International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 06 Issue: 08 | August - 2022 Impact Factor: 7.185 ISSN: 2582-3930

Study and Analysis of Connecting Rod with different Condition and Parameters by using CATIA and ANSYS

Mukesh Ahirwar

ME IVth sem student, Department of Mechanical Engineering, Jabalpur Engineering College Jabalpur (M P), India

Prof. A.K. Jain

Associate professor, Department of Mechanical Engineering, Jabalpur Engineering College Jabalpur (M P), India

Abstract- The connecting rod is an important part of the IC motor as it transmits the movement of the piston to the drive shaft and converts the piston-turning motion into a rotating motion and vice versa. From the application point of view, it is essential that the connecting rod be lightweight and of good strength under fatigue or reverse loading. For this purpose in general the connecting rod material is carbon steel from aluminum alloy. Choosing a connecting rod for good engine performance is very difficult. The material used in the connecting rod must be chosen wisely because during the manufacturing process it must undergo various production processes and subsequent heat treatment process, which is very important for strength and toughness. Accordingly, the connecting rod made of high-strength carbon fiber will be compared with the connecting rod composed of stainless steel and aluminum alloy. The results can be used to improve weight reduction and to modify the linkage design.

Keywords- Connecting Rod, Catia, ANSYS Workbench

1. INTRODUCTION

The connecting rod is a member that connects the piston and the crankshaft. Materials are used, such as structural steel, aluminum alloy, titanium, and cast iron [1]. The connecting rod package needs to be tailored to fit the engine and customer needs, says Kerry Novak of Crower [2].

The small end of the connecting rod is attached to the end of the piston with a gudgeon pin/wrist pin by appropriate pressure; The large end is attached to the crankshaft using fasteners. The stresses on the connecting rod are always high due to the pressure of the combustion chamber, inertial forces, which leads to a higher value of the pressures. According to Fiji [3], connecting rod failure, commonly called "rod throw" is one of the most common causes of catastrophic engine failure in automobiles. However, connecting rod failures are not uncommon because major auto companies try to maintain a very high safety factor of 2 or 3 above. To provide warranty, car companies must possess robust design and manufacturing capability. With all of these factors in mind, a lot of engines fail or stall due to a failed connecting rod assembly, leaving companies to consider the connecting rod to be a high-risk component.

Forged steel is currently a mustang material to enhance the ecological environment. AA is mostly used in aerospace applications; This material is used to handle high pressure values. Forged Steel (FS) - The cosmetic trend of using aluminum alloys as a CR member primarily began to reduce weight, but as engine design evolved day by day, engineers returned to steel.

2. LITERATURE REVIEW

Axial pressures and bending pressures act on the connecting rod inside the combustion chamber. He also says that the axial stress is due to the combustion chamber pressure and inertia forces and that the bending stress is due to the centrifugal action of the connecting rod when connected to the crankshaft. Tony George Thomas adds that fatigue and failure are very high due to the fluctuation of these loads.

Yogesh says that 50-90% of connecting rod failures are due to fatigue failure, so it is very important to consider fatigue failure in the connecting rod design and great care must be taken by the Computer Aided Engineering (CAE) team in a company to perform a fatigue analysis and come up with a redesign proposal , if it is necessary. 2016 Ford Eco Boost Mustang uses forged steel as a connecting rod member. There has always been a war thug in the automobile industry to choose the type of connecting rod material. In this thesis, 7075 forged steel and aluminum are used as connecting rod material. CAE analysis is performed to select the best materials.

Computer Aided Engineering (CAE) team in a company performs analysis of all real world problems using many different software by applying real world constraints to get solutions. Each company is equipped with a CAE team, which performs detailed analysis on the connecting rod in every automobile company by applying combustion chamber constraints such as pressure, inertia forces and suppression of connecting rod linear motion that were absolutely necessary. This team comes up with real-time results after performing the analysis and makes suggestions for redesign, if needed. Once the CAE team approves the design, the actual production of the part begins. The connecting rod specified in this analysis is under investigation to verify stresses and fatigue life of the component. Moreover, if the connecting rod fails in the design requirements, a new design proposal is submitted when necessary.

The temperature generated inside the CC is about 300°C for the 4 stroke IC, which is taken by the piston head, says VelivelaLakshmikanth, Dr. Amar NageswaraRao. As we can see in the picture, the temperature effects are very high at the piston head and the temperature drops to 50°C at the edge of the piston (the piston skirt is the side part of the piston that comes into contact with the piston ring). By the time the temperature effects reach CR, they keep decreasing, which is why temperature effects are neglected.

3. MESH AND MESH SENSITIVITY

Solving a complex object to find the results of stress and fatigue life without using finite element analysis is laborious, takes many hours of work and often results in human errors in solving complex equations. It has stripped the entire component into small elements, and this process of breaking up the body is called crosslinking. These small items are solved individually to obtain solutions. Then the solution of each element is summed up individually to get a final solution. One must understand that the obtained solution is not exact, but approximate solutions that engineers can trust.

3.1 MESH

A very fine mesh was generated in critical regions such as the slice area and CR edges. These are the sections in CR where there is a maximum probability. Network connections are created in the aggregation to communicate during the network operation and make the aggregation a single model for the results of the analysis.

3.2 MESH SENSITIVITY ANALYSIS

The purpose of this analysis is to obtain an accurate output solution. In this thesis, they are made to eliminate stress and fatigue plots precisely. The relationship between the input value and the output values is understood using network sensitivity analysis. The output results for different input element sizes from 8 mm to 2 mm (element size) were studied.

4. METHODOLOGIES

CATIA V5 software is used to create the connecting rod. Catia software is able to develop various types of engineering, assembly, sheet metal work, etc. using different types of modules. To develop a 3D model of the connecting rod, part design assembly was used.

5. STATIC STRUCTURAL ANALYSIS

This type of analysis deals with constant load conditions only and ignores the effects of loads that change over time, for example inertia and damping effects. However, inertial loading due to self-weight, reciprocating and rotational motion can be considered, where stress, deformation and factor safety plots are obtained by performing a static structural analysis. The stress diagrams that Von misses are used in this analysis because they give detailed stress diagrams against the yield limit and are also often used because they give an outline of all ductile materials in theory of plasticity.

Figure 1 Modeling of Connecting Rod

Figure 2 Modeling of Connecting Rod (Thick Type)

6 Results and Discussion

CATIA V5 software is used to create the connecting rod. Catia software is able to develop various types of engineering, assembly, sheet metal work, etc. using different types of modules. To develop a 3D model of the connecting rod, part design assembly was used. This section contains a static analysis of carbon steel plates. In this study, 38,217 nodes and 6582 items were selected in the networking area. The node is the intersection of the elements.

6.1 Static Structural

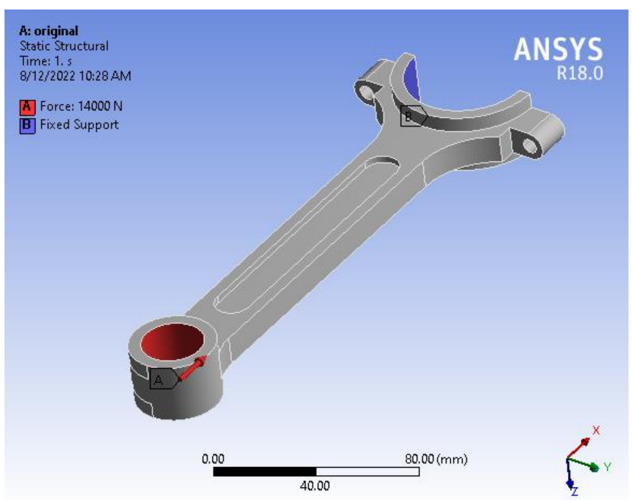

Figure 3 Static Structural

Equivalent Stress

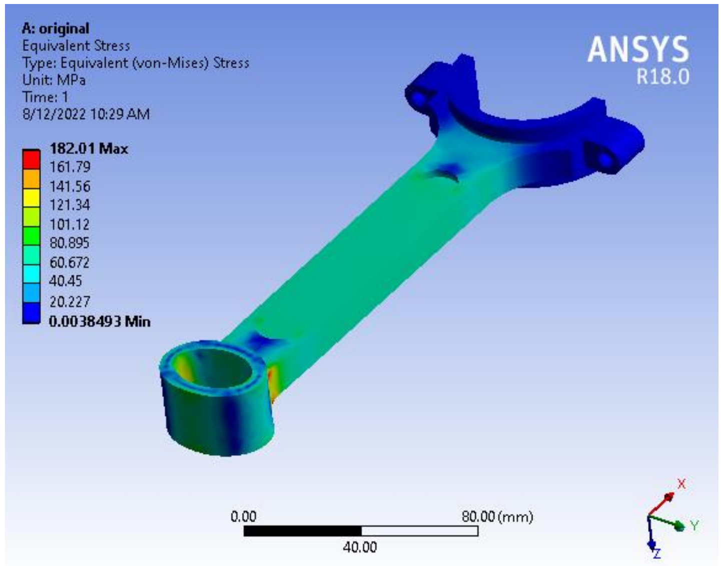

Figure 4 Equivalent Stress

Total Deformation

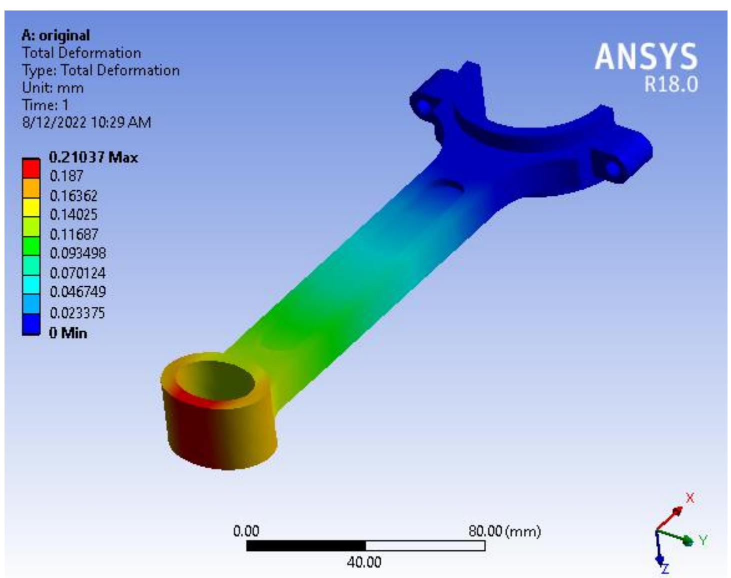

Figure 5 Total Deformation

Safety Factor

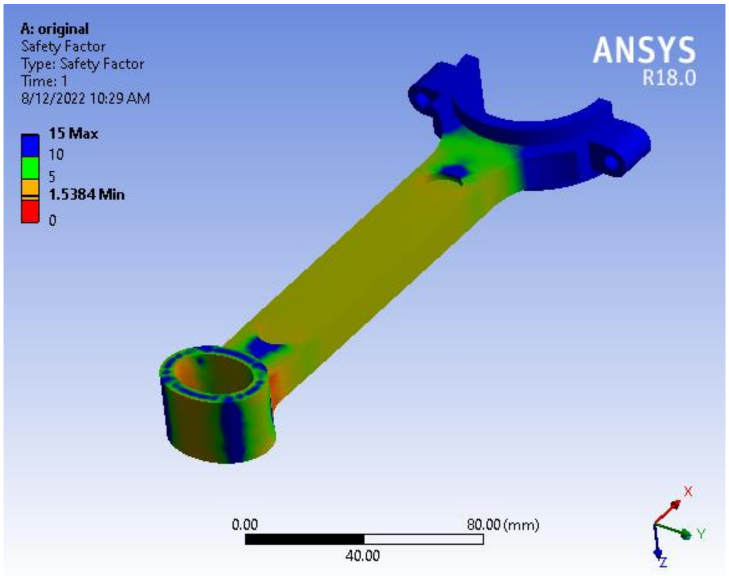

Figure 6 Safety Factor

Equivalent Elastic Strain

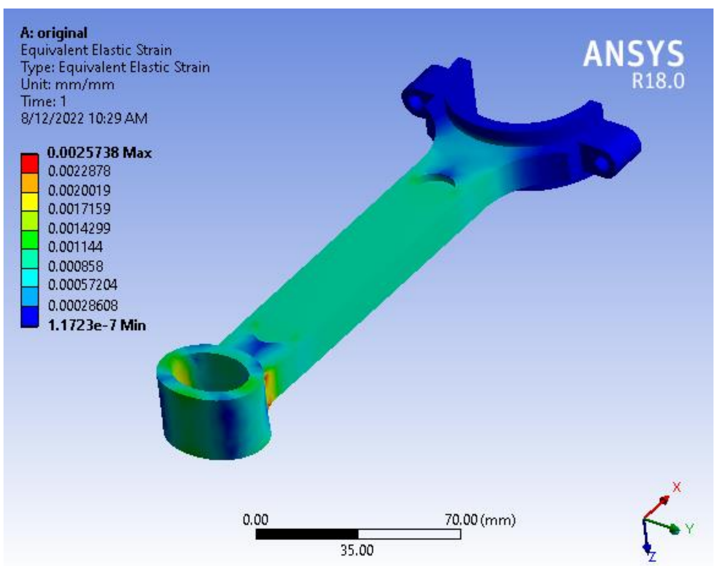

Figure 7 Equivalent Elastic Strains

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 06 Issue: 08 | August - 2022 Impact Factor: 7.185 ISSN: 2582-3930

Minimum Principal Elastic Strain

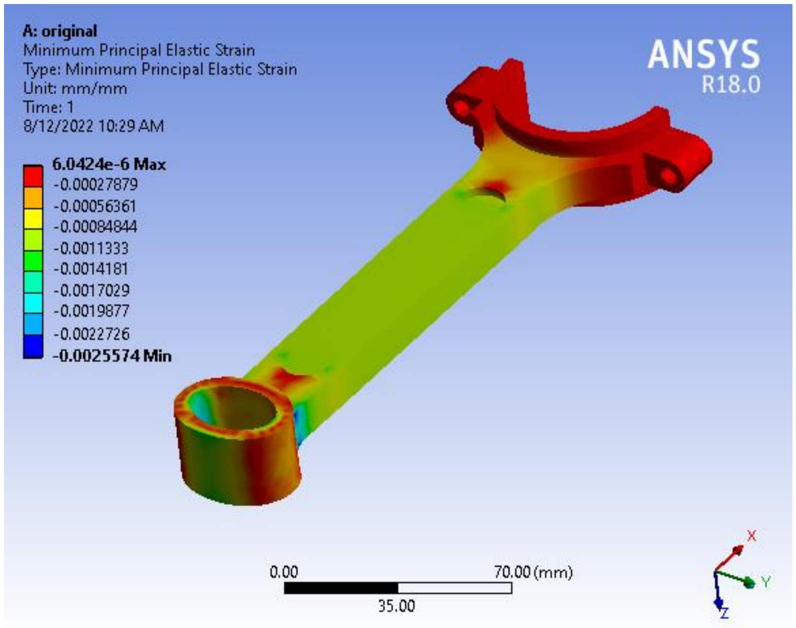

Figure 8 Minimum Principal Elastic Strain

Minimum Principal Stress

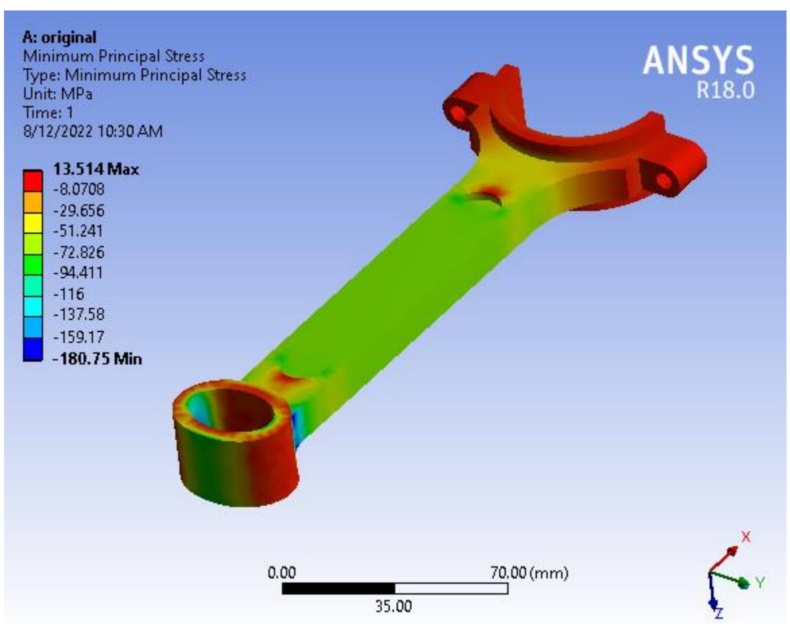

Figure 9 Minimum Principal Stress

6.2 Static Structural (shell thick 3mm)

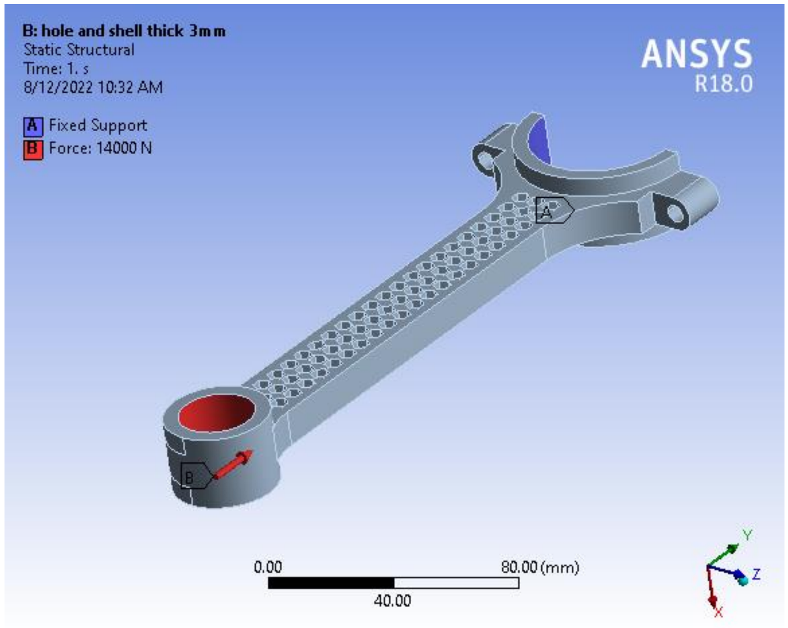

Figure 10 Static Structural (shell thick 3mm)

Equivalent Stress

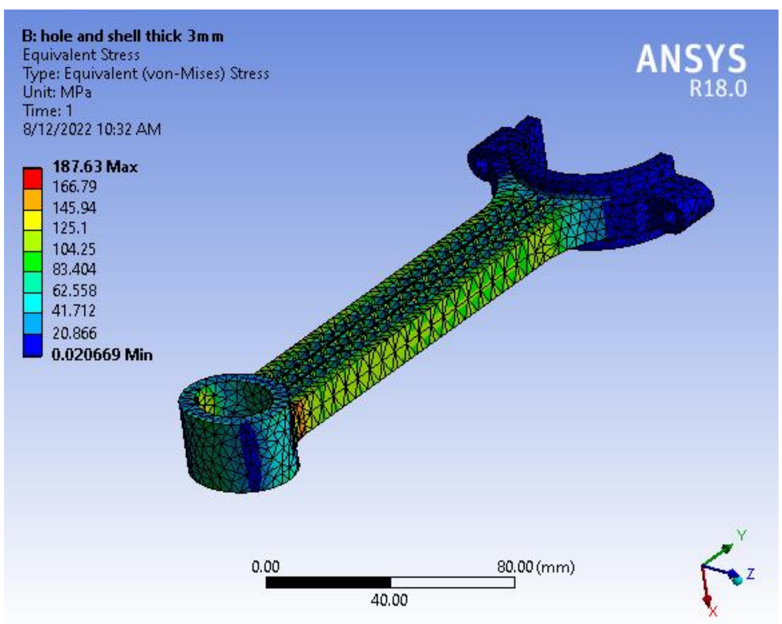

Figure 11 Equivalent Stress

Total Deformation

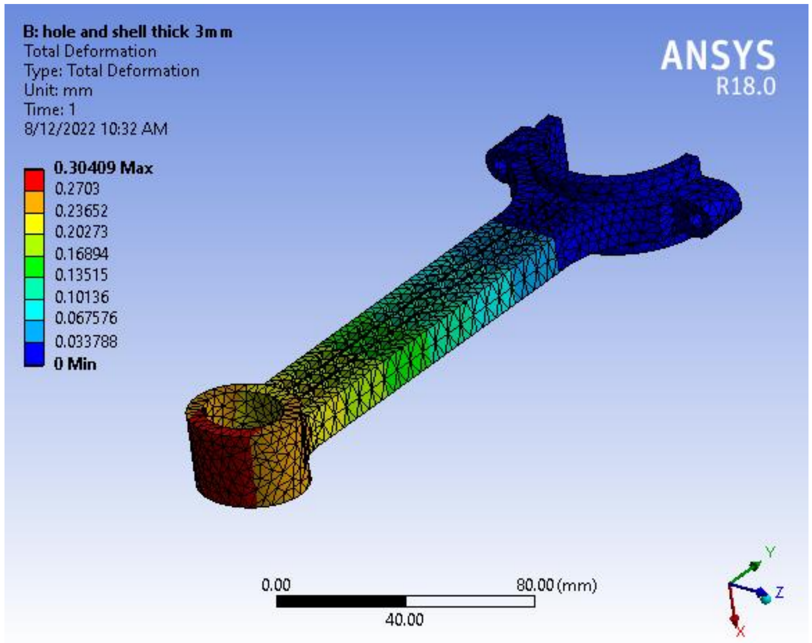

Figure 12 Total Deformation

Safety Factor

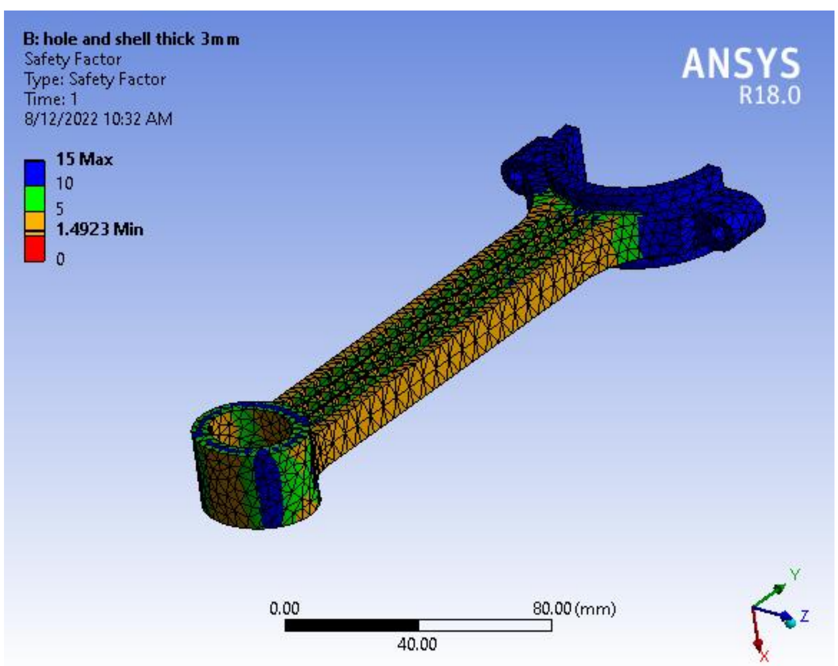

Figure 13 Safety Factor

Maximum Principal Elastic Strain

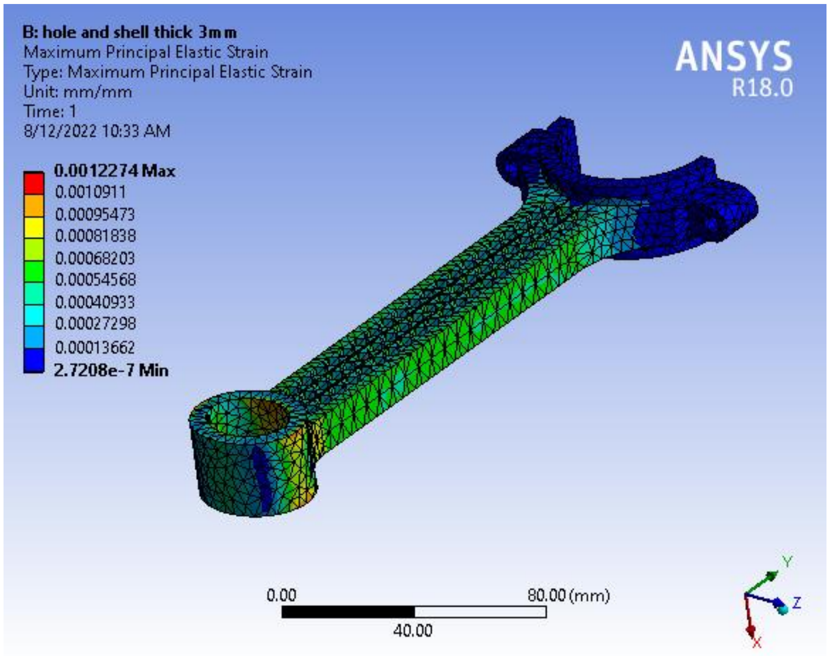

Figure 14 Maximum Principal Elastic Strain

Minimum Principal Stress

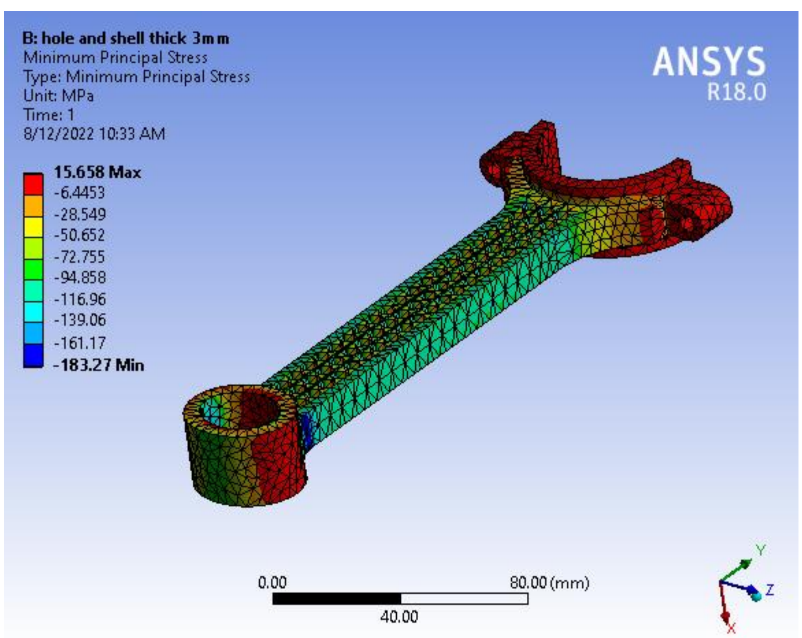

Figure 15 Minimum Principal Stress

7. CONCLUSIONS

In Static Structural with applied load as 14000N, the Equivalent Stress, Total Deformation and Minimum Principal Stress was found such as 182.01 Mpa maximum, 0.21037mm maximum and 13.514 Mpa maximum, which is shown in Figure 4, 5 and 9 respectively. In Static Structural (shell thick 3mm) with applied load as 14000N, the Equivalent Stress, Total Deformation and Minimum Principal Stress was found such as 187.63 Mpa maximum, 0.30409 maximum and 15.658 Mpa maximum, which is shown in Figure 11, 12 and 15 respectively. It was found that connecting rod with shell thick of 3 mm is best for applied load as 14000 N.

REFERENCES

[1] M. PavanKalyan, and N. Charishma. "Design and Comparative Analysis of Connecting Rod Using Composite Materials." International Journal of Engineering in Advanced Research Science and Technology 4 (2015).

[2] Raviraj Yashwant, and AbhayArunUtpat. "Analysis of Connecting Rod Used in Two Wheeler under Static Loading by FEA."International Journal of Engineering Trends and Technology (IJETT) – Volume 20 Number 1 – Feb 2015.

[3] Arun L.R, Mohammed Faheem. "Analysis and optimization of connecting rod using ALFASiC composites." International Journal of Innovative Research in Science, Engineering and Technology, Vol. 2, Issue 6, June 2013.

[4] P. H. J. Venkatesh. "Modelling and analysis of connecting rod using 4340 alloy steel and alsic-9."International Journal Of Engineering Sciences & Research Technology, 4.(12): December, 2015.

[5] Pranav G., and Santosh B. Jaju. "Analysis & optimization of connecting rod." Emerging Trends in Engineering and Technology (ICETET), 2009 2nd International Conference on. IEEE, 2009.

[6] Vinayak Kallannavar, Dr. B. S.Manjunath 'Dynamic Analysis Of Automotive ChassisUsing FEA' International Research Journal of Engineering And Technology (IRJET) E-ISSN: 2395-0056 Volume: 02 Issue: 09, Dec-2015.

[7] Cho, J. R., et al. "Nonlinear finite element analysis of swaging process for automobile power steering hose." Journal of Materials Processing Technology 170.1- 2 (2005): 50-57.

[8] Helnwein, Peter, et al. "A new 3-D finite element model for cord-reinforced rubber composites, application to analysis of automobile tires." Finite elements in analysis and design 14.1 (1993): 1-16.

[16]. Sung, S. H., and Donald J. Nefske. "A coupled structural-acoustic finite element model for vehicle interior noise analysis." Journal of vibration, acoustics, stress, and reliability in design 106.2 (1984): 314-318.

[9]. Pifko, A. B., and R. Winter. "Theory and application of finite element analysis to structural crash simulation." Computational Methods in Nonlinear Structural and Solid Mechanics. 1981. 277-285.

[10] Grujicic, M., et al. "Finite element analysis of the effect of up-armouring on the off-road braking and sharp-turn performance of a high-mobility multi-purpose wheeled vehicle." Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering 223.11 (2009): 1419-1434