection

Pistons are usually made of materials that can withstand high temperatures and thermal stresses. Temperature analysis helps to select materials that match the operating conditions and expected temperature profiles.

Performance and emissions

Temperature distribution affects combustion efficiency, emissions output and overall engine performance. An accurate analysis can guide adjustments to optimize these factors.

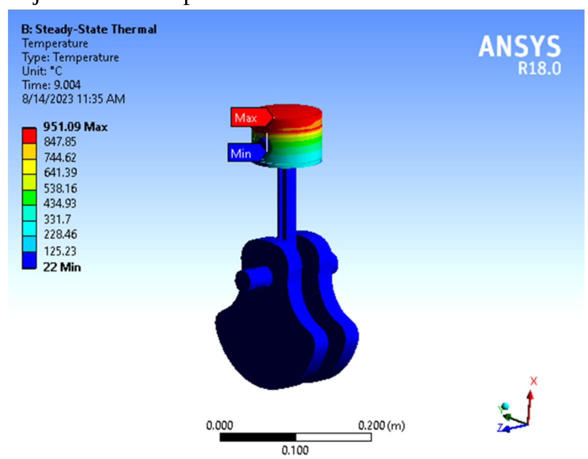

Figure.9. Temperature Increments

Directional heat flux refers to the rate of heat transfer per unit area in a specific direction. Heat flow describes the amount of heat energy that passes through a given area in a given period of time. The "directional" aspect implies that heat transfer is being measured or analyzed in a particular direction or orientation.

Heat flux is typically represented by the symbol "q" and is measured in units of watts per square meter (W/m²) or other equivalent units.

Directional heat flux can vary depending on factors such as the temperature gradient, material properties, and the presence of insulation or other obstructions. Depending on the context, there are different types of directional heat flow:

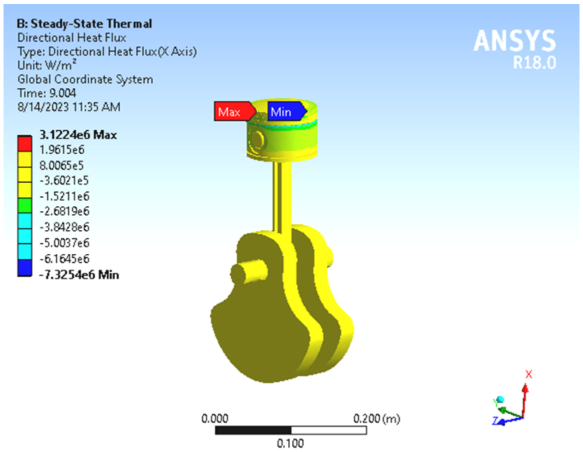

Figure.10. Directional heat flux

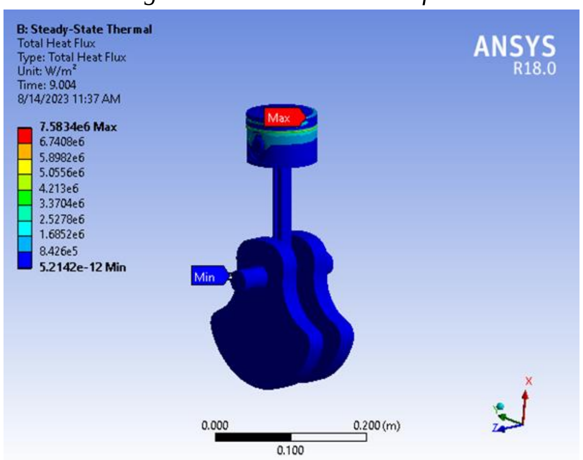

Figure.11. Total heat flux

8. CONCLUSIONS

In order to analyze the temperature of the Piston, Connecting rod and Crank shaft assembly, integrated simulations based on finite element analysis (FEA) or computational fluid dynamics (CFD) techniques are usually used. These simulations take into account the complex interactions between the piston, combustion gases, cooling systems, and other components within the engine. In conclusion, piston temperature analysis is an important aspect of internal combustion engine design and optimization. By understanding how heat is transferred and distributed within the piston, engineers can make informed decisions to improve engine efficiency, reliability, and overall performance. The above study gives an idea about the design of connecting rods. It describes the different stresses to consider when designing a connecting rod with various materials used and compares the effect of all materials. A finite element analysis of the connecting rod was performed in ANSYS Workbench 18.0 considering all loading conditions. Maximum compression pressure is achieved between the pin end of the connecting rod and the rod. The maximum shear stress is achieved at the end of the pin. So the probability of connecting rod failure can occur in the installed section at any point, but the probability of failure is higher at the piston end than at the crank end. Static, Dynamic and Thermal analysis is done on piston, connecting rod and crankshaft assembly. Materials considered for the piston are Aluminum alloy 6061 and Aluminum alloy 2618, the connecting rod is Aluminum alloy 6061 and Titanium and the crankshaft is EN308 and High Alloy steel. Pressures are increased and displacement is less when Al 2618 Piston, Titanium Connecting rod and High Alloy Steel Crankshaft are adopted. The heat transfer rate is higher considering the materials of Al 6061 Piston, Al 6061 Connecting rod and EN308 Crankshaft. Figure.2. shows the total deformation of $3.19951 X 10^{-,7}$, Figure.4. is shows the equivalent Stress of $2.8704 X 10^{-8}$, Figure.6. is shows the equivalent elastic strain of $2.862 X10^{-6}$, Figure.9. is shows the temperature increments of 951.09 °C, Figure.10. is shows the directional heat flux of $3.1224 X 10^{6,}$ and Figure.11. is shows the total heat flux of $7.5834 X 10^{6}$.

REFERENCE

1. B. Kuldeep, L.R Arun, "Analysis and optimization of connecting rod using ALFASiC composites", International Journal of Innovative Research in Science, Engineering and Technology, Vol. 2, Issue 6, pp. 2480-2487, 2013.
2. Bhandari V.B. "Design of Machine Elements" TATA McGraw Hill Publications, Jan.2008.
3. Desai Fanil, Jagtap Kirankumar, Deshpande Abhijeet, "Numerical and experimental analysis of connecting rod", International Journal of Emerging Engineering Research and Technology, Vol. 2, Issue 4, pp. 242-249, 2014.
4. Kadam Sujata D., Date Rohini R., Kadam Ashwini K., Malgave Sujit S,"Review on optimization of connecting rod by using
5. Kumar Amit, P. P Bhingole and Kumar Dinesh, Dynamic analysis of Bajaj Pulsar 150cc
6. Moncy Mobin, "Design and analysis of connecting rod using Aluminium Silicon

Carbide", International Journal of Nano Corrosion Science and Engineering, ISSN Online: 2395-7018, pp. 116-124,2016.

7. More Gajanan Dinkarrao, Mane V.V, "Overview of fatigue failure of connecting rod used in a light commercial vehicle (LCV) through FEA", International Journal of Innovative Research in Science, Engineering and Technology, Vol. 3, pp. 1-6, 2016.
8. P. Arshad Mohamed Gani, T. Vinithra Banu, "Design and analysis of Metal Matrix Composite connecting rod", International Journal of Engineering Research and General Science, Vol. 3, Issue 2, pp. 2091-2730,2015.
9. Ramakrishna G., Venkatesh P.H.J, "Modelling and analysis of connecting rod using 4340 Alloy Steel And AlSiC-9", International Journal of Engineering Science and Research and Technology, ISSN: 2277-9655, pp. 54-58.
10. Rao G. Naga Malleshwara, "Design optimization and analysis of a connecting rod using Ansys", International Journal of Science and Research (IJSR), pp. 225-229, 2013.
11. Sailaja G., Sadaq S. Irfan, Yunus Shaik Vaseem "Dynamic analysis of a connecting rod using FEA", International Journal on Mechanical Engineering and Robotics, Vol-5, pp. 1-4,2017.
12. Saurabh Kunal, Singh Akhand Pratap, "A review paper on design analysis of internal combustion components", International journal of advance research in Science and Engineering, Vol. 06, pp. 495-498, 2017.
13. Singh Puran, Pramanik Debashis, Singh Ran Vijay, "Fatigue and structural analysis of connecting rod's material due to (C.I) using FEA", International Journal of Automotive Engineering and Technology, Vol. 4, Issue 4, pp. 245 253,2015.
14. Singh Vikas, Verma Sumit Kr, "Design and Analysis of Connecting Rod for Different Material Using Ansys Workbench 16.2" International Journal for Research in Applied Science & Engineering Technology, Vol. 5 Issue V, May 2017 IC, Value, 45.98, ISSN. 2321-9653.
15. Thakare Nikhil U., Bhusale Nitin D., "Finite element analysis of connecting rod using Ansys", International Journal of Advances in Science Engineering and Technology, Vol. 3, pp. 82-86, 2015.