

DESIGN OF CONNECTING ROD OF AIR COMPRESSOR: A TOPOLOGY OPTIMIZATION APPROACH

¹Aishwarya B. Dhavale, ²Dr.P.M.Bagade

Department of Mechanical Engineering, Padmabhushan Vasantdada
Patil Institute of Technology, Bavdhan,
Savitribai Phule Pune University, India

Abstract: To obtain optimized weight of connecting rod of an air compressor which can withstand the load acting on it due to compressed gases. The connecting rod is used to convert the rotatory motion of electric motor to the oscillatory motion of the piston. The rod is subjected to the cyclic loads of compression and tension. The optimized connecting rod with reduced weight results in the reduced inertial forces in the system. The objective is to design and analysis the structural stress distribution of connecting rod using real time condition during process. Depending upon loading condition FE analysis of connecting rod can be done. Based upon loading gas pressure and compressor specification loading can be calculated. The modeling of the connecting rod is done on Catia V5 and analysis work is done on ANSYS 19.2. The parameters like Von misses stress, Von misses strain and displacement was obtained from ANSYS software which shows reduction in weight and improvement in strength The overall results of the analysis are also validated by the analytical method and UTM machine. Thus the process results in the optimized and effective connecting rod.

Keywords: Connecting Rod, weight, finite element analysis, Topology optimization.

I. INTRODUCTION

Positive displacement compressors are used to draw in and capture a volume of air in a chamber. Their function is to reduce the volume of the chamber to compress the air. Air pressure is generated by rotating impellers in other types of compressors, positive displacement is more common and includes the models used by homeowner, woodworkers, mechanics, and contractors. Here air pressure is increased by reducing the size of the space that contains air. Most of the compressors which are run across doing this job with a reciprocating piston. In reciprocating compressor a piston is used, which moves inside a cylinder, to compress the air. When the piston moves down, the air is drawn in. When the piston moves up, the air is compressed. Air intake and exhaust are the two valves used in an air compressor.



Fig. 1 Reciprocating Compressor

The function of connecting rod in compressors is to transmit the power from crankshaft to the piston to compress the air. The role of connecting rod in the conversion of rotary motion into reciprocating motion. The lighter connecting rod and the piston will result in greater compression and less vibration as reciprocating weight is less.



Fig.2 Air Compressor

There are different types of production and material application methods used in the creation of connecting rods. The common types of connecting rods material are steel and aluminum. The common types of manufacturing processes are casting, forging and powdered metallurgy. A connecting rod consists of a pin-end, a shank section, and a crank-end. Pin-end and crank-end pinholes at the upper and lower ends are drilled or machined to permit the accurate fitting of bearings.



Fig.3 Connecting Rod

The optimization of the connecting rod had already started as the early year 1983 by Webster and his team. Nowadays customer and industries are looking for the best from the best. That's why optimization is considered important in industries. Optimization of the component is to reduce the time of production of a component, make it durable and cheaper.

II. LITERATURE REVIEW

[1] **J.Chao** [2018] analysed the failure of an oblique split connecting rod with serrated joint faces in a medium-speed gas engine at a cogeneration plant. Engine failure occurred suddenly after 20,000 h of service by malfunctioning without the engine control system providing an automatic engine shutdown signal. The failure affected two trains, which were mounted on the same crankpin and to the crankshaft itself. The analysis was concentrated on the connecting rod in which fatigue features on the fracture surface were observed. The location of the fracture in this connecting rod occurred close to the lower joint within the vicinity of the connecting rod axis. Beach marks on the fracture surfaces indicated that the crack initiated from the big end internal surface, approximately 5 mm from the oil groove, away from any geometric stress concentration. To determine the fracture cause, a material characterization and an analysis of the material state near the fracture initiation point was conducted. The analysis was extended to the bearing shells of the damaged trains and to the bearing shells of a connecting rod that did not fail. The results suggest that the mechanism of failure of the engine may have probably been fretting-fatigue produced by the relative micro-movements between the bearing back and the housing bore.

[2] **S.V.UmaMaheswara, T.V.Hanumanta Rao, K. Satyanarayana, B. Nagaraju** [2018]. Connecting Rod is an important component in reciprocating type of I.C engines. The chances of fatigue failure in connecting rod are higher when it is subjected to alternative compressive and tensile stresses in a cycle of the engine. More over the connecting rod is also subjected to buckling stress as it acts as a column. Hence, it is necessary to determine the fatigue life of the existing connecting rod materials and also look for alternative materials. Using the output of a test, which was conducted on computerized variable compression ratio (V.C.R), Kirloskar Diesel Engine equipment provided with a pressure transducer, kinematic and dynamic analysis of a connecting rod was carried out at a compression ratio of 16.5. Further the results of Dynamic analysis were utilized for carrying out Quasi-Dynamic stress analysis. We embarked on performing the kinematic and dynamic analysis of connecting rod using MATLAB at a compression ratio of 16.5, 17.5 and 18.5 and also at four different critical crank angles with the existing materials namely Forged Steel, Aluminum Alloy and Titanium alloy. Further, the analysis was performed by using ANSYS workbench for determining the Vonmises stresses, Deformations, Life and Factor of safety.

[3] **C. Juarez, F. Rumiche, A. Rozas, J. Cuisano, P. Lean** [2016] This paper presents the results of a failure of analysis connecting rod of the generator, the generator is a device which converts mechanical energy into electrical and fuel used to run the generator is diesel The study and analysis included an analysis of the connecting rod material as well as the fracture zone. visual inspection, fractography, magnetic particle inspection, chemical analysis, tensile and hardness testing, metallographic, and microanalysis were the experiment and testing techniques performed. AISI/SAE 4140 low alloy steel was used for fabrication of connecting rod, as per application appropriate chemical composition, mechanical properties, and microstructure were selected. After the analysis connecting rod was fractured at the body in a section close to the head; the origin of the fracture was located at the con-rod lubrication channel from a machining tool, embedded in its surface as a result of a deficient manufacturing process. The analyzed area acted as a nucleation site for cracks that propagate through the connecting rod at section by a fatigue mechanism, reducing its section and finally producing its catastrophic failure.

[4] **G Gopala, Dr L Suresh Kumar, K Vijaya Bahskar Reddy, M Uma MaheshwaraRaod , G Srinivasulu** [2017] The paper deals with the study of an assembly of the Piston, Connecting rod and Crankshaft of a four-wheeler petrol engine. The components of the assembly have to be rigid and the assembly has to move as a mechanism. Hence, Rigid-body and flexible body analysis were

included. So, the forces in the components as the engine reciprocates have to be calculated and these forces are used to calculate the dynamic stresses in the component of interest i.e. the connecting rod. It is proposed to replace with two new sets of materials for the components of the assembly and check the parameters by performing the static, dynamic and thermal analysis. In this project, the main parts of the assembly i.e. engine piston, connecting rod and crankshaft are modeled and assembled as per the given design. And the Finite Element Analysis is done in Ansys. The meshing is done in Hyper Mesh.

[5] **K. Bari, A. Rolfe, A. Christofi, C. Mazzuca and K.V. Sudhakar** [2017] This research paper explains an investigation of the failure of connecting rod from a motorcycle engine considering the root cause of and mechanisms which leads to its premature failure. Including the root cause, the other task from this study was to improve the existing designs or methods to avoid failures in the future. These results were validated using a finite element analysis (FEA) simulation. A Scanning Electron Microscope was used for investigating the mechanisms of fracture modes, optical microscopy for studying the microstructures and visual inspection were primarily utilized to determine the root cause of the failure. In conclusion, it was determined that the root cause for the premature failure of the connecting rod was the presence of scale build-up inclusions, which led to micro cracking during fatigue loading of the component.

[6] **Zhou Shia, Shuqing Koua** [2018] Quality defects of fracture surface are easily produced in the fracture splitting process of the connecting rod. Presently, the resulting mechanism of this type of defect is lacking in evidence and there is no effective control method. Based on fracture mechanics and the actual production, this study simulates the process of connecting rod fracture splitting using finite element method. Three-dimensional crack propagation state is described, and the stress and deformation of the crack tip are analyzed. The essence of the fracture surface defects, such as drop dregs, steps, and crack bifurcations, are mainly due to the change of the crack path and structure during crack propagation. These defects mainly occur at the boundary positions of the fracture surface such as bolt hole and end faces, which is due to the boundary's stress and deformation characteristics of the fracture surface. The boundary is in the plane stress state, the crack tip undergoes shear deformation and has a large plastic zone which leads to a severe twist of the crack tip opening angle. Fracture-splitting defects were the result obtained by the cause of the crack propagation path to be twisted and bent.

[7] **Slavko Rakić, Dr Ugljesa Bugarić** [2017] This paper presents failure analysis of the connecting rod used in a 12-cylinder diesel engine set on a special vehicle. The connecting rod was fractured during the engine test in laboratory conditions. Chemical and metallographic analysis, as well as mechanical testing, have confirmed that the material properties of the connecting rod met the requirements of standard specification and technical documentation. Stress state of the connecting rod under maximum load was evaluated by performing Linear finite element (FE) analysis. The output of the FE analysis indicated that the position of the fracture is consistent with the zone of highest stress. Fractographic analysis has not been able to reveal the main cause of the mechanism of fracture due to the substantially damaged fracture surface. Inadequate machining, absence of polishing and highest stress in the region of fracture were identified as the main causes of failure. Hence, the engine was working at maximum load for a longer period of time that also led to the breakage of the connecting rod.

[8] **D. Gopinatha, Ch. V. Sushma** [2015] Every vehicle that uses an internal combustion engine requires at least one connecting rod depending upon the number of cylinders in the engine. It undergoes high cyclic loads of the order of 10⁸ to 10⁹ cycles, which range from high compressive loads due to combustion, to high tensile loads due to inertia. Therefore, durability of the component is of critical importance. Because of these factors, the connecting rod has been used as the topic of research for different aspects mainly production, materials, performance simulation, etc. The main problem definition of research was to explore weight reduction and create better opportunities for the production of forged steel, aluminium and titanium connecting rods. This has entailed performing a detailed load analysis. Hence, this study has dealt with two main subjects, first, static load stress analysis of the connecting rod for three different materials, and second, optimization for the weight of forged steel connecting rod. In this research, firstly a proper geometrical model was developed using CATIA.

[9] **Mohammed Mohsin Ali Ha** [2015] The Connecting rod transfers the load on the piston to the crank by causing the latter to turn, thus converting the reciprocating motion of the piston into a rotary motion of the crankshaft. Connecting rods are subjected to forces which are generated by mass and fuel combustion. The connecting rod is modeled using CATIA software and FE analysis is carried out using ANSYS Software. Load distribution plays an important role in fatigue life of the structure. Bush failure changes the loading direction and distribution. The present study is concentrated around the fatigue life due to concentrated load and cosine type load distribution on the bigger end.

[10] **Zhu Jun-chao** [2016] Dynamic lubrication analysis of connecting rod is a very complex problem. Some factors have a great effect on lubrication, such as clearance, oil viscosity, oil supplying hole, and surface roughness. This work establishes the EHD characteristic of bearing and bush to analyze the peak oil film pressure, min oil film thickness and nodes force with consideration of deformation and cavitation. Then connecting rod components are created to calculate the stress distribution with consideration of film lubrication or not. Finally, contract the results of two conditions. The results show that: the stress distribution of two conditions are different in individual positions, but the stress calculation method considering the oil film lubrication is more consistent with the actual working conditions.

III. PROBLEM DEFINITION

Each day consumers are looking for the best from the best. That's why material optimization is really important in the industry. Material Optimization of the component is to make less time to produce the product that is stronger, lighter and less cost. Influence of design and weight on performance of the connecting rod. Hence, it is an effect on the manufacture of credibility. The tensile and compressive stresses are produced due to pressure, and bending stresses are produced due to centrifugal effect & eccentricity. Hence connecting rods are designed generally of I-section so that maximum rigidity with minimum weight. Significant increments in weight and performance can be achieved by changing the structural design and also material.

Pneumatic Compressors are subjected to less dynamic loads as compared to automotive engines. Exercise work has been done on automotive Connecting Rods in terms of thermal and structural optimization.

IV. OBJECTIVES

- The main aim of the project is to determine the Von Misses stresses, Shear stresses, and Equivalent Alternating stress, Total Deformation, Fatigue Analysis and Material Optimization in the existing Connecting rod.
- The result comparison between the Aluminum material and Plastic material of connecting rod.
- In this Project, the static FEA of the connecting rod has been performed by the use of the software. Hence work can be extended further to study the effect of loads on the connecting rod under dynamic conditions.
- Experimental stress analysis of Model using Strain gauging & lifting load applied through UTM.
- Experimental buckling analysis can also be used to calculate the stresses which will provide more reasons to compare the different values obtained.
- Comparative analysis between FEA & Experimental results.

V. METHODOLOGY

1. Modeling current connecting rod.
2. Analyzing for stresses and deformation.
3. Topological optimization for the model.
4. Results from topological optimization will be implemented on the existing model
5. Analyzing for stresses and deformation on the optimized model.
6. Machining the existing connecting rod of air compressor as per the optimization result.
7. Finding high strain portion from CAE software.
8. Mounting strain gauge on the same portion.
9. Preparing fixtures to hold the connecting rod for experimental testing.
10. Correlating results of both CAE and experimental.

SPECIFICATIONS OF COMPRESSOR:

Company of Compressor = Speed

H.P.= 7.5

RPM = 640

Tank in Size = 21*52 (In Inch) = 290 PSI

Working Pressure = 183 PSI = 1.26 MPa

Bore*Stroke = 70*70 (In mm)

Piston Displacement = 27mm

VI. BASIC CALCULATION

$$\text{Area} = l * b = 3.2 * 3.8 \text{ mm}^2$$

$$\text{Pressure} = \text{Force} / \text{Area}$$

$$\text{Force} = \text{Pressure} * \text{Area} = 1.26 * 1216$$

$$= 1532.16 \text{ N}$$

Geometry of model

Geometry

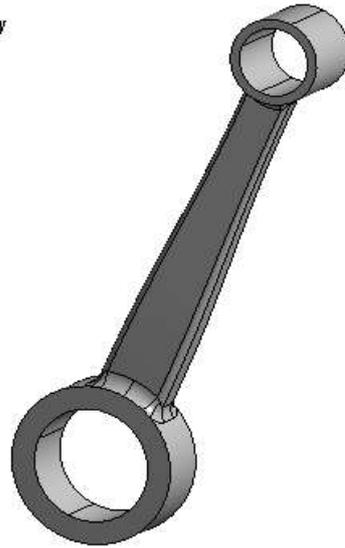


Fig.4 Aluminum Connecting Rod

FINITE ELEMENT ANALYSIS

The Finite Element Method is a numerical approximation method, in which complex structure divided into the number of small parts that is pieces and these small parts are called as finite elements. These small elements connected to each other by means of small points called as nodes. As finite element method uses matrix algebra to solve the simultaneous equations, so it is known as structural analysis and it's becoming primary analysis tool for designers and analysts.

The three basic FEA process is

- a) Pre-processing phase
- b) Processing or solution phase
- c) Post-processing phase

Modal analysis is done to determine the natural frequencies and mode shapes of a structure. Important parameters designing structure for dynamic loading conditions are natural freq, mode shapes.

FEA Pre Processing:

The pre-processing of the sheet metal is down for the purpose of dividing the problem into nodes and elements, developing eqn. for an element, and applying boundary conditions, initial conditions for applying loads. The information required for the pre-processing stage of the sheet metal is as follows,

- The values of Young's modulus, poissons ratio, density, and yield strength for steel are taken from the material library of the FEA package.
- Material properties
 Material- Structural Steel
 Young's Modulus- 200 GPa
 Poissons Ratio- 0.3
 Density- 7850 kg/m³

Yield Strength- 520 MPa..

Properties of Outline Row 3: Aluminum Alloy 1E			
	A	B	C
1	Property	Value	Unit
2	Density	2770	kg m ⁻³
3	Isotropic Elasticity		
4	Derive from	Young's Modulus...	
5	Young's Modulus	7.3E+10	Pa
6	Poisson's Ratio	0.33	
7	Bulk Modulus	6.9608E+10	Pa
8	Shear Modulus	2.6692E+10	Pa
9	Bilinear Isotropic Hardening		
10	Yield Strength	2.8E+08	Pa
11	Tangent Modulus	9E+08	Pa

Fig. 5 .Material Properties of Aluminium

MESH

Multi-purpose, intelligent, highly automated performance product ANSYS is used for meshing. It produces the most appropriate mesh for accurate, efficient Metaphysics solutions. A mesh suited for any particular analysis can be generated with a single click of the mouse for a particular product. The expert can control all the options used to generate the mesh and fine-tune it.

The Finite Element Method is a numerical approximation method, in which complex structure divided into a number of small parts that is pieces and these small parts are called as finite elements. These small elements connected to each other by means of small points called as nodes. As finite element method uses matrix algebra to solve the simultaneous equations, so it is known as structural analysis and it's becoming primary analysis tool for designers and analysts.

The three basic FEA process are

- a) Preprocessing phase
- b) Processing or solution phase
- c) Post processing phase

Modal analysis is done to determine the natural frequencies and mode shapes of a structure. The natural freq. and mode shapes are important parameters in the design of a structure for dynamic loading conditions.

FEA Pre Processing:

The pre-processing of the sheet metal is down for the purpose of the dividing the problem into nodes and elements, developing eqn. for an element, and applying boundary conditions, initial conditions for applying loads. The information required for the pre-processing stage of the sheet metal is as follows,

- Material properties: The values of young's modulus, poisons ratio, density, and yield strength for steel are taken from material library of the FEA package.

Material properties

Material- Structural Steel

Young's Modulus- 200 GPa

Poisons Ratio- 0.3

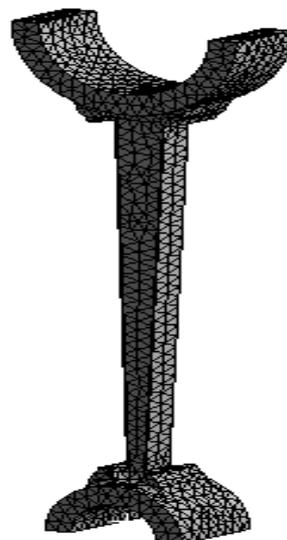
Density- 7850 kg/m³

Yield Strength- 520 MPa..



Statistics	
<input type="checkbox"/> Nodes	23620
<input type="checkbox"/> Elements	11000

Fig. 6. Meshing of Aluminium Connecting Rod



Statistics	
<input type="checkbox"/> Nodes	17333
<input type="checkbox"/> Elements	8552

Fig. 7. Meshing of Optimized Model

Boundary Condition

A boundary condition of the model is the setting of known value for displacement or an associated load. Either the load or the displacement can be set for a particular node. The most used types of loading available in FEA study include force, pressure, and temperature. These can be applied to any points, surfaces, edges, nodes, and elements or remotely offset from a feature of part, assembly or product.

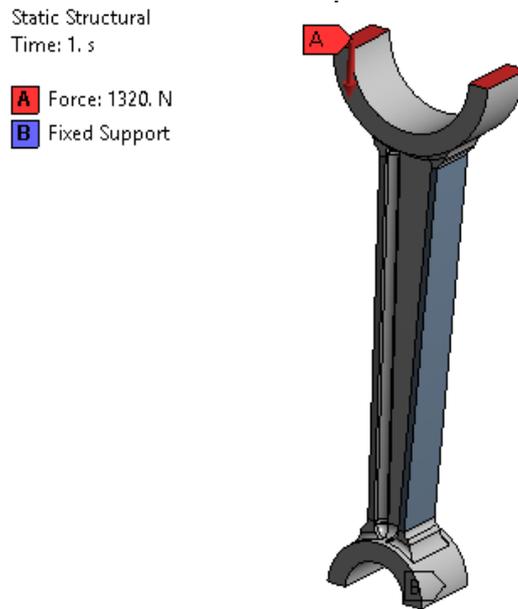


Fig.8 Boundary Condition on Aluminum Connecting Rod

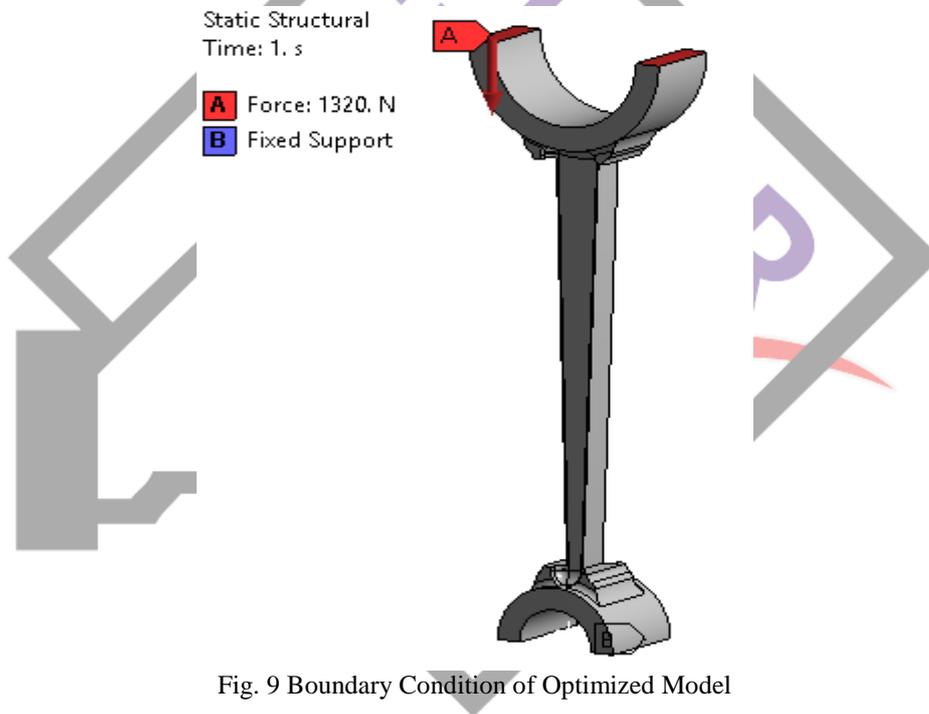


Fig. 9 Boundary Condition of Optimized Model

Total Deformation

The total deformation & directional deformation are general terms in finite element methods irrespective of software being used. Directional deformation can be used as the displacement of the system in any particular axis or user-defined vector. Total deformation can be defined vector sum all directional displacements of the systems or in space.

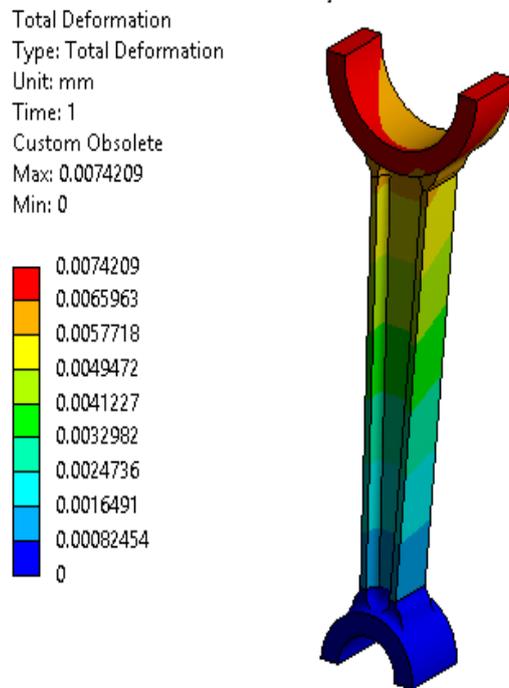


Fig.10. Total Deformation of Connecting Rod

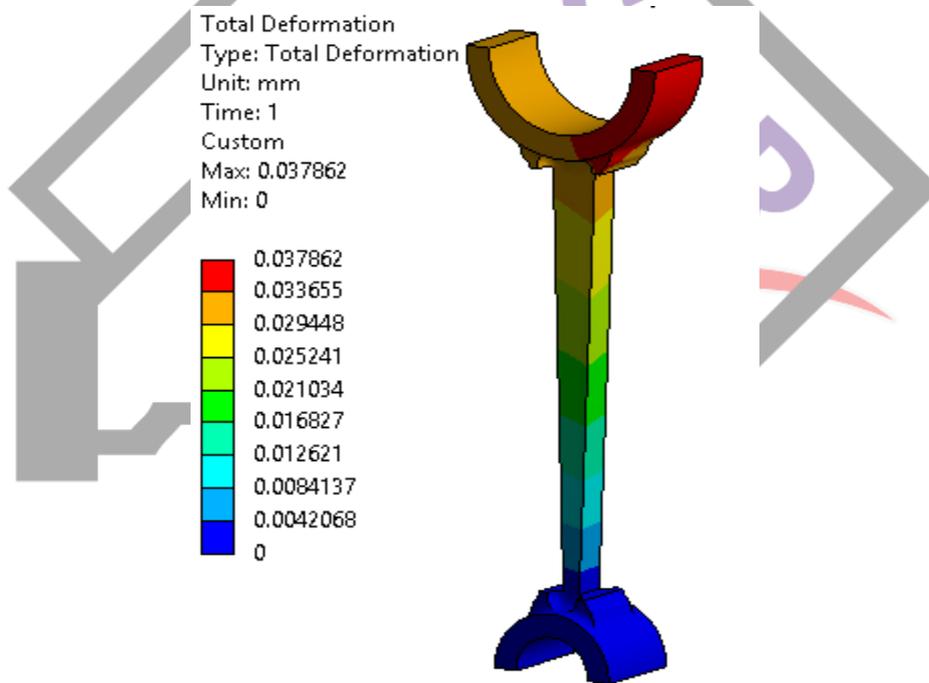


Fig. 11. Total Deformation of Optimized Model

Equivalent Stress

Equivalent stress is equivalent to the principal stresses by the equation

$$\sigma_e = \left[\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{2} \right]^{1/2}$$

Equivalent stress is mostly used in designing product or part because it allows any arbitrary three-dimensional stress state to be represented as a single positive stress value. Equivalent stress is represented as the maximum equivalent stress failure theory used to predict any yielding in a ductile material.

$$\epsilon_e = \frac{1}{1+\nu} \left(\frac{1}{2} [(\epsilon_1 - \epsilon_2)^2 + (\epsilon_2 - \epsilon_3)^2 + (\epsilon_3 - \epsilon_1)^2] \right)^{\frac{1}{2}}$$

The von Mises or equivalent strain ϵ_e is expressed as:
Where:

ν' = effective Poisson's ratio

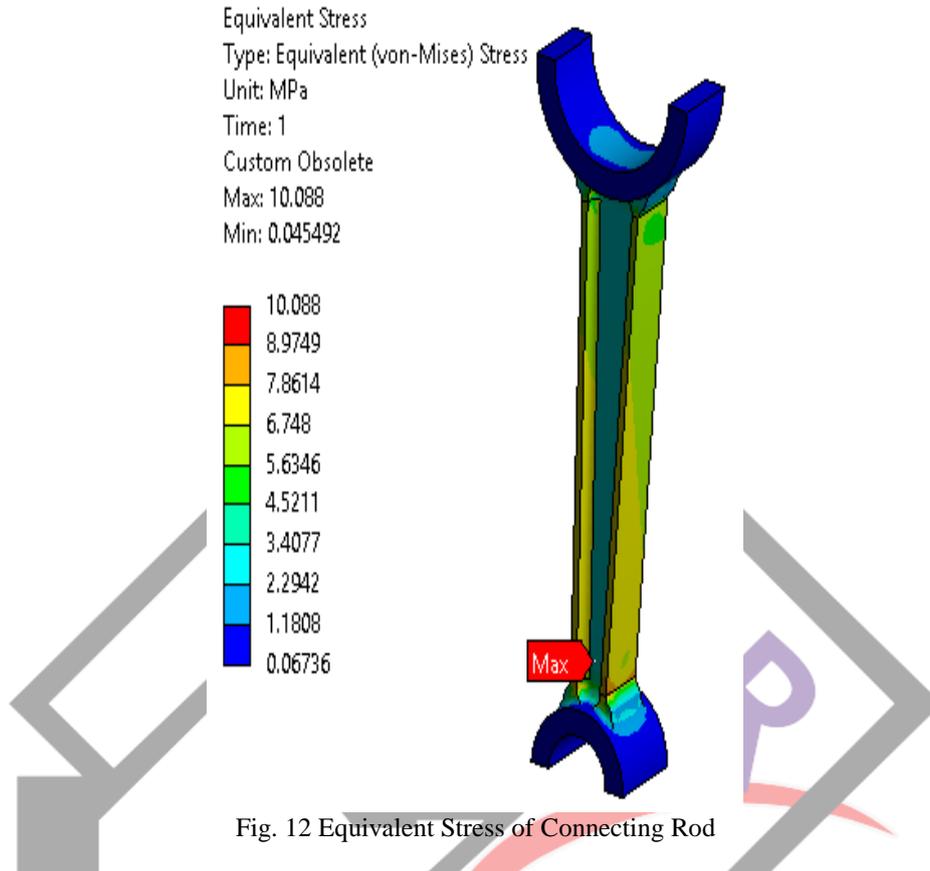


Fig. 12 Equivalent Stress of Connecting Rod

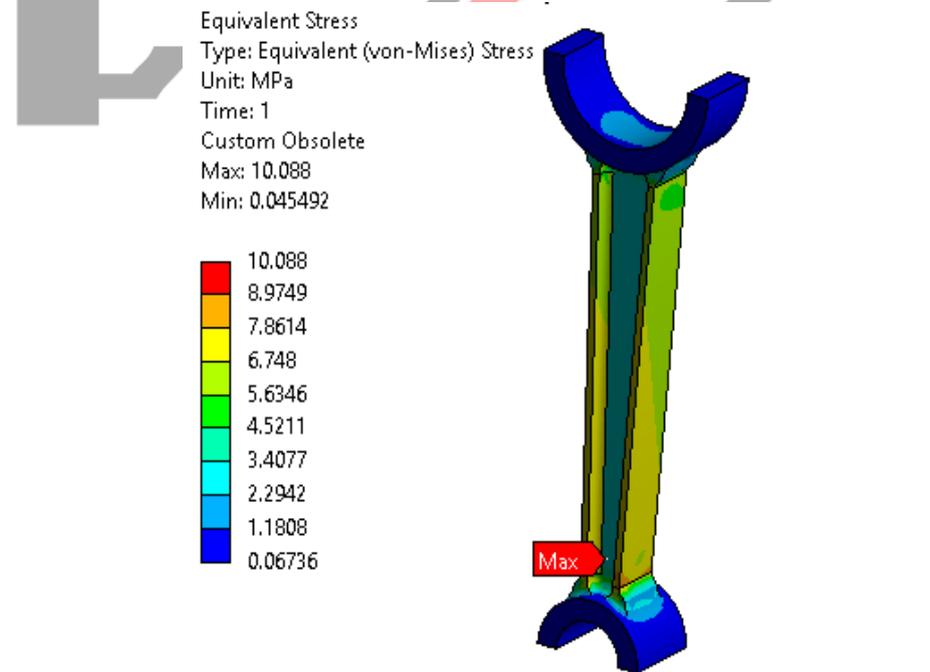


Fig. 13. Equivalent Stress of Optimized Connecting Rod

Minimum principal strain

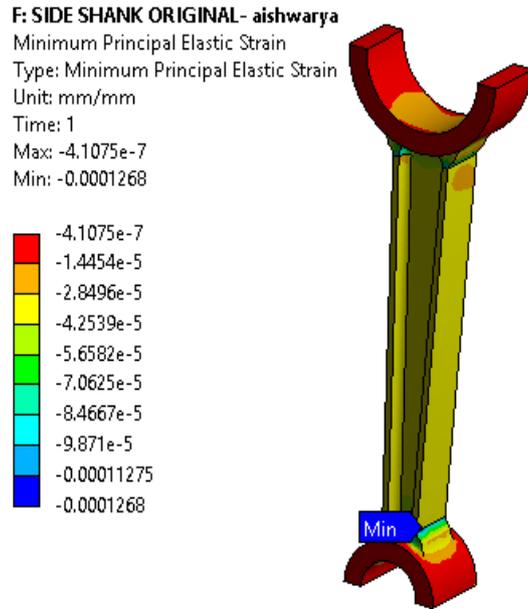


Fig. 14. Minimum Principle Strain of Connecting Rod

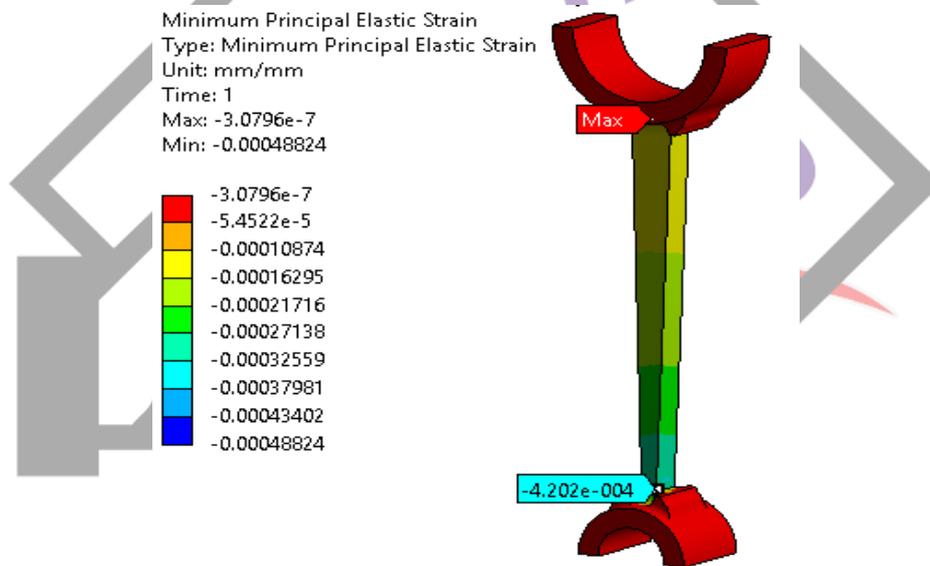


Fig. 15. Minimum Principle Strain of Optimized Connecting Rod

VII. EXPERIMENTAL TESTING

Universal tester is also known as a universal testing machine (UTM which can be used to test the tensile strength and compressive strength of materials. An earlier name for a tensile testing machine is a tensometer. It can perform many standard tensile and compression tests on materials, components, and structures (in other words, that it is versatile) hence named as the "universal".

The set-up and usage are detailed in a test method, often published by a standards organization. This specifies the sample preparation, fixturing, gauge length (the length which is under study or observation), analysis, etc.

The specimen is placed in the machine between the grips and an extensometer if required can automatically record the change in gauge length during the test. The machine itself can record the displacement between its crossheads on which the specimen is held even if an extensometer is not fitted. UTM not only records the change in length of the specimen but also all other extending / elastic components of the component and its drive systems including any slipping of the specimen in the grips.

Once the machine is operated it starts to apply an increasing load as per input on the specimen. The controls system and its associated software record the load and extension or compression of the specimen throughout the tests.



Fig.16. Experimental Testing

VIII. RESULT & CONCLUSION

The Connecting Rod is optimized. The weight of Connecting rod before optimization is 234.6g and connecting rod after optimization is 133g. So, the Optimization is done. The strain value obtained after the analysis result is 3.0769 and the strain value obtained after the experimental testing is 3.267. So, the results are nearly equal. So, the results are validated.

REFERENCES:

- [1] J.Chao, "Fretting-fatigue induced failure of a connecting rod" ScienceDirect, Volume 96, February 2019.
- [2] S.V.UmaMaheswara, T.V.HanumantaRao, K. Satyanarayana, B. Nagaraju, "Fatigue Analysis of Sundry I.C Engine Connecting Rods" ScienceDirect, Volume 5, Issue 2, Part 1, 2018, Pages 4958-4964.
- [3] C. Juarez, F. Rumiche, A. Rozas, J. Cuisano, P. Lean, "Failure analysis of a diesel generator connecting rod" ScienceDirect, Volume 7, October 2016, Pages 24-31.
- [4] G Gopala, Dr L Suresh Kumar, K Vijaya Bahskar Reddy, M Uma MaheshwaraRaod, G Srinivasulu, "Analysis of Piston, Connecting rod and Crank shaft assembly", ScienceDirect, Volume 4, Issue 8, 2019, Pages 7810-7819.
- [5] K. Bari, A. Rolfe, A. Christofi, C. Mazzuca and K.V. Sudhakar, "Forensic Investigation of a Failed Connecting Rod from a Motorcycle Engine", Science Direct, Volume 9, October 2017, Pages 9-16.
- [6] Zhou Shia, Shuqing Koua, "Analysis of quality defects in the fracture surface of fracture splitting connecting rod based on three-dimensional crack growth" Science Direct, Volume 10, September 2018, Pages 1022-1029.
- [7] Slavko Rakić, Dr Ugljesa Bugaric, "Failure analysis of a special vehicle engine connecting rod" Volume 17, September 2017, Pages 98-109.
- [8] D.Gopinatha, Ch.V.Sushma, "Design & Optimization of Four-Wheeler Connecting Rod Using Finite Element Analysis", Volume 2, Issues 4-5, 2015, Pages 2291-2299
- [9] Mohammed Mohsin Ali Ha, Mohamed Haneef "Analysis of Fatigue Stresses on Connecting Rod Subjected to Concentrated Loads at The Big End", Volume 2, Issues 4-5, 2015, Pages 2094-2103.
- [10] Zhu Jun-chao, Zhu Han-hua, Fan Shi-dong, Xue Liang-jun, Li You-feng, "A Study on the influence of Oil film lubrication to the Strength of Engine Connecting Rod Components", ISSN