Stress Analysis Of Connecting Rod For Diesel Engine With Different Materials

Dipanker Meena[1], Kamal Negi[2], Karan Pradhan[3]

[1,2,3] *Dept. of Mechanical Engineering, Delhi Technological University, New Delhi-110042*

***

Abstract - *The connecting rod is an important component of an engine assembly as it serves the purpose of transferring energy from the piston to the crankshaft. Its primary function is to convert the linear, back-and-forth motion of the piston into the rotary motion of the crankshaft. The connecting rod primarily undergoes tensile and compressive loading under the cyclic engine process. In this project, Static analysis was done on the connecting rod. Different materials are used for the analysis, like structural steel, titanium, and aluminum alloy. The study was done on several research papers, and after that, critical dimension data of the connecting rod was decided upon. A three-dimensional model of the connecting rod and its assembly was created on DS SolidWorks 2019. The model was then imported as a geometry parameter in Ansys 2023 R1 for finding von mises stress, Total Deformation, and safety factor. Then they were compared to choose the best material amongst them for the connecting rod.*

Key Words: Connecting Rod, Structural Analysis, Finite Element Analysis, Titanium, Steel, 42CrMo4.

1. INTRODUCTION

Connecting rods are of utmost importance in the functioning of an internal combustion engine. Their primary role is to convert the reciprocating motion of the piston into rotary motion in the crankshaft, enabling the vehicle's wheels to rotate. Comprising a small end connected to the piston and a big end attached to the crankshaft, the connecting rod experiences various forces during each rotation of the crankshaft. These forces include compression when the piston moves downward and tension when the piston moves upward. Any damage to the connecting rod, such as cracking or buckling, can result in engine immobilization. Such failures lead to substantial financial losses due to material damage and pose significant safety risks, as accidents resulting from these failures can have severe and potentially fatal consequences.

Many materials are used to make connecting rods. Every material has its advantages and disadvantages. They are used depending on the engine requirement, the purpose of the engine (work type), and other factors that matter in the engine's working.

In this research the best material for the connecting rod has been chosen by comparing the von mises stresses, total deformation and factor of safety of different materials on Ansys software. The connecting rod assembly has been shown in Fig –1.

1.1 Parts of the Connecting Rod

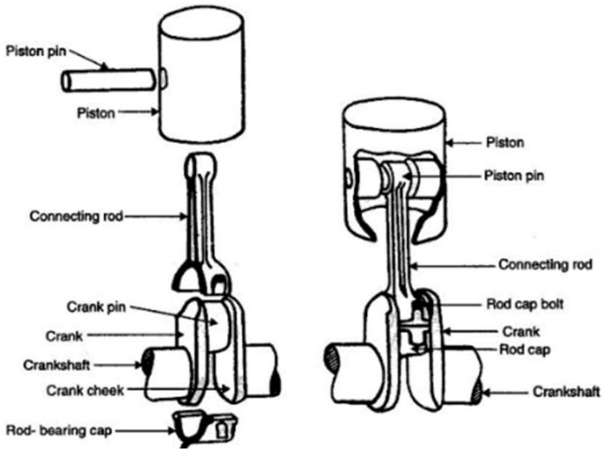


Fig -1: Connecting Rod

1. Small End
2. Big End
3. Bush bearing
4. Bearing Insert
5. Bolt and Nut
6. Shank
7. Crankshaft
8. Piston
9. Bearing cap

2. LITERATURE REVIEW

Comparisons were made between two different aluminum alloys and forged steel regarding von Mises stress, total deformation, and factor of safety [1].

The optimization of IC engine connecting rods involved the selection of forged steel or C-70, with a focus on size optimization and material choice [2][3].

ANSYS software was utilized to assess parameters like von Mises stress, strain, deformation, factor of safety, and weight reduction in two-wheeler pistons, indicating that aluminum alloy exhibited higher factors of safety, reduced stress, decreased weight, and greater stiffness compared to forged steel [4].

An optimization approach to connecting rods was proposed, involving an increase in torque of the bolt assembly to reduce stress amplitude [5].

Load and stress analysis of connecting rods for weight reduction was conducted [6].

Optimal designs for connecting rods were analyzed using ANSYS Workbench and CATIA V5R19, resulting in improved outcomes within permissible limits and safe stresses [7].

The design of a two-wheeler connecting rod was carried out analytically, followed by FEA for stress calculations and thermal analysis, leading to a significant reduction in weight [8].

Simulation studies involving SolidWorks and ANSYS were performed to assess von Mises stresses and frequencies of connecting rod materials, aiding in material selection [9].

The strength simulation analysis of a low-speed diesel engine's connecting rod was conducted using professional simulation software, followed by a static strength test using a servo universal testing machine [10].

3. MATERIALS USED

A variety of materials are utilized in the production of connecting rods which includes different grades of structural steel, aluminum, and titanium. Steel rods are the most commonly produced and employed due to their high strength and durability, making them suitable for both daily driving and endurance racing. However, one downside to using steel rods is their weight, which places added stress on the rotating assembly and consumes more power. This study examines the use of alternative materials for connecting rods and conducts an analysis of their properties. In this work, the below mentioned materials were taken for connecting rod simulation.

Structural Steel,

42CrMo4,

T6-7075 Aluminum,

Ti-6Al-4v,

C-70(SAE-AISI 1070) Carbon steel

4. METHODOLOGY

In this research work, a connecting rod was meticulously crafted employing the aid of DS SolidWorks 2019, a sophisticated computer-aided design (CAD) software application renowned for its prowess in generating intricate three-dimensional models of mechanical components, assemblages, and blueprints. Its pervasive adoption by engineers, designers, and manufacturers spanning a spectrum of industries, ranging from automotive and aerospace to consumer goods and industrial machinery, attests to its indispensable nature. This exceptional software offers an extensive array of utilities for the construction of intricate geometries, assemblages, and simulations, along with the generation of detailed technical drawings and animated representations. Capitalizing on a parametric design methodology, this software empowers users to effortlessly manipulate and revise models by effecting modifications to pertinent parameters or dimensions.

. The dimensions taken were those, which are of common use in the market. Each of the parts of the connecting rod was modeled and was merged together in the assembly.

Another software that was used for conducting Finite Element Analysis was Ansys Workbench 2023 R1. It is a software commonly used for stress/strain analysis and to determine the factor of safety of a proposed prototype.

The assembly of Solidworks is imported into Ansys as a geometry to perform FEA. Forces were applied on the connecting rod based on a number of research texts and

5. MODELING AND ANALYSIS OF CONNECTING ROD

Solidworks is a comprehensive 3D solid modeling software that provides users with the ability to design and analyze models in a virtual environment. It empowers designers, engineers, and professionals to conceptualize ideas and explore different design possibilities by creating accurate and detailed 3D models, assemblies, and drawings. By leveraging the capabilities of Solidworks, users can significantly reduce the time, effort, and expenses associated with physical prototyping. The software offers a streamlined and efficient approach to design, enabling users to visualize and evaluate their designs before moving forward with production.

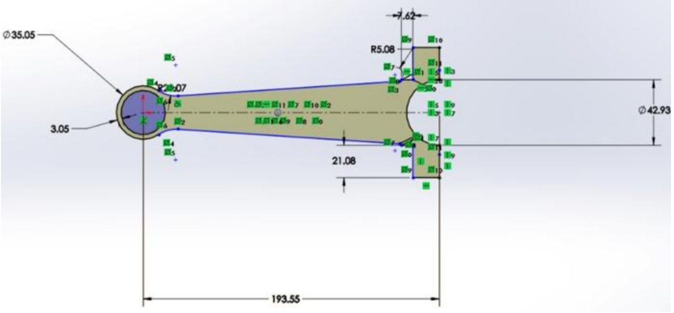


Fig -2: Dimensions of Connecting Rod

In this research, a connecting rod model was designed in SolidWorks 2019 software. The connecting rod was fabricated with a length of 193.55 mm, diameter of the smaller end, 35.05 mm, and the diameter of the bigger end, 42.93mm, shown in Fig -2.

Connecting rod assembly includes piston rod, cap end, nut, and bolt. The 3-Dimensional model of the assembly is shown in Fig -3.

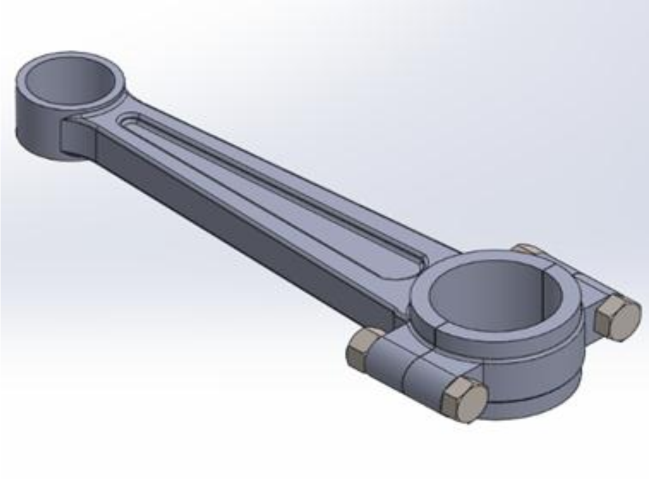


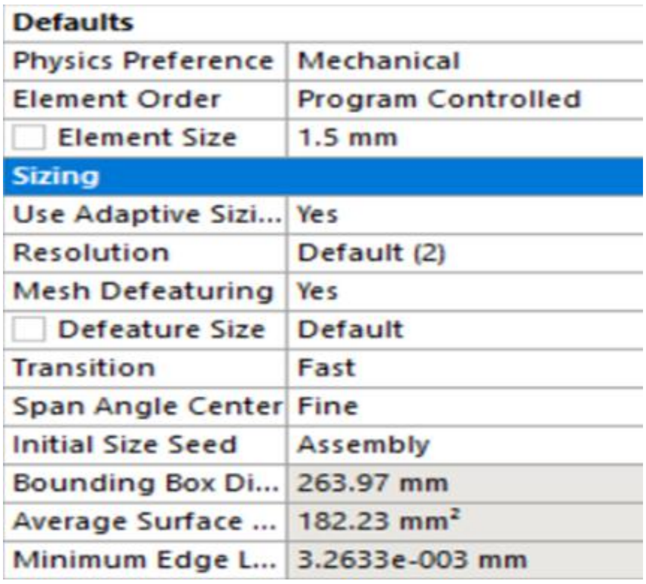
Fig -3: Connecting Rod Assembly

FEA, or Finite Element Analysis, is founded on the concept of dividing a given domain into a finite number of smaller regions called elements. By employing the variational or weighted residual method, FEA constructs a systematic approximate solution for these elements. This approach enables the reduction of the original problem, which involves an infinite number of unknowns, into a finite number of unknowns. The division of the domain into elements allows the representation of the unknown field variable through assumed approximating functions within each element.

ANSYS Workbench serves as the foundation for a comprehensive range of cutting-edge engineering simulation technologies in the industry. Its distinctive project schematic view seamlessly connects the entire simulation process, providing guidance to users at each step. With ANSYS Workbench, even intricate multi-physics analyses can be conducted effortlessly using a drag-and-drop approach. The platform facilitates automatic geometry sharing between fluid and structural analyses, streamlining data storage and enabling easy exploration of the effects of geometry modifications in both analyses. In the research, the Static Structural module of ANSYS was utilized.

5.1 Static Structural Analysis

Static analysis involves the meticulous examination of equilibrium conditions exhibited by objects when subjected to external forces. It encompasses both linear and nonlinear approaches, with the latter accommodating an extensive array of nonlinear phenomena such as significant deformations, plasticity, creep, stress stiffening, and contact elements. The current focus pertains specifically to static analysis, a comprehensive exploration of the effects resulting from sustained loading conditions on a given structure. Notably, this analysis disregards the intricate dynamics introduced by inertia and damping effects inherent in time-varying loads. The primary objective of static analysis lies in the precise determination of essential parameters including displacements, stresses, strains, and forces within structures or components, all of which arise from loads that lack substantial influence from inertia and damping mechanisms. Additionally, the process of meshing, an indispensable aspect of numerical analysis, assumes a significant role. Meshing involves the intricate segmentation of complex geometries or domains into simpler, more manageable elements. This segmentation facilitates enhanced analytical techniques and contributes substantially to the accuracy and efficiency of the overall analysis. To illustrate this concept further, Figure 4 provides a visual representation of intricate mesh details, while Figure 5 demonstrates the process of meshing specifically applied to the connecting rod under investigation.



Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
☐ Element Size	1.5 mm
Sizing	
Use Adaptive Sizi...	Yes
Resolution	Default (2)
Mesh Defeaturing	Yes
☐ Defeature Size	Default
Transition	Fast
Span Angle Center	Fine
Initial Size Seed	Assembly
Bounding Box Di...	263.97 mm
Average Surface ...	182.23 mm²
Minimum Edge L...	3.2633e-003 mm

Fig -4: Mesh Details

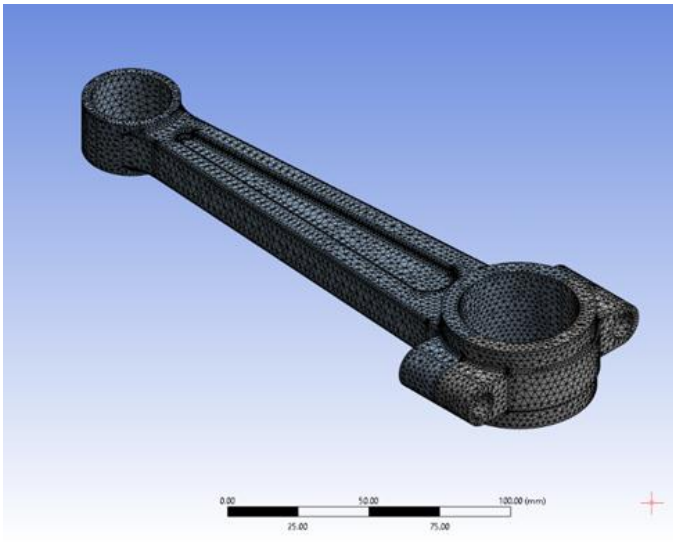


Fig -5: Connecting Rod Mesh Model.

Establishing boundary conditions is an essential and crucial step in Finite Element Analysis (FEA). This process involves defining constraints for the meshed model to simulate real-life scenarios accurately. The boundary conditions encompass fixed support, forces, and pressure that are applied to the model. By specifying these constraints, the analysis can determine the deformation, fatigue, stress, and strain values of the connecting rod while adhering to acceptable design limits. In the analysis of the connecting rod, a fixed support was applied at the bigger end, which is connected to the crankshaft. Additionally, pressure and forces were exerted at the smaller end, connected to the piston, as illustrated in Figure 5.

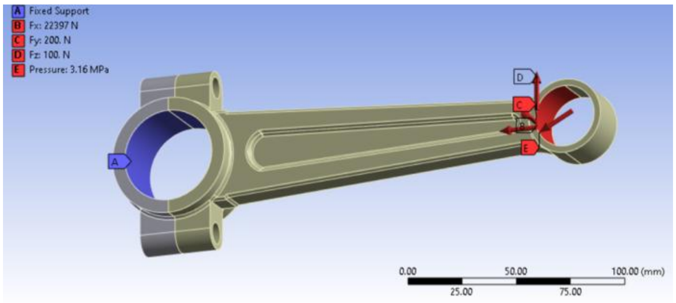


Fig -6: Application of Load/Forces

The Static Structural module of ANSYS has been used to perform the simulation on steel, aluminum and titanium alloys which are used nowadays for the production of connecting rods for vehicles. Following are the materials which were used in the research for simulation: structural steel, 42CrMo4, T6-7075 aluminum, Ti-6Al-4v and C-70.

1) Structural Steel

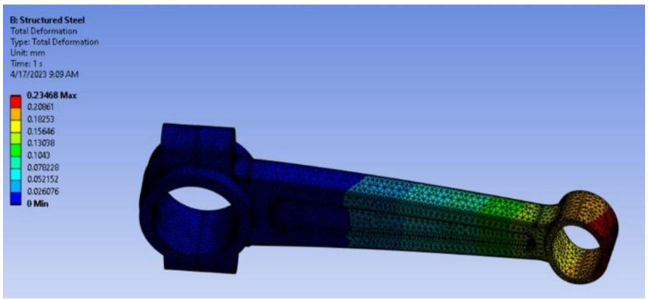


Fig -7: Total Deformation

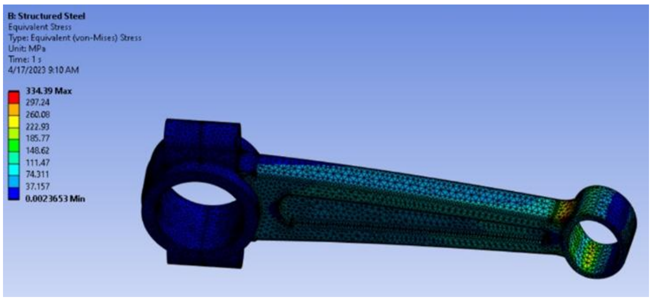


Fig -8: Equivalent Stress

Total Deformation and Equivalent stress of Structural Steel has been shown in Fig -7 and Fig -8 respectively.

2)42CrMo4

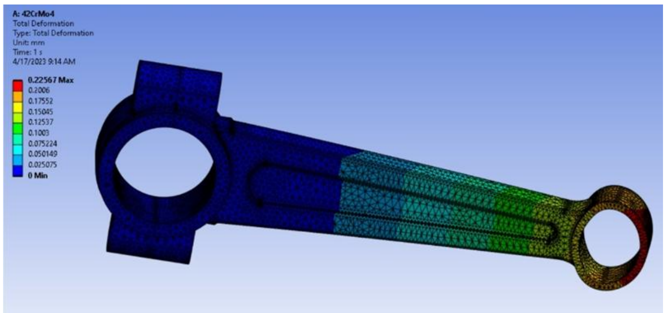


Fig -9: Total Deformation

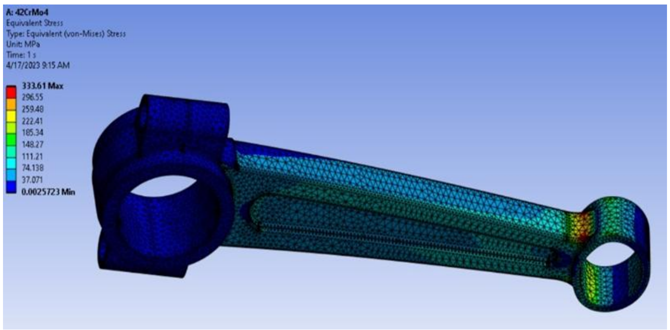


Fig -10: Equivalent Stress

Total Deformation and Equivalent stress of 42CrMo4(an engineering alloy steel containing chromium and molybdenum) has been shown in Fig -9 and Fig -10 respectively.

3) T6-7075 Aluminum

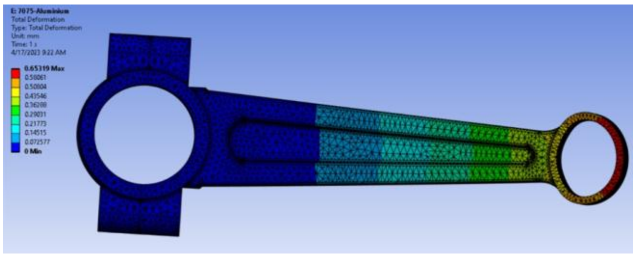


Fig -11: Total Deformation

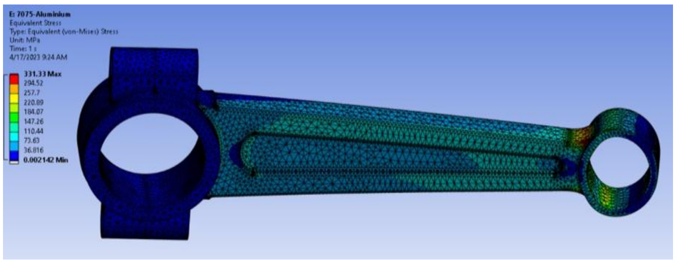


Fig -12: Equivalent Stress

Total Deformation and Equivalent stress of T6-7075 Aluminium(an Aluminium alloy) has been shown in Fig -11 and Fig -12 respectively.

4) Ti-6Al-4v

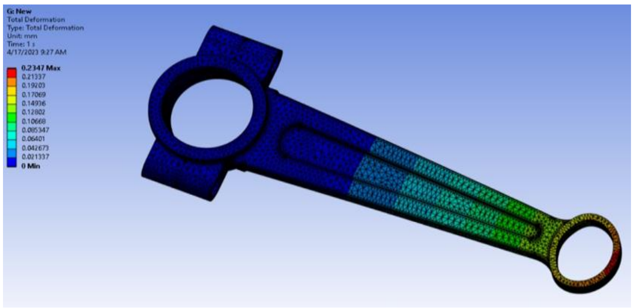


Fig -13: Total Deformation

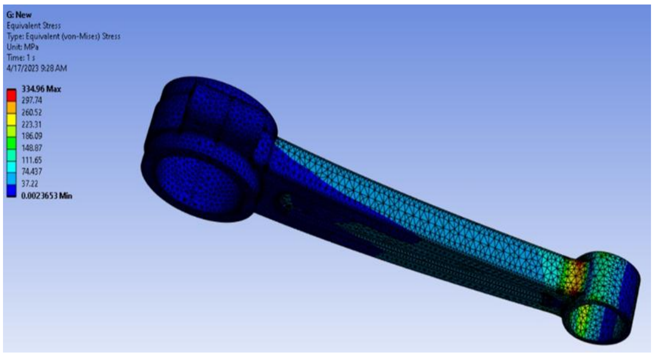


Fig -14: Equivalent Stress

Total Deformation and Equivalent stress of Ti-6Al-4v(a titanium alloy composed of 90% titanium, 6% aluminum and 4% vanadium) has been shown in Fig -13 and Fig -14 respectively.

5) C-70(SAE-AISI 1070) Carbon steel

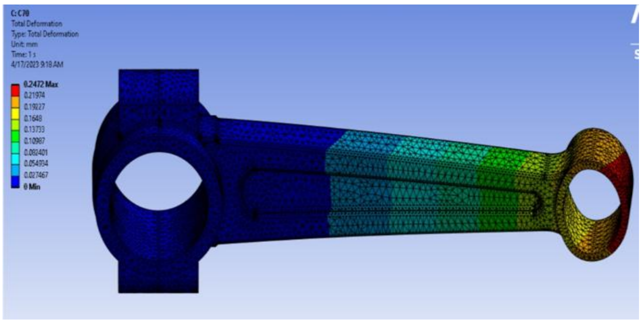


Fig -15: Total Deformation

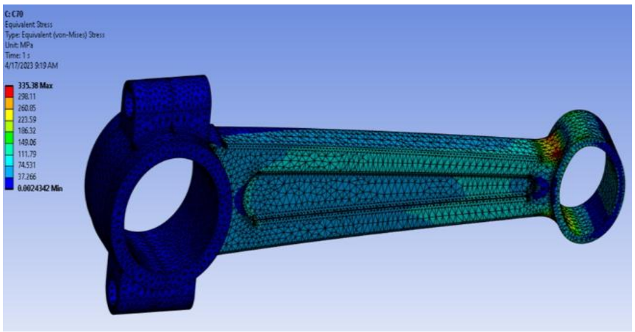


Fig -16: Equivalent Stress

Total Deformation and Equivalent stress of C-70(Carbon Steel) has been shown in Fig -15 and Fig -16 respectively.

6. RESULTS AND DISCUSSIONS

Table -1: Comparison of Values

Materials	**Total Deformation (mm)**	**Equivalent Stress (MPa)**	**Safety Factor**
Structural Steel	0.23468	334.39	0.7476
42CrMo4	0.22567	333.61	2.6977
T6-7075 Aluminum	0.65319	331.33	1.5181
Ti-6Al-4v	0.23410	334.96	3.2840
C-70	0.24720	335.38	1.4312

A comparison of values of total deformation, equivalent stress, and factor of safety for five different materials has been shown in Table 1. From the table it can be concluded that the minimum total deformation has been shown by 42CrMo4(Steel alloy) while the maximum total deformation has been shown by T6-7075 Aluminum.

The minimum equivalent stress has been shown by T6-7075 Aluminum and maximum equivalent stress has been shown by C-70(Carbon steel).

The highest factor of safety has been shown by Ti-6Al-4v (a Titanium alloy) while Structural steel has the lowest factor of safety.

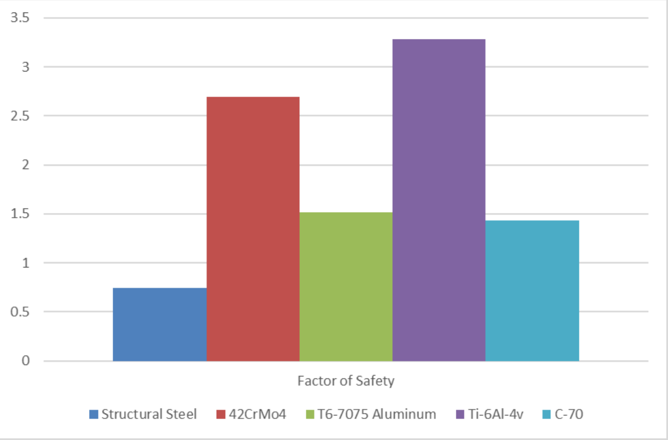


Chart -1: Factor of Safety

7. CONCLUSIONS

From the static analysis the stress is found maximum at the small end of the connecting rod which is connected to the piston.

It has been found that titanium alloy is the best material for manufacturing connecting rods as it has a high factor of safety although it has more weight than aluminum alloy. The only thing that is a barrier to implement Titanium alloy is cost, availability, and its repair.

42CrMo4 is the second choice for production as it has the lowest value of total deformation and has a high factor of safety after Titanium alloy.

Structural steel is not effective regarding mass, and it increases the weight of the connecting rod. It also has the lowest factor of safety among the chosen materials.

ACKNOWLEDGEMENT

This Bachelor of Technology dissertation titled "Static Analysis of Connecting Rod" has been carried out under the guidance and supervision of Prof. R.C Singh and the aegis of the Mechanical Engineering Department, Delhi Technological University.

In this research journey, we came across several people who helped and traveled with us to achieve this ambition of our life. Therefore, for this achievement of ours, we should express our sense of gratitude towards those persons who unconditionally supported, helped, and encouraged us in all situations. We are highly thankful to the Almighty for blessing us and giving us the strength, energy, and patience to overcome all the difficulties and reach this ambition.

Firstly, we would like to thank **Prof. S. K. Garg** (Head of the Department) for giving us the opportunity to work on this project.With our profound gratitude and deep regards, we would also like to thank **Prof. R.C. Singh** for their personal input, enthusiasm, meticulous manuscript checking, constructive comments, and remarks. Except for the vision they set for us, they helped us conceptualize the project and helped in the actual building and procedure to complete it.

We thank **Ms. Yamika Patel** for their continuous support and guidance throughout the project.

We want to thank our professors for their guidance and patience and for imparting in-depth knowledge about the subjects, which helped complete our project.

REFERENCES

[1] R A Savanoor, Abhishek Patil, Rakesh Patil and Amit Rodagi, Int. J. Mech. Eng. & Rob. Res. 2014, ISSN 2278 – 0149 Vol. 3, No. 3.

[2] Kumbha Sambaiah, DR. Rama Rao and DR. M Mahesh "Optimization for thermal analysis connecting rod of internal combustion engine by using two different materials", International Journal of Applied Research 2017; 3(12): 278-282

[3] Ashish Kumar and Er. Shubham Parmar "Analytical Study and Design Analysis of Connecting Rod of Mahindra Pijo by Finite Element Analysis", International Journal of Scientific Development and Research (IJSDR) www.ijsdr.org

[4] Mr. H D. Nitturkar, Mr. S M. Kalshetti and Mr. A R. Nadaf "Design and Analysis of Connecting Rod using Different Materials", International Research Journal of Engineering and Technology (IRJET).

[5] Griza, S.; Bertoni, F.; Zanon, G.; Reguly, A.; Strohaecker, T. Fatigue in engine connecting rod bolts due to forming laps. Eng. Fail. Anal. 2009, 16, 1542–1548.

[6] Ramani et al Int. Journal of Engineering Research and Applications, Vol. 4, Issue 3(Version 1), March 2014, pp.216- 220

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

[8] Vivek C. Pathade, Bhumeshwar Patle and Ajay N. Ingale, "Stress analysis of I.C engine connecting rod by FEM", International Journal of Engineering and Innovative Technology, Vol-1, Issue-3, pp. 12-15, March 2014

[9] Chinnari Natraj, Bantu Sandhya, Agatha Naveen, Seera vivek and Seera N. Kumar,"Design, Static and Modal analysis of connecting rod of four stroke Spark Ignition engine", Anil Neerukonda Institute of Technology and Science, India,2022.

[10] W. Gao, G. Wang, J. Zhu, Z. Fan, Xiaobo "Structural Optimization Design and Strength Test Research of Connecting Rod Assembly of High-Power Low-Speed Diesel Engine" College of Power and Energy Engineering, Harbin Engineering University, Harbin 150001, China,2022.