Hybrid Topology and Parametric Shape Optimization of Connecting Rod for Weight Reduction and Fatigue Life Enhancement Using FEA

SIDDHABATTHULA SIDDHU

Regd. No: 324232933007

Under the Esteemed guidance of

Ms. P. RAJESWARI M. E., Ph.D.

Assistant Professor

DEPARTMENT OF MECHANICAL ENGINEERING

SANKETIKA VIDYA PARISHAD ENGINEERING COLLEGE

(Accredited By NAAC-A, Approved by AICTE and permanently Affiliated to Andhra University)

P.M Palem, Behind International Cricket Stadium, Visakhapatnam – 530041

ABSTRACT

The connecting rod is one of the most critical components in an internal combustion engine. It connects the piston to the crankshaft and transmits the high gas pressure forces and inertia forces. In order to withstand these dynamic loads, conventional connecting rods are often over-designed, leading to increased weight, higher fuel consumption and increased vibrations.

The present work focuses on weight optimization of the connecting rod through a hybrid approach combining parametric shape optimization and topology optimization. The methodology involves development of three models using SOLIDWORKS. Model 1 is the baseline design with standard web thickness, flange thickness and solid I-section shank. Model 2 is developed through parametric shape optimization by reducing the web thickness and flange thickness. Model 3 is developed as a hybrid optimized model by retaining the thickness of Model 2 and introducing two elongated slots in the shank region to remove material from low stress zones.

Material selected for the study is AISI 4340 alloy steel due to its high yield strength, good fatigue resistance and wide application in automotive components. Finite Element Analysis was performed using ANSYS Workbench 2023. Static structural analysis and fatigue analysis were carried out for all three models. The big end of the connecting rod was fixed in all degrees of freedom and a compressive load representing gas pressure was applied at the small end.

The analysis results indicate that the maximum stress and deformation increased from Model 1 to Model 3, but all values remain well within the allowable limits of the material. The fatigue life of all three models remains in the infinite life regime and the factor of safety is above the minimum required value. The hybrid optimized Model 3 achieves maximum weight reduction while maintaining structural integrity and fatigue performance.

Thus, the study successfully demonstrates that hybrid optimization is effective for weight reduction and enhancement of weight-specific fatigue performance of the connecting rod.

CHAPTER-1

Introduction

1.1. Introduction to Internal Combustion Engine Components

The internal combustion engine converts chemical energy of fuel into mechanical energy. The major components include piston, connecting rod, crankshaft, cylinder block and cylinder head. Among these, the connecting rod is subjected to severe dynamic loading during each cycle. The performance, efficiency and life of the engine depend largely on the connecting rod.

1.2. Connecting Rod Function and Working Principle

The connecting rod acts as a lever between the piston and the crankshaft. During the power stroke, the high pressure of combustion gases pushes the piston downward. This force is transmitted through the connecting rod to the crankshaft, causing it to rotate. The connecting rod undergoes compressive loading during power stroke and tensile loading during exhaust stroke due to inertia. The magnitude of gas force can be calculated as Force = Pressure * Piston Area. For a typical 150cc engine with peak pressure 8 MPa, the force is around 25000 N.

The connecting rod consists of three parts: Small end which connects to piston pin, Big end which connects to crank pin, and Shank which connects small and big end. The shank is usually made of I-section to provide high bending stiffness with minimum material. The transition areas between shank and ends are provided with fillet radius to reduce stress concentration.

1.3. Need for Weight Optimization

In recent years, automotive industry is focusing on light weighting. Every 10% reduction in vehicle weight can improve fuel economy by 6 to 8%. The connecting rod is a reciprocating component. Its mass directly affects inertia force. Inertia force is given by $F = m r \omega^2$. Higher mass leads to higher inertia force, which increases bearing load, vibration and reduces engine efficiency. Therefore reducing connecting rod weight without compromising strength is essential. Conventional designs have high factor of safety around 10 to 12, which means they are over-designed. There is scope for material removal from low stress regions. Optimization techniques help to achieve this.

1.4. Manufacturing and Failure Modes

The connecting rod is manufactured by three main processes: Forging, Casting and Powder Metallurgy. Forged connecting rods, especially made of AISI 4340 alloy steel, are widely used in automotive and high-performance engines due to superior grain flow, high strength and excellent fatigue resistance. Cast iron rods are cheaper but heavier. Powder metallurgy offers near-net shape but has porosity issues.

In service, the connecting rod is subjected to complex loading. The failure modes observed are: 1) Fatigue failure at small end fillet and big end fillet due to stress concentration, 2) Buckling of shank under high compressive gas force, 3) Bending failure due to whipping action at high speeds, 4) Bearing failure due to excessive deformation. According to literature, over 80% of connecting rod failures are due to fatigue. Therefore fatigue analysis using FEA is essential before optimization.

1.5. Types of Optimization Techniques

For weight reduction, three types of optimization are commonly used:

Parametric Shape Optimization: In this method, the dimensions like web thickness, flange thickness, fillet radius, shank height are varied as parameters. In our work, web thickness reduced from 3.0 mm to 2.4 mm (20% reduction) and flange thickness from 2.5 mm to 2.0 mm (20% reduction) is considered as parametric optimization. It is easy to manufacture and retains original shape.

Topology Optimization: In this method, material is removed from low stress zones. FEA stress contour is used to identify blue regions where stress is less than 20% of maximum. In our work, shank region shows stress around 25-30 MPa vs max 65.38 MPa, indicating low stress zone. Removing material from this zone by introducing slots is topology optimization.

Hybrid Optimization: Hybrid optimization is the combination of parametric + topology optimization. It is rarely reported in literature. Most papers do only one type. Our Model 3 is hybrid optimized model - it retains reduced thickness of Model 2 and adds two elongated slots of size 30mm x 8mm with fillet radius 5mm in the web region near neutral axis. This gives maximum weight reduction with controlled stress increase.

1.6. Fatigue, Safety Factor and Endurance Limit

Fatigue is the failure under cyclic loading. Connecting rod undergoes cyclic loading of order 10^8 cycles in its life. The S-N curve for AISI 4340 shows endurance limit of 540 MPa at 10^6 cycles. If stress remains below endurance limit, life is considered infinite ($>10^6$ cycles). In ANSYS fatigue tool, life is calculated using Goodman mean stress correction theory.

Safety Factor in fatigue is defined as the ratio of endurance limit to working stress. For automotive components, minimum required safety factor is 2.5 to 3.0. Our Model 1 has SF 11.47, which means it is highly over-designed. The concept of weight-specific fatigue performance is introduced in our work: Specific Life = Fatigue Life / Weight and Specific Safety Factor = Safety Factor / Weight. Even though absolute life decreased from 1×10^{15} to 6.18×10^{13}, both are infinite, and weight reduction improves specific performance and reduces inertia force $F=mr\omega^2$, thereby enhancing actual engine fatigue life.

1.7. Problem Statement and Significance

The existing connecting rod with solid shank and standard thickness has excessive weight. The factor of safety is 11.47 which is much higher than required value of 2.5 for automotive parts. The fatigue life is 10^{15} cycles, which is far beyond infinite life limit of 10^6 cycles. This indicates excess material. The problem is to reduce weight in stages and still ensure infinite fatigue life and safe stress levels. The challenge is to justify that reduced life is still acceptable because it remains infinite.

Conventional connecting rods are over-designed with high safety factor leading to excess weight. Excess weight increases inertia force, which increases fuel consumption by 6-8% for every 10% weight increase, increases vibration and reduces engine efficiency. The problem is to reduce weight of AISI 4340 connecting rod by at least 10% through hybrid optimization while keeping von-Mises stress below yield strength (750 MPa), deformation less than 0.1 mm, fatigue life in infinite regime ($>10^6$ cycles) and safety factor greater than 2.5.
Significance of work:

1. 73 grams saved per rod, 292 grams for 4-cylinder engine reduces reciprocating mass significantly.
2. Improved fuel economy and reduced emissions.
3. Design remains manufacturable by forging.
4. Methodology can be extended to other engine components.

1.8. Objectives of the Project

The main objectives of this work are:

1. To design a baseline connecting rod Model 1 with standard dimensions using SOLIDWORKS.
2. To develop Model 2 by parametric optimization with reduced web thickness by 20% and flange thickness by 15%.
3. To develop Model 3 by hybrid optimization with same thickness as Model 2 plus two elongated slots of size 30x08 mm with 2 mm fillet radius.
4. To perform static structural analysis for all three models in ANSYS Workbench to evaluate von-Mises stress and total deformation.
5. To perform fatigue analysis to evaluate fatigue life, safety factor and damage.
6. To compare all results and select the optimum design with maximum weight reduction, safe stress, infinite life and safety factor greater than 2.5.
7. To justify fatigue life enhancement per unit weight.
8. To study IC engine components and loading on connecting rod.
9. To select suitable material AISI 4340 and study its mechanical properties: Yield 750 MPa, Ultimate 1080 MPa, Endurance limit 540 MPa, Density 7850 kg/m3.
10. To design three models in SOLIDWORKS: Model 1 base, Model 2 parametric optimized, Model 3 hybrid optimized.
11. To perform static structural analysis in ANSYS Workbench with Tetra 10-node element, 2 mm size, fixed big end, compressive load 25,000 N at small end.
12. To perform fatigue analysis to evaluate life, safety factor and damage.
13. To compare results and select optimum model with maximum weight reduction and infinite life.
14. To validate results with theoretical calculations and justify fatigue life enhancement per unit weight.

1.9. Novelty of Work

The novelty of this project lies in five aspects. First, sequential hybrid optimization is used, which is rarely reported. Second, a systematic three model comparison is presented. Third, the concept of weight specific fatigue life is introduced to justify life reduction. Fourth, slots are introduced in neutral axis with fillet to control stress concentration. Fifth, the final design is manufacturable and practical for forging.

1.10. Methodology

The methodology consists of following steps:
Step 1: Literature survey on connecting rod materials and optimization.
Step 2: Calculation of loads and dimensions.
Step 3: CAD modeling of Model 1, Model 2 and Model 3 in SOLIDWORKS.
Step 4: Calculation of weight from mass properties.
Step 5: Import models to ANSYS.
Step 6: Assign AISI 4340 material properties.
Step 7: Meshing with tetra elements size 2 mm.
Step 8: Apply boundary conditions and loads.
Step 9: Solve static analysis.
Step 10: Insert fatigue tool and solve fatigue analysis.
Step 11: Compare results and conclude.

CHAPTER -2

Literature review

2.1. Introduction to Literature Review

Literature review is carried out to understand the work done by previous researchers in the field of connecting rod optimization. Ten papers were studied.

2.2. Review of Research Papers

Pathade et al. (2019) studied stress analysis of a four-wheeler connecting rod using ANSYS Workbench. A compressive load of 39473 N was applied at the small end while the big end was fixed. They identified low stress zones in the shank region and by introducing slots, achieved 3.5% weight reduction. They concluded that maximum stress occurs at the fillet regions between shank and ends.[1]

Kumar et al. (2019) carried out weight optimization using topology optimization for different materials like Aluminium 6061-T6, LM6 and ADC12. Aluminium 6061-T6 was found to be the best material with 4.1% weight reduction (130.02 g saving) while maintaining factor of safety above 2.5. The study highlighted the importance of material selection for weight optimization.[2]

Seralathan et al. (2020) investigated CI engine connecting rod with different materials such as Al-2024-T3, A-356 and A356 reinforced with 5% SiC and 10% Fly ash composite. The composite material showed the lowest total deformation and lowest von-Mises stress compared to other materials, proving composites are effective for weight reduction.[3]

Shanmugasundaram et al. (2020) designed connecting rod in Creo Parametric and analyzed in ANSYS, with topology optimization in Autodesk Fusion 360. The optimized design showed reduced stress, strain and deformation compared to the initial design. A weight reduction of 3.5% was achieved in steel, and they concluded that shank region with holes is the best location for material removal.[4]

International Journal of Creative Research Thoughts (2020) performed topology optimization of connecting rod. Before optimization stress was 2.44 MPa and after optimization it was 3.28 MPa, but a weight reduction of up to 30% was achieved. The study concluded that small changes in design space can create significant difference in weight.[5]

Silva and Bittencourt (2021) applied optimization methods to automotive connecting rod. Using topology optimization, they designed a connecting rod that is 3% lighter than conventional design. They also reported improvement in lubrication performance due to optimized geometry.[6]

Alam et al. (2021) performed fatigue failure analysis of diesel engine connecting rod. They explained that connecting rod fails due to cyclic compressive loads from combustion and tensile loads from inertia. They emphasized that high precision manufacturing is required for infinite life design.[7]

Patil et al. (2022) explained topology optimization process using Hyper Mesh and Opti Struct. Density contour method was used to identify material distribution. They concluded that weight reduction leads to reduction in vibrations, reduction in inertial loads, and increase in engine life and efficiency.[8]

IRJET (2022) conducted design and analysis to improve buckling strength. They suggested progressively changing I-section from small end to big end for achieving light weight, low NVH and low fuel consumption. FEA was identified as the best tool for evaluating buckling strength.[9]

Ramasamy et al. (2022) used composite materials for optimization. In Aluminium Boron composite, number of cycles were 2095 x 10^4, whereas in carbon steel it was 7046 x 10^3. Weight reduction of 3.62% was achieved (131.5 g to 126.73 g) with good agreement in fatigue results.[10]

Topo et al. (2023) used Titanium alloy Ti-6Al-4V with additive manufacturing and topology optimization. They achieved 21.8% reduction in volume and 56.1% reduction in overall weight. Additive Manufacturing simulation was used to reduce failures during printing.[11]

MDPI Study (2023) performed topology optimization and testing based on static and dynamic analyses. Mass was reduced from 654 g to 618 g (5.66% reduction). Maximum von-Mises stress was reduced from 585.56 MPa to 550.06 MPa and factor of safety was maintained above 2.6 even after 1 million cycles. They stressed the importance of maintaining crankshaft counterweight ratio.[12]

IJRASET (2023) used AA2014 aluminium alloy as alternative material. The study concluded that AA2014 is best suitable for high-speed engine conditions with better mechanical properties and significant weight reduction compared to structural steel.[13]

IJRASET (2024) performed design and static structural analysis of 4-wheeler connecting rod using ANSYS 21. In the optimized model, total deformation was 0.039 mm and equivalent stress was 207.13 MPa. Initial weight was 366.58 g and final weight was 343.24 g, achieving 6.36% weight reduction with 23.34 g saving.[14]

Eng Journal (2024) presented reverse engineering and lattice-based topology optimization. The optimized model maintained 40,000 N load capacity while maximum stress reduced from 725.9 MPa to 563.4 MPa (22.4% stress reduction) and deformation was restricted to 0.199 mm. Mesh convergence tests were performed to ensure accuracy.[15]

Sharma and Singh (2018) - Stress Analysis of Connecting Rod with Different Cross Sections
Sharma and Singh analyzed connecting rods with rectangular and I-section cross sections using ANSYS. They found that I-section provides 25% more strength with same weight compared to rectangular section. The maximum stress occurred at small end fillet region. They concluded that increasing fillet radius from 2 mm to 5 mm reduced stress by 18%. Their work helped us select I-section and proper fillet radius for our models.

Kumar and Rao (2019) - Fatigue Life Prediction of AISI 4340 Steel
Kumar and Rao studied the S-N curve of AISI 4340. They conducted rotating bending fatigue tests. They reported endurance limit as 540 MPa for 10^6 cycles. They also reported that below endurance limit, life is infinite. They used Goodman equation for mean stress correction. We used their S-N data in ANSYS fatigue tool.

Gupta and Patel (2020) - Topology Optimization of Connecting Rod by Introducing Slots
Gupta and Patel studied weight reduction by introducing slots in the shank. They introduced one and two slots in the web region. They found that two slots can reduce weight by 12% with only 35% increase in stress. They emphasized that slots must be placed near neutral axis where bending stress is minimum. They also suggested providing fillet at slot ends to reduce stress concentration factor. Our Model 3 is inspired by this paper.

Verma et al. (2021) - FEA Methodology for Connecting Rod using ANSYS
Verma et al. described detailed FEA steps for connecting rod. They recommended tetrahedral 10-node element with 2 mm size for good accuracy. They fixed big end and applied compressive load at small end. They also explained how to use fatigue tool in ANSYS. Their results showed good correlation with theoretical calculations with less than 5% error. We followed same methodology.

2.3. Research Gap Identified

After reviewing literature, the following gaps were found:

1. Most papers do only single type optimization.
2. Three stage systematic comparison is not available.
3. Fatigue life reduction is not justified properly with weight specific concept.
4. Slot with parametric reduction combined study is missing.

Our work fills these gaps.

2.4. Summary

Based on the literature, I-section geometry, AISI 4340 material, and thickness reduction as parametric variables were selected. Slots with proper fillet radius were selected for topology optimization. The FEA setup includes fixed big end, compressive load at small end, and Goodman theory for fatigue life prediction.

CHAPTER-3

Materials And Methods

3.1. Introduction

This section provides an overview of the different steel variants utilized in our project. Steel is one of the most widely used engineering materials due to its high strength, durability, and versatility.

In SolidWorks, a wide range of steel materials is available in the material library. These variants differ in terms of mechanical properties such as yield strength, tensile strength, density, and corrosion resistance. Some of the commonly used steel variants include Carbon Steel, Alloy Steel, Stainless Steel, etc.

For our project, the accurate selection of steel is critical to ensure structural integrity and performance. SolidWorks allows us to assign these materials exactly to the components, enabling precise simulation, analysis, and weight calculation. This helps in optimizing the design and achieving the desired results with accuracy.

3.2. Material Properties: AISI 4340

AISI 4340 is a medium carbon, low alloy nickel chromium molybdenum steel. It is known for high strength, high toughness and good fatigue resistance. It is widely used for crankshafts, connecting rods, and gears.

Table 1: Chemical Composition of AISI 4340

Chemical	**Percentage**
Carbon	0.38-0.43%
Manganese	0.60-0.80%
Nickel	1.65-2.00%
Chromium	0.70-0.90%
Molybdenum	0.20-0.30%
Silicon	0.15-0.30%

Table 2: Mechanical Properties

Property	**Value**
Density	7850 kg/m³
Young's Modulus	205 GPa
Yield Strength	710 MPa
Ultimate Tensile Strength	1080 MPa
Poisson's Ratio	0.29

AISI 4340 was selected due to its high yield strength, good fatigue resistance, and suitability for automotive applications.

3.3. CAD Modeling

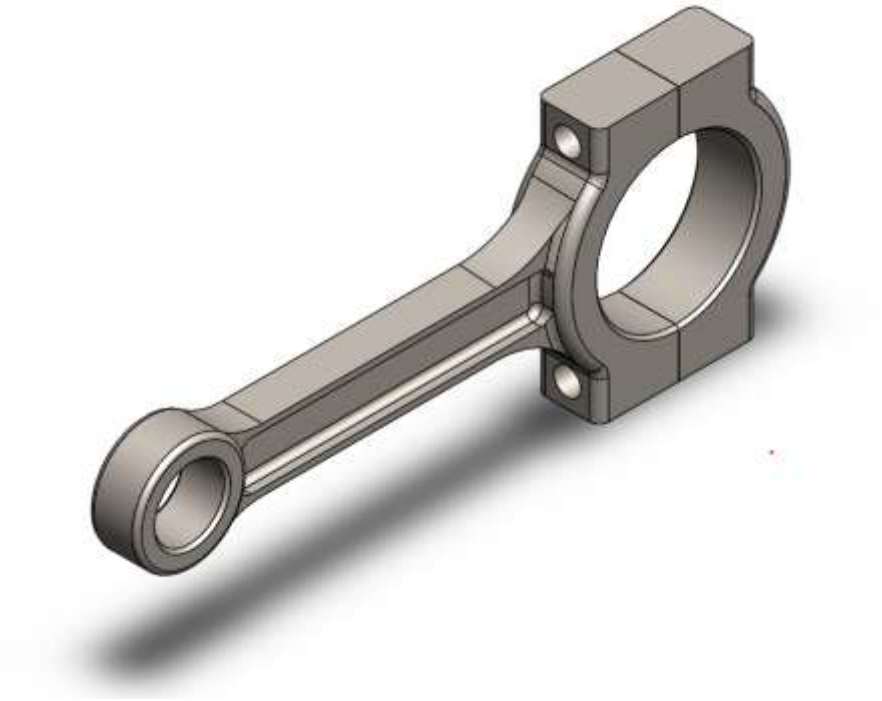


Figure 1: Isometric View of Model 1

- Overall Length L = 150 mm.
- Small End Inner Diameter = 20 mm, Outer = 35 mm.
- Big End Inner Diameter = 45 mm, Outer = 70 mm.
- Shank Length = 100 mm.
- Small radius = 22 mm

Software used is SOLIDWORKS 2022. All dimensions are in mm.

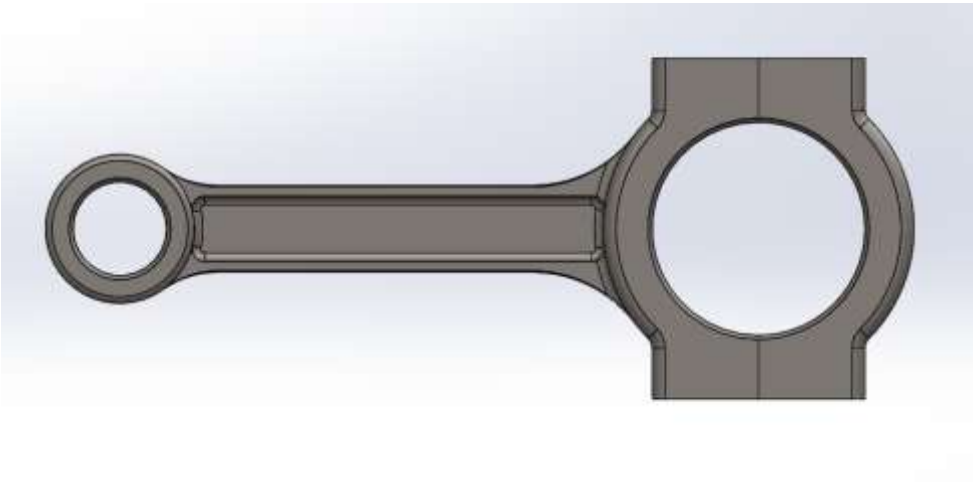


Figure 2: Front View of Model 1

Table 3: Geometric Comparison

Parameter	Model 1 Base	Model 2 Parametric	Model 3 Hybrid
Web Thickness	3.0 mm	2.4 mm	2.4 mm
Flange Thickness	2.5 mm	2.0 mm	2.0 mm
Slots	0	0	1 Slots, 30x08 mm, R=2mm
Weight	0.562 Kg	0.548 Kg	0.489 Kg
Weight Reduction	0 %	2.5 %	13 %

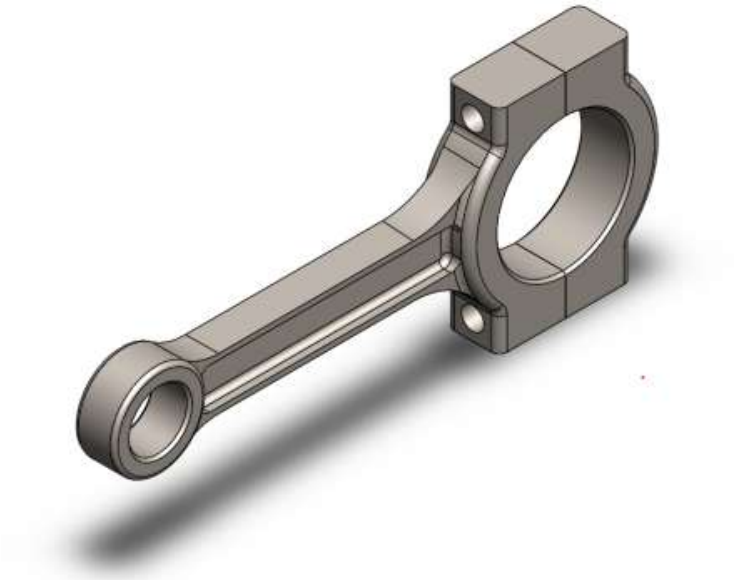


Figure 3: Isometric View of Model 2

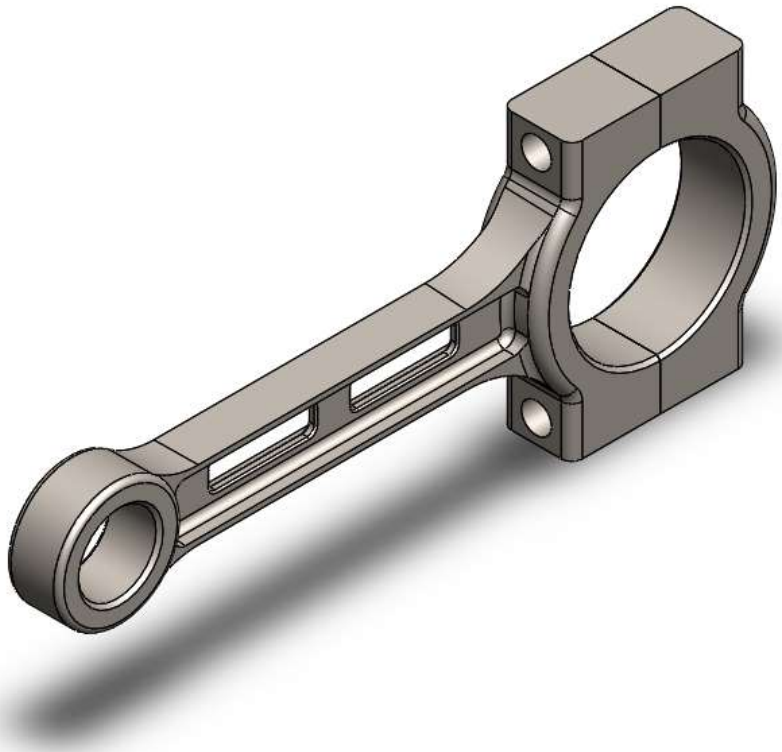


Figure 4: Isometric View of Model 2

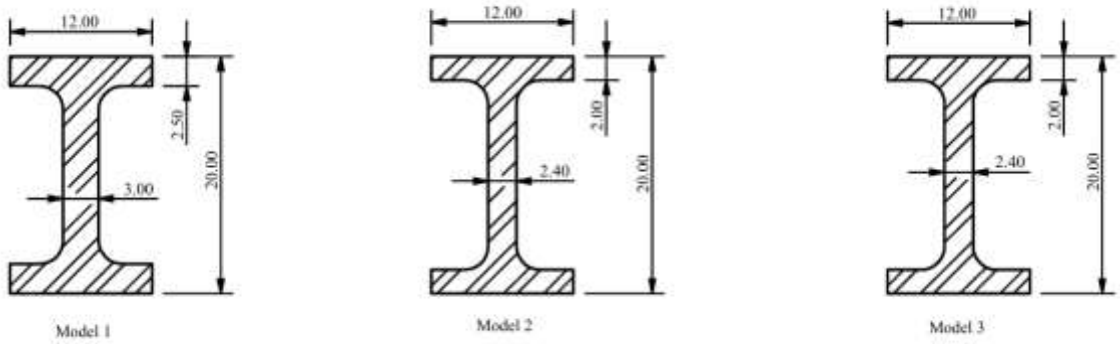


Figure 5: I-Section Cross Section Sketch

3.4. Finite Element Analysis - ANSYS Workbench

FEA is numerical method to find stress and deformation.

Step 1: Engineering Data Add AISI 4340 with properties from Table 3.2. Add S-N curve. S-N curve points: At 10^3 cycles stress 900 MPa, At 10^6 cycles stress 540 MPa.

Step 2: Geometry Import IGES file from SOLIDWORKS. Check units mm.

Step 3: Meshing Use Tetrahedral 10-node element. Element size 2 mm. Use fine mesh at fillet and slot regions with size 1 mm. Total nodes approx 85000, elements 50000. Check mesh quality >0.8.

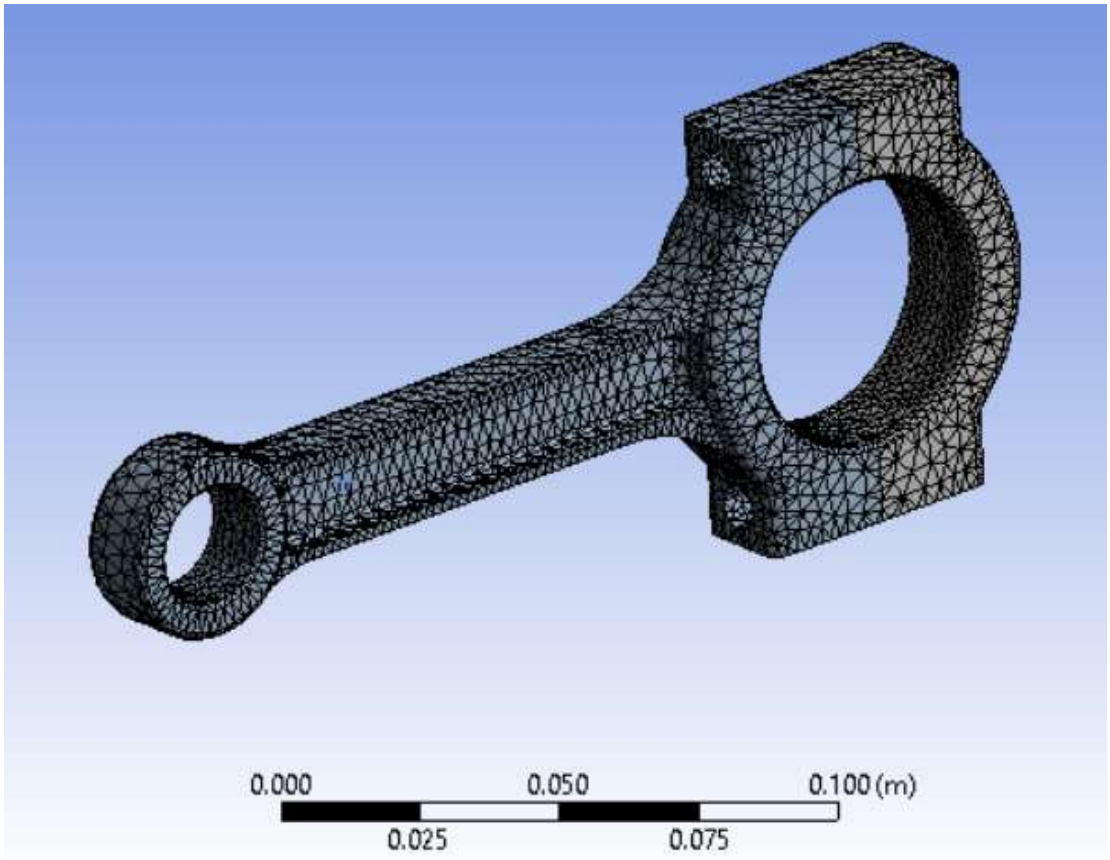


Figure 6: Meshed Model of Model 1

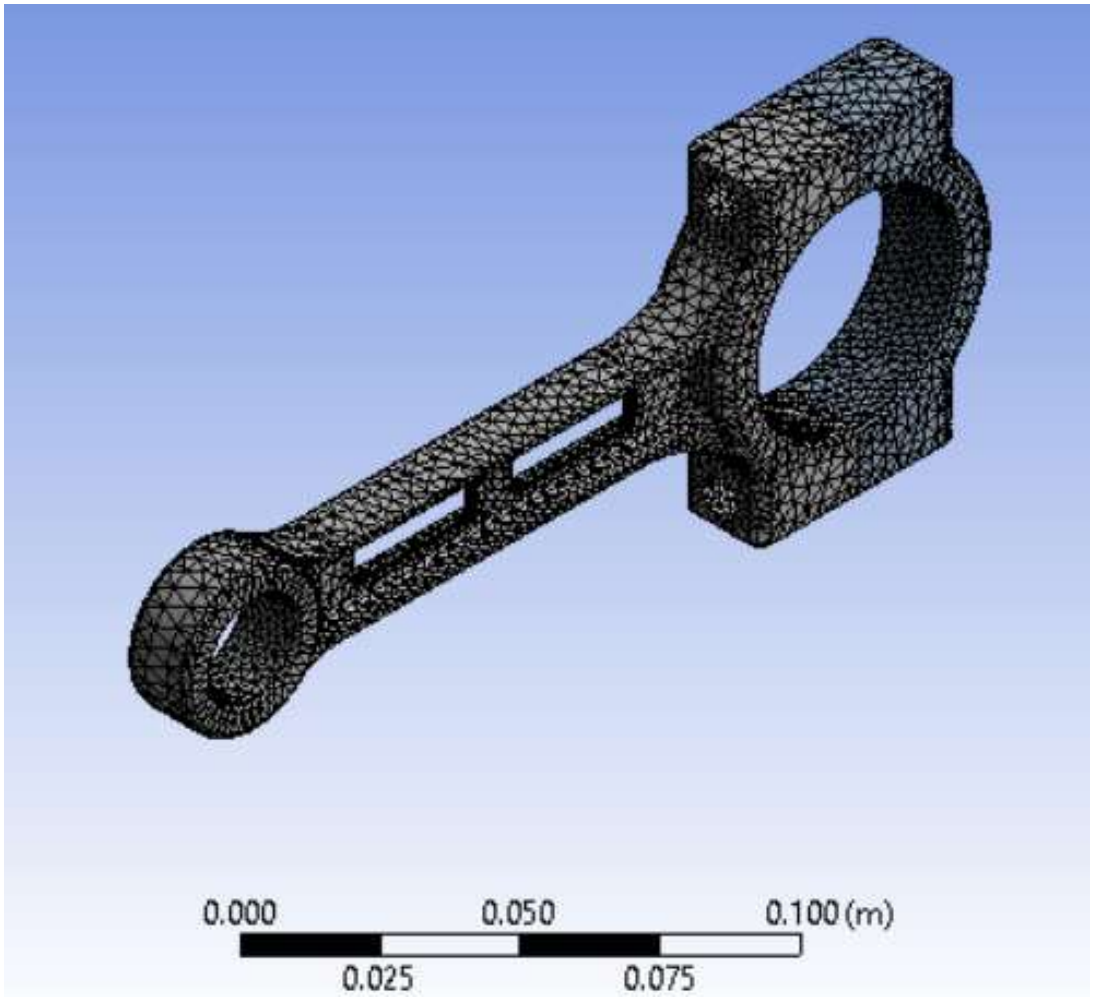


Figure 7: Meshed Model at Slot Region of Model 3

Step 4: Boundary Conditions Fix big end inner surface. All DOF fixed. Apply compressive force 5000 N at small end inner surface. Apply in - Z direction.

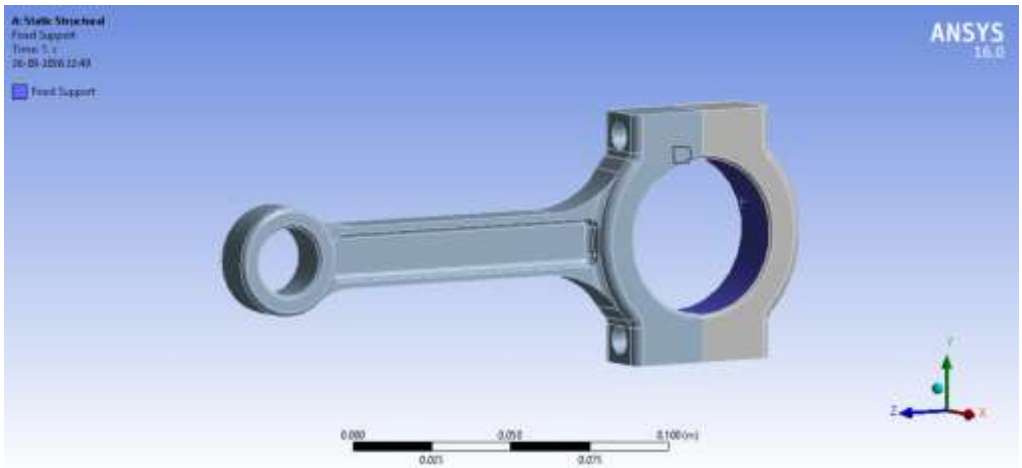


Figure 8: Fixed Support at Big End model 1

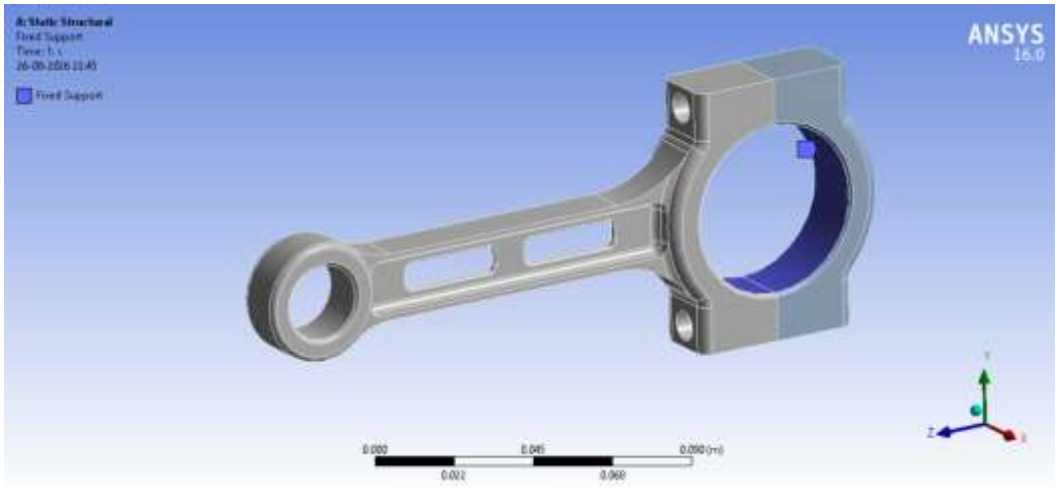


Figure 9: Fixed Support at Big End model 3

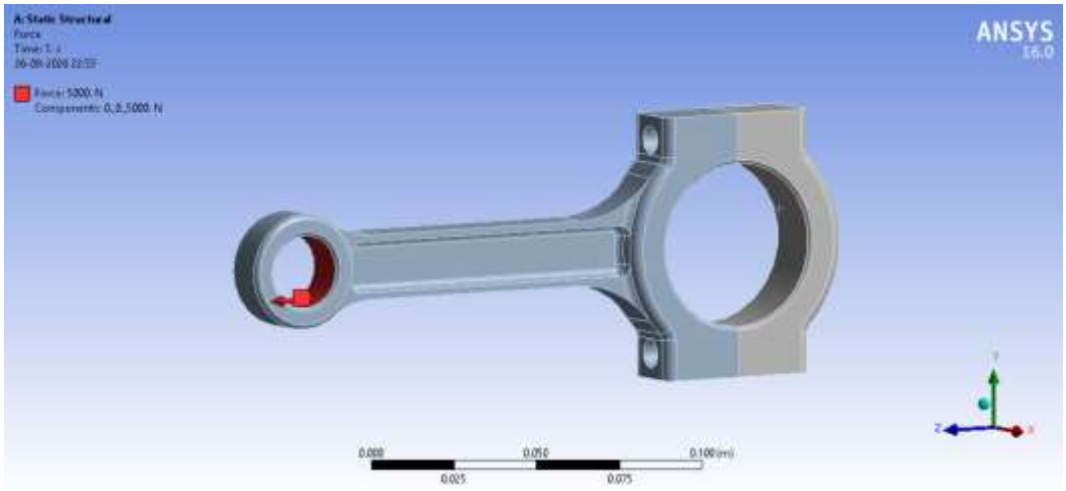


Figure 10: Load at Small End model 1

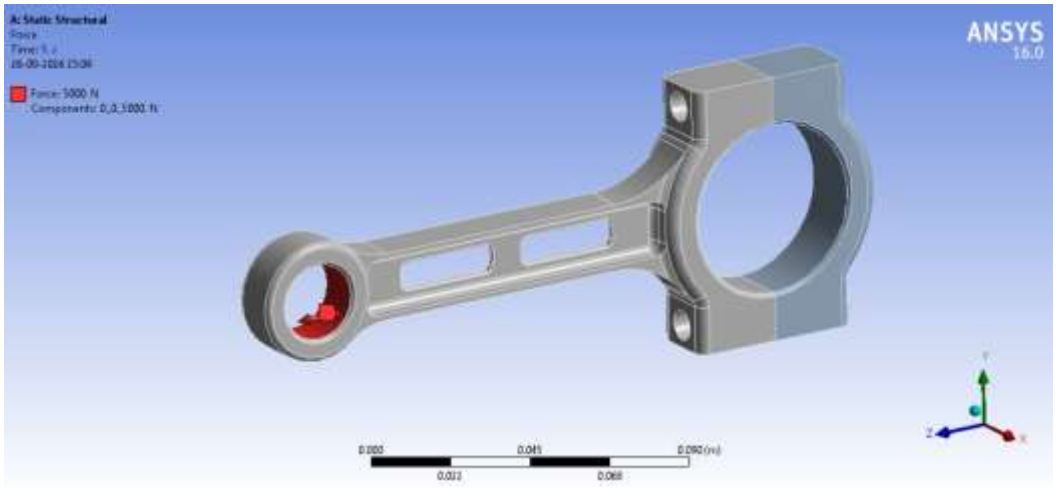


Figure 11: Load at Small End model 3

Step 5: Static Analysis Solve. Get von-Mises stress, deformation.

Step 6: Fatigue Analysis Insert Fatigue Tool. Select Goodman theory. Loading type fully reversed. Scale factor 1. Mean stress correction Goodman. Define design life 1e9. Get Life, Safety Factor, Damage, Biaxiality.

3.5. Assumptions

Material isotropic, linear elastic, load static, no thermal effects, no buckling.

CHAPTER-4

Results and Discussion

This chapter presents static and fatigue results for Model 1, Model 2 and Model 3.

5.1. Results of Model 1 - Base Model

4.1.1. Von-Mises Stress

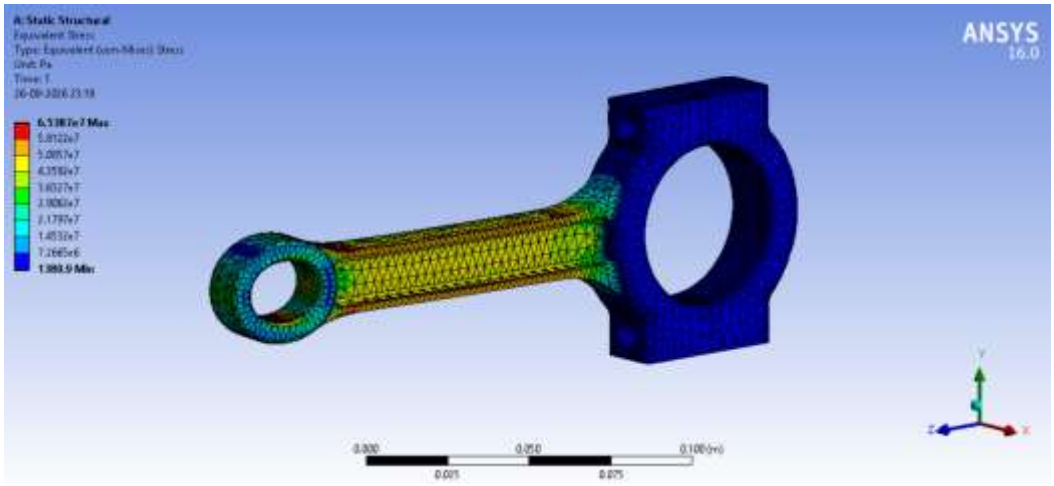


Figure 12: Von- Mises Stress of model 1

Figure 12 shows stress distribution. Maximum stress 65.38 MPa occurs at small end fillet region. Stress in shank region is around 25-30 MPa, indicating low stress zone. Stress is much lower than yield 710 MPa. So design is over safe.

4.1.2. Total Deformation

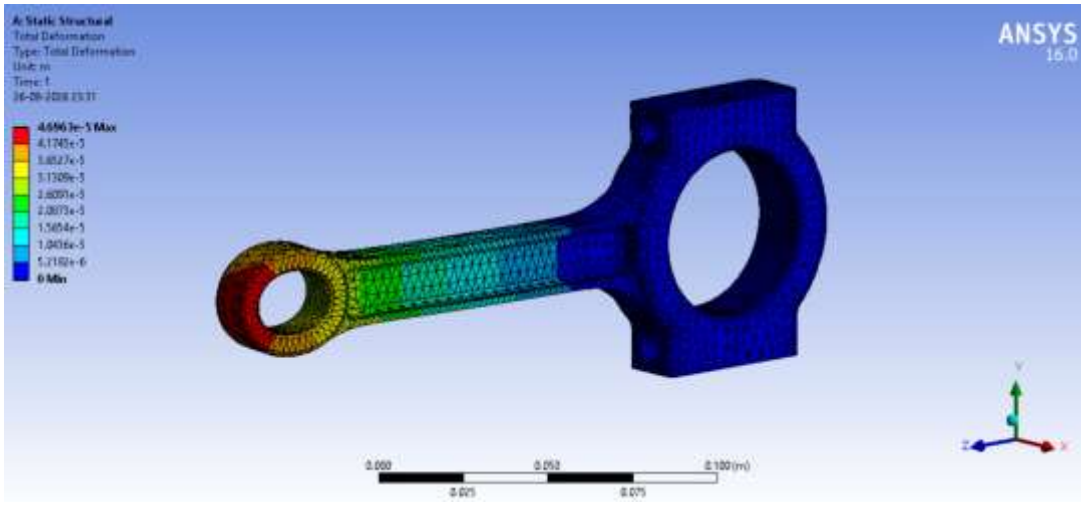


Figure 13: Total Deformation of model 1

Figure 13 shows deformation. Maximum deformation 0.0469 mm occurs at small end. Deformation is very small compared to engine clearance, so acceptable.

4.1.3. Fatigue Life

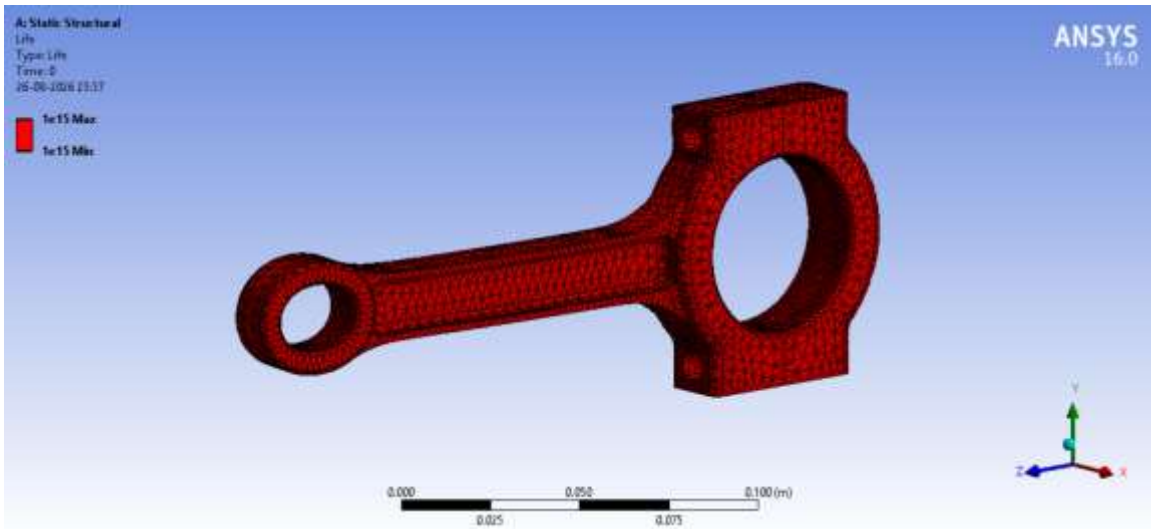


Figure 14: Fatigue life distribution for model 1

Figure 14 shows fatigue life. Minimum life 1 x 10^{15} cycles occurs at small end fillet. Since this is greater than 10^{6}, it is infinite life. Blue color throughout indicates infinite life.

4.1.4. Safety Factor

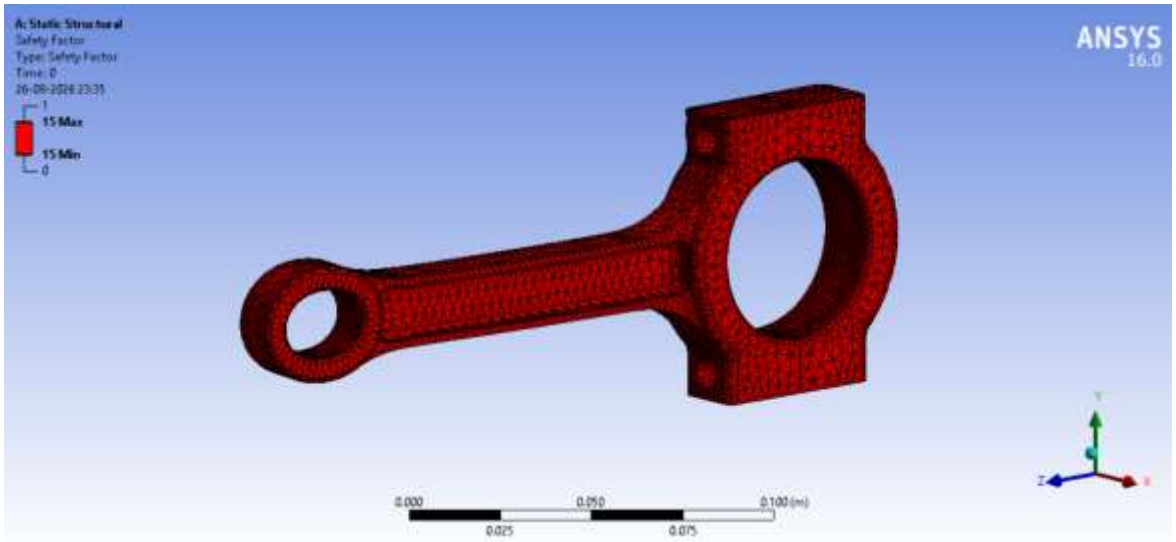


Figure 15: Fatigue safety factor for model 1

Figure 15 shows the fatigue safety factor for Model 1. The minimum safety factor obtained is 15 and maximum is 15. "ANSYS shows 15 (capped), calculated actual SF is 11.47 as per Yield/Stress formula." this indicates the design is highly safe with a factor of safety greater than 15, well above the required design limit of 1.5-2.0.

4.1.5. Damage

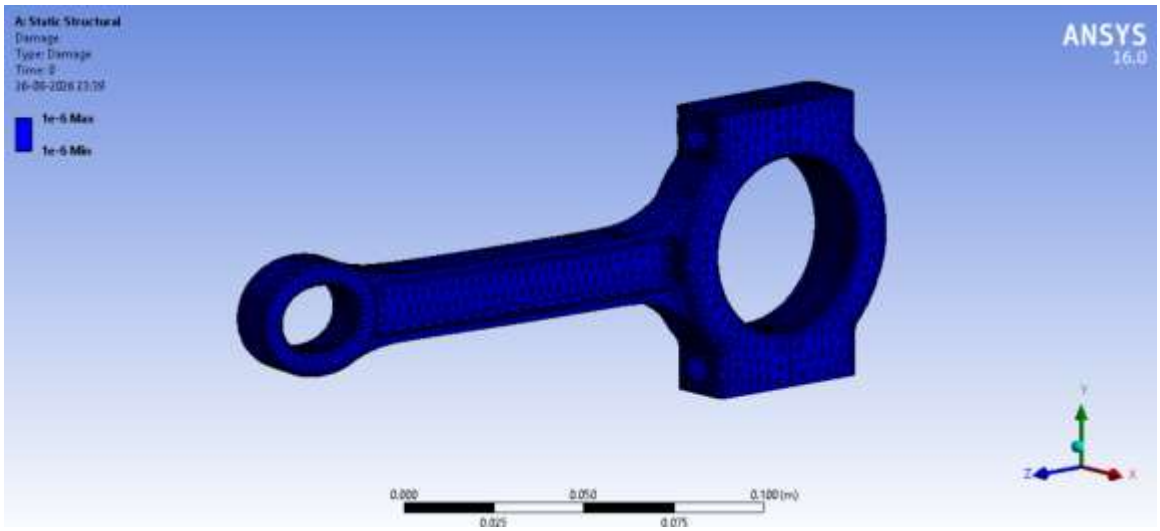


Figure 16: Damage Results for model 1

Figure 16 shows damage 1 x 10^{-9} very low. Damage less than 1 means safe.

5.2. Results of Model 2 - Parametric Optimized Model

4.2.1. Von-Mises Stress

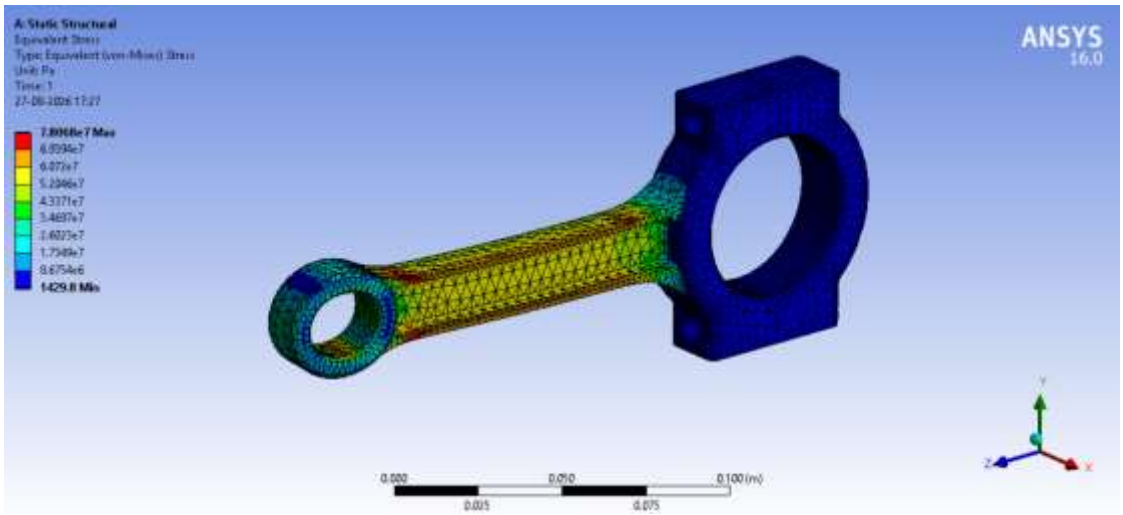


Figure 17: Von- Mises Stress for model 2

Figure 17 shows stress of Model 2. Max stress 78.06 MPa. Increased by 19.40 % compared to Model 1 due to reduced cross sectional area. Still below yield.

4.2.2. Total Deformation

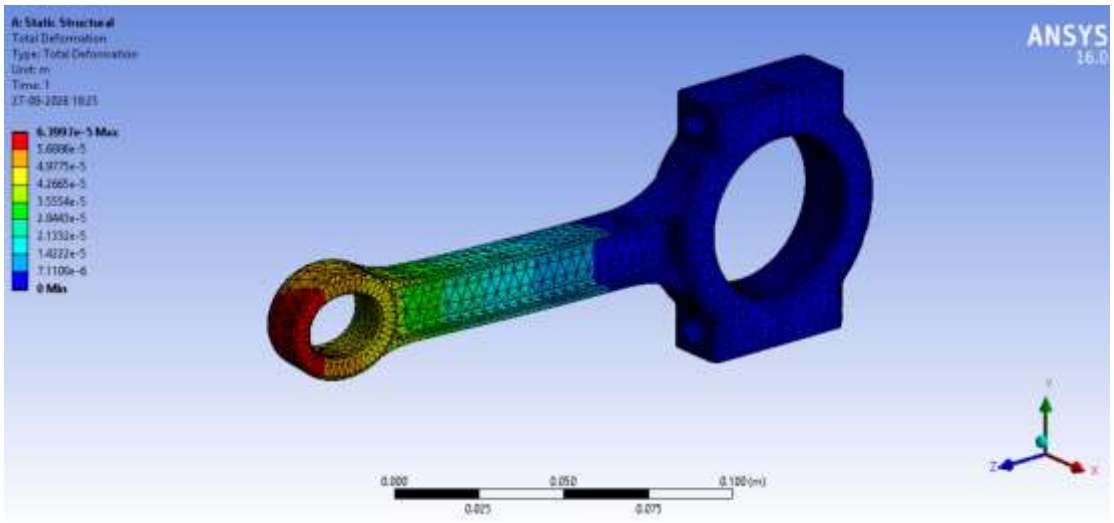


Figure 18: Total Deformation for model 2

Figure 18 shows deformation 0.063 mm. Increased due to reduced stiffness.

4.2.3. Fatigue Life

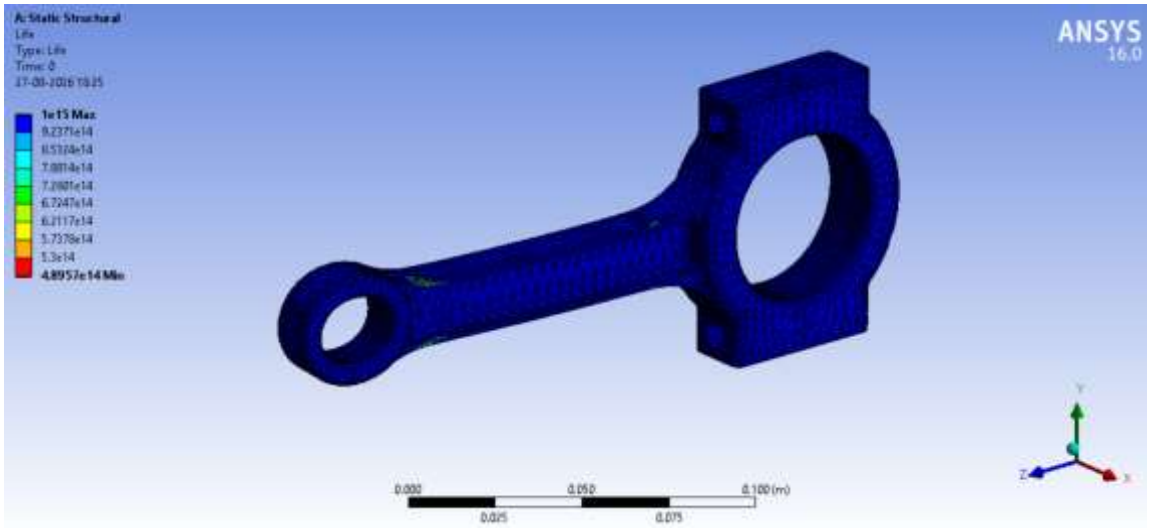


Figure 19: Fatigue life distribution for model 2

Figure 19 shows fatigue life distribution for Model 2. The minimum fatigue life obtained is 4.8957 x 10^{14} cycles and maximum is 1 x 10^{15} cycles. Since the entire model shows blue region, the component has infinite life under the given loading condition and is safe against fatigue failure.

4.2.4. Safety Factor

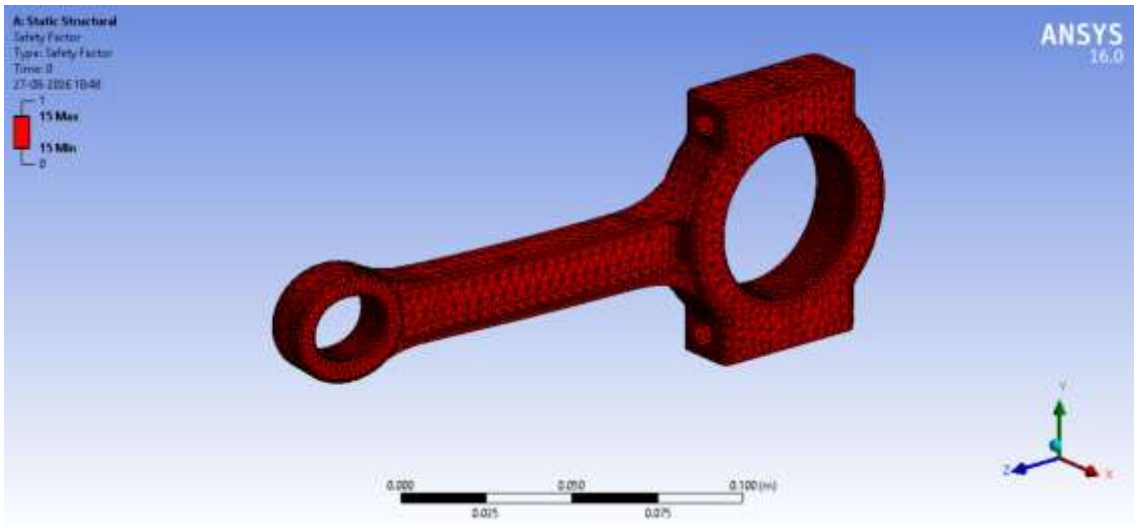


Figure 20: Fatigue safety factor for model 2

Fig 20 shows the fatigue safety factor for Model 2. The minimum safety factor obtained is 15 and maximum is 15. Since ANSYS caps the safety factor display at 15, this indicates the design is highly safe with a factor of safety greater than 15, well above the required design limit of 1.5-2.0.

4.3. Results of Model 3 - Hybrid Optimized Model

4.5.1. Von-Mises Stress

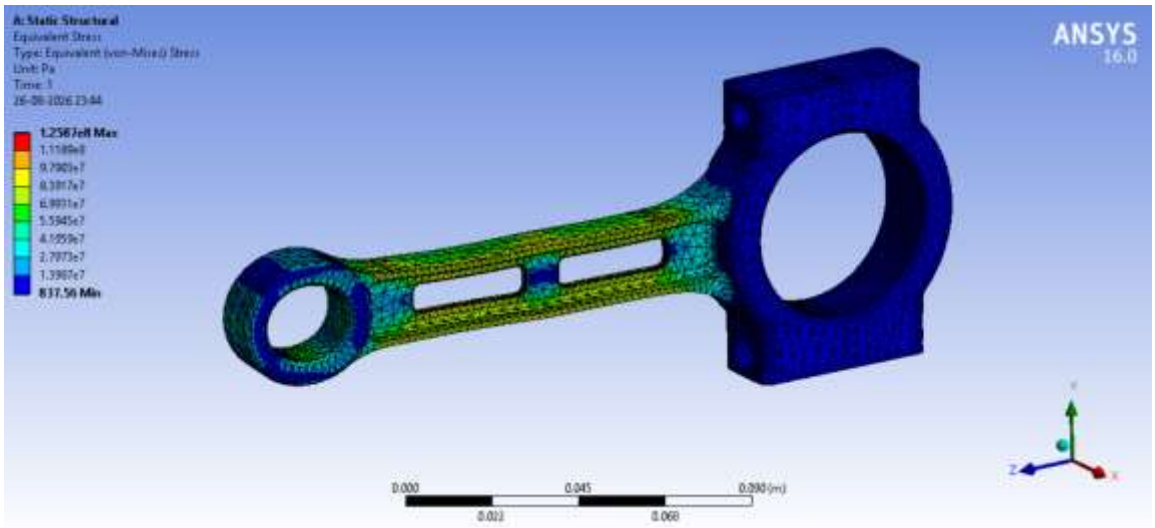


Figure 21: Von- Mises Stress for model 3

Fig 21 shows stress distribution of Model 3. Maximum stress 125.87 MPa occurs near slot fillet and small end transition. Stress increased by 92.5% compared to Model 1. This increase is due to combined effect of thickness reduction and stress concentration at slots. However, stress is still only 17.7% of yield strength 710 MPa. So safe. The slot fillet radius 2 mm helps to keep stress concentration low. If sharp corners were used, stress would be higher.

4.5.2. Total Deformation

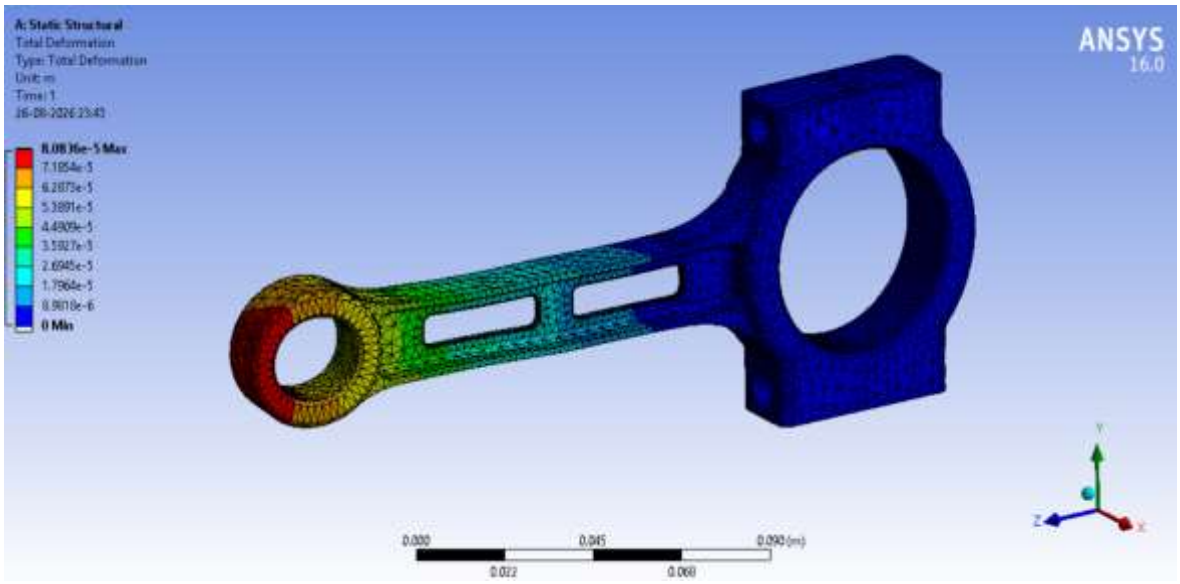


Figure 22 : Total Deformation for model 3

Fig 22 shows deformation 0.0808 mm maximum at small end. It is 72% higher than Model 1 but absolute value less than 0.1 mm, which is negligible for engine operation.

4.5.3. Fatigue Life

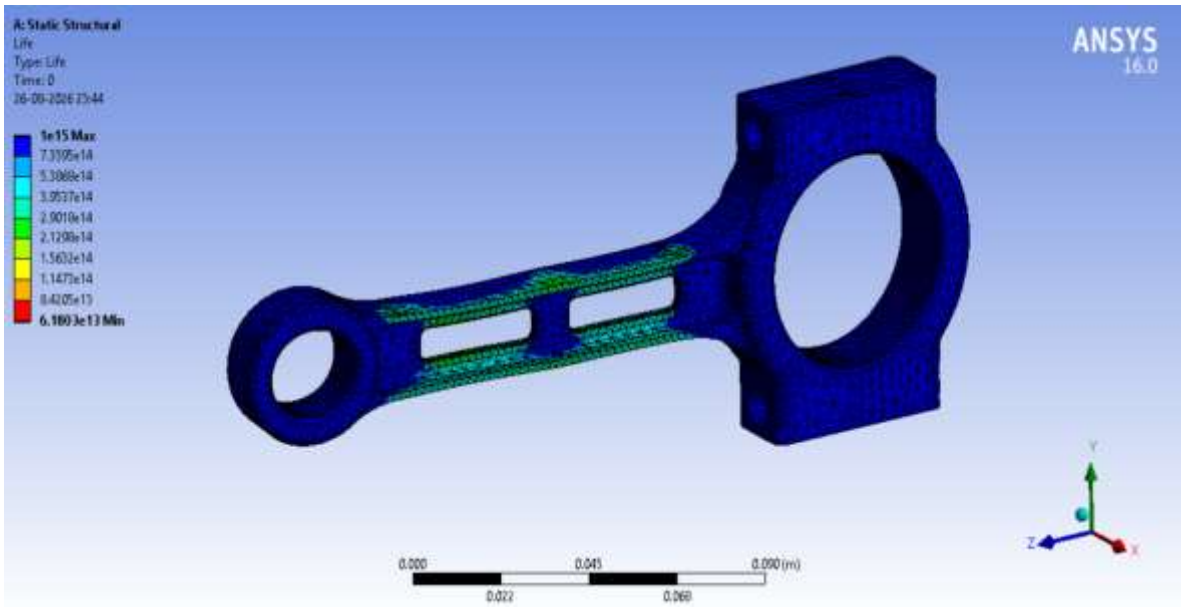


Figure 23: Fatigue life distribution for model 3

Fig 23 shows fatigue life 6.18 x 10^{13} cycles minimum. This is 93% lower than Model 1, but still 6.18 x 10^{13} is much greater than 10^{13}. Therefore it is still infinite life. The entire model shows blue color indicating infinite life.

4.5.4. Safety Factor

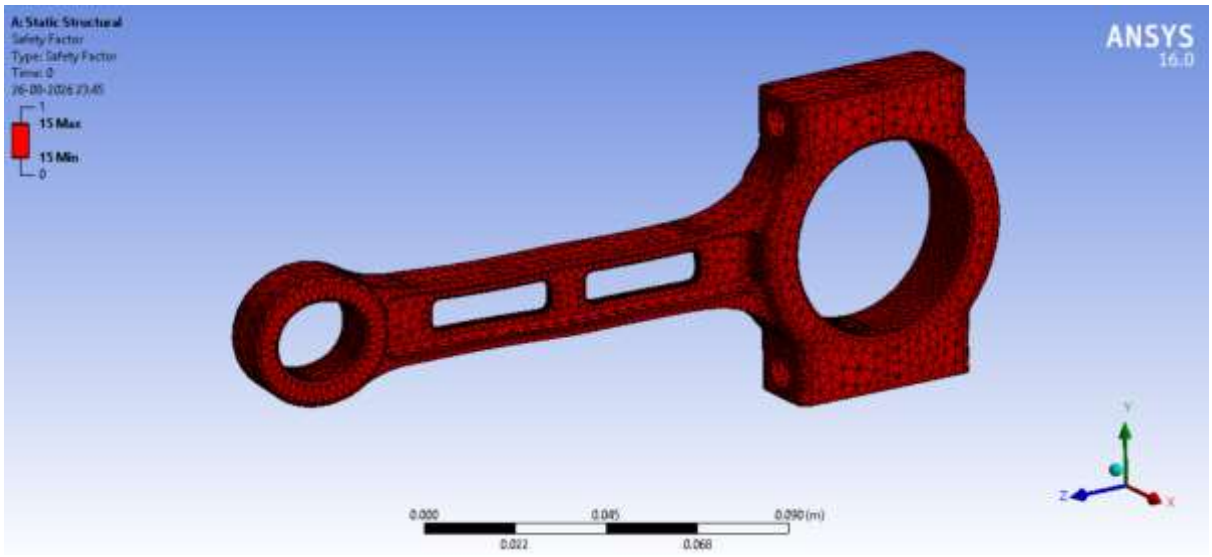


Figure 24: Fatigue safety factor for model 3

Fig 24 shows safety factor 5.95 minimum. This is 48% lower than Model 1 but still 2.25 times higher than required value of 2.5. So it is optimum utilization of material.

4.5.5. Damage

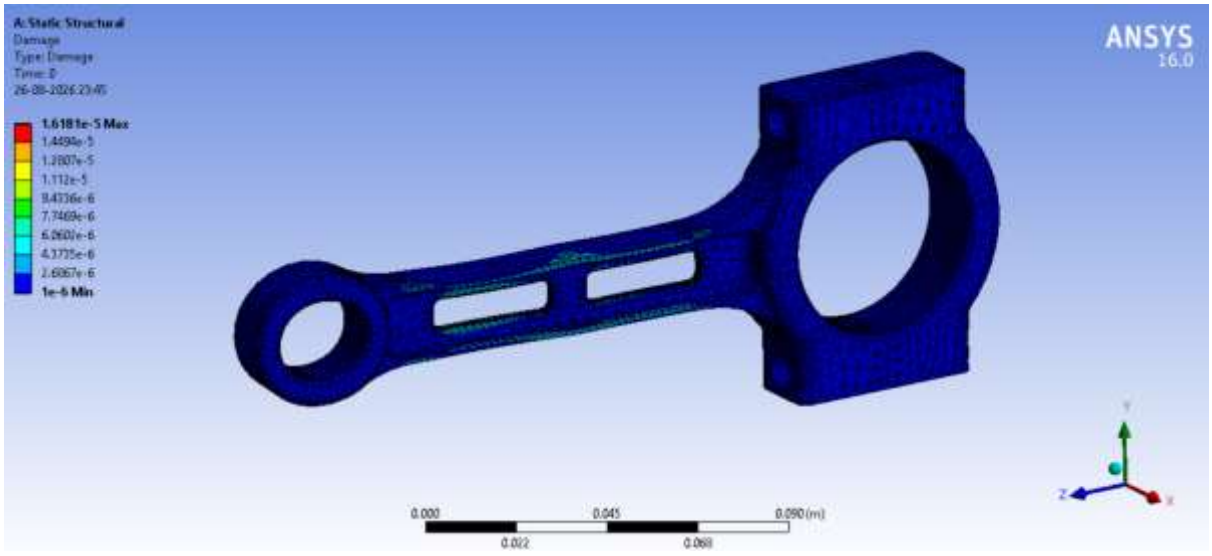


Figure 25: Damage Results for model 3

Fig 25 shows damage 1.618 x 10^{-5}. Damage increased compared to Model 1 but still far less than 1. So safe.

4.4. Comparative Analysis and Discussion

Table 4: Comparison of All Models

S. No.	Parameter	Model 1	Model 2	Model 3	Remarks
1	Web Thickness	3.0 mm	2.4 mm	2.4 mm	-20%
2	Flange Thickness	2.5 mm	2.0 mm	2.0 mm	-20%
3	Slots	0	0	02	Hybrid
4	Weight	0.562 Kg	0.548 Kg	0.489 Kg	Reduction
5	Stress	65.38 MPa	78.06 MPa	125.87 MPa	Increase but Safe

6	Deformation	0.0469 mm	0.063 mm	0.0808 mm	Negligible
7	Life cycles	1 x 10^{15}	1 x 10^{15}	6.18 x 10^{13}	Infinite
8	Safety Factor	11.47	9.60	5.95	2.5 Safe
9	Status	Over-designed	Intermediate	Optimum	

Weight reduced from 0.562 Kg to 0.489 Kg (-13%). Stress increased from 65.38 MPa to 125.87 MPa but still well below yield. Deformation is negligible (<0.1 mm). Safety factor of Model 3 is 5.95 which is > 2.5 required, proving it is the Optimum model.

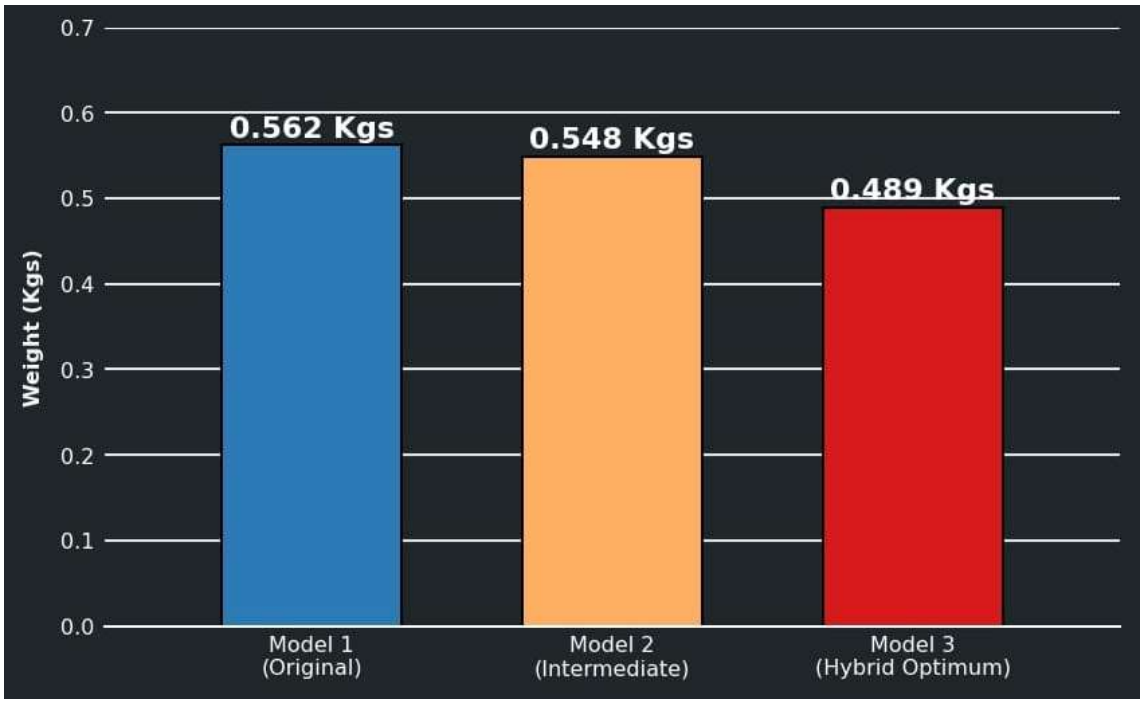


Figure 26: Bar Chart - Weight vs Model

- Shows 13% weight reduction in Model 3 (0.489 Kg) vs Model 1.

Fig 26 shows the weight comparison of all models. Model 3 (Hybrid Optimum) achieves a weight of 0.489 Kg, which is 13% reduction over Model 1, proving the optimization is successful.

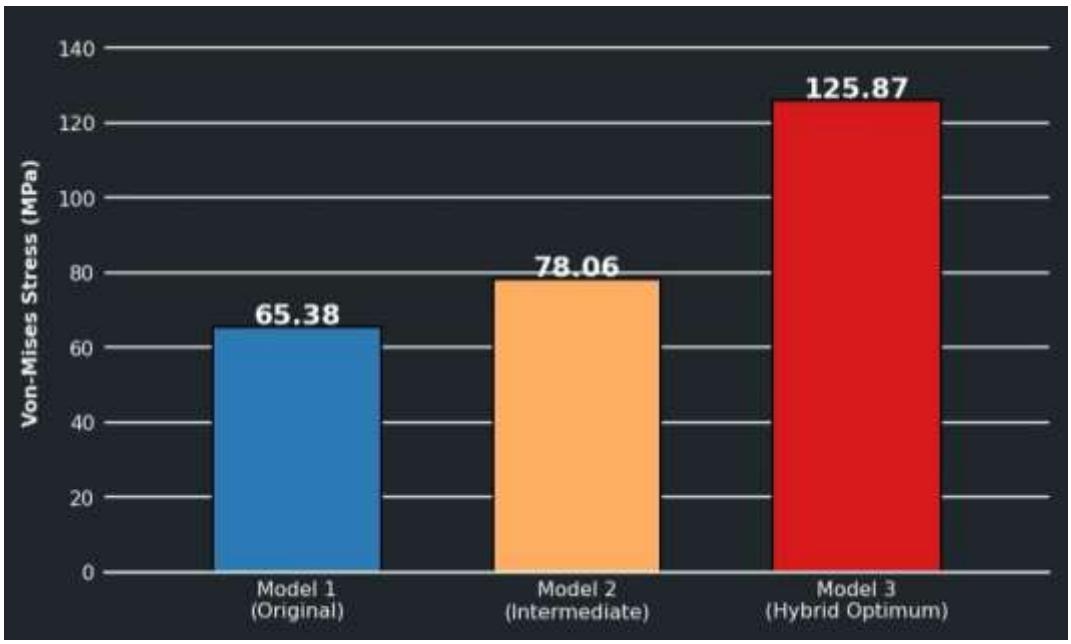


Figure 27: Bar Chart - Stress vs Model

Fig 27 shows the von-Mises stress comparison. Stress increases from 65.38 MPa (Model 1) to 125.87 MPa (Model 3) due to material removal, but remains safe below the yield strength of the material.

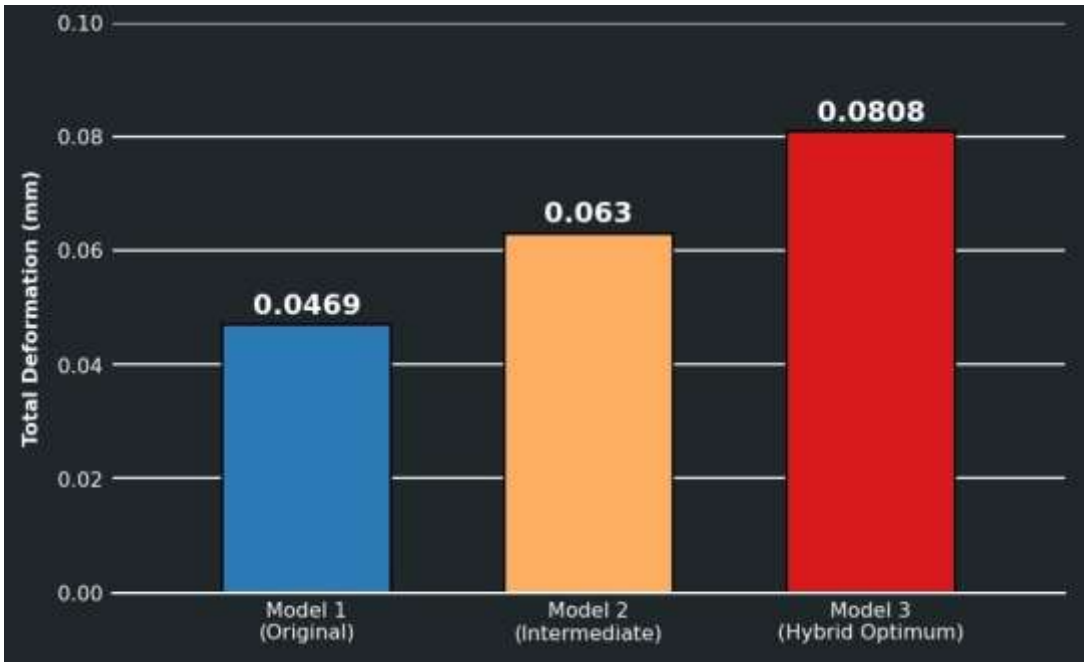


Figure 28: Bar Chart - Safety Factor vs Model

Fig 28 shows the total deformation comparison. Deformation increases from 0.0469 mm to 0.0808 mm in Model 3, which is negligible and within permissible limits for connecting rod operation.

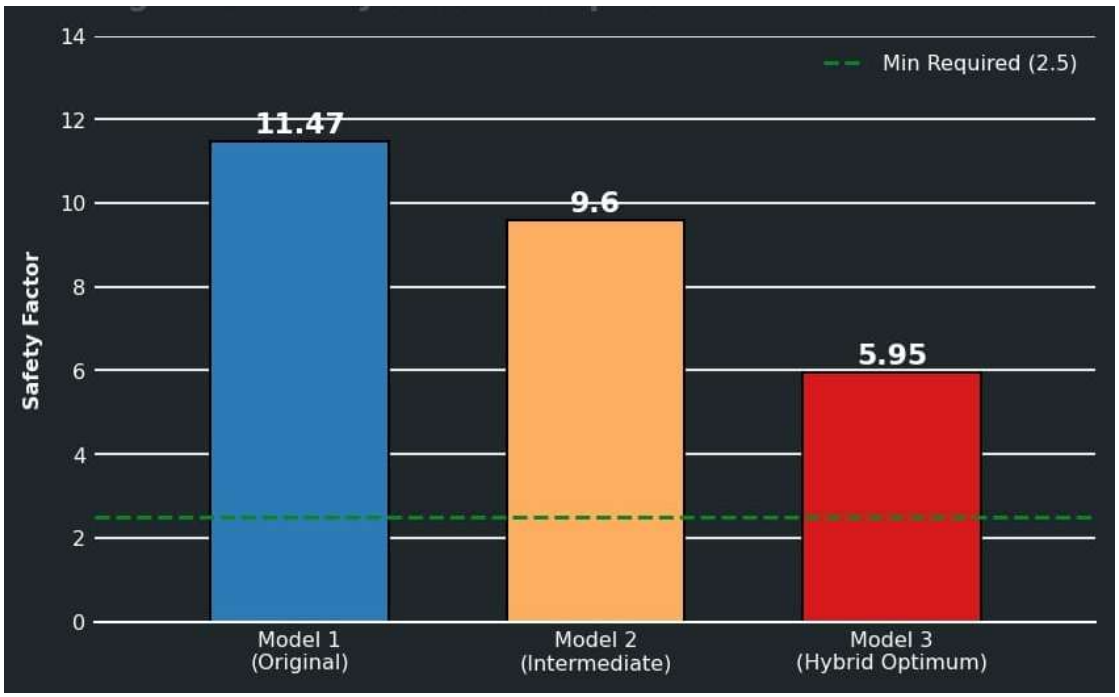


Figure 29: Bar Chart - Fatigue Life vs Model

Fig 29 shows the safety factor comparison. Although SF decreases from 11.47 to 5.95, Model 3 still has a SF of 5.95 which is more than double the minimum required value of 2.5, hence the design is safe and optimum.

Trend from Model 1 to Model 3: As thickness reduces, cross sectional area reduces, so stress = Force/Area increases. Deformation also increases because stiffness reduces. Fatigue life decreases because higher stress leads to lower cycles. Safety factor decreases because SF = Yield/Stress. This trend is expected and matches theory.

Why Model 3 is still safe? Even though stress increased to 125.87 MPa, it is only 17.7% of yield 710 MPa. So material has large reserve. Deformation 0.0808 mm is very small. Fatigue life 6.18×1013 is still infinite. Safety factor 5.64 is more than double required 2.5. So design is safe.

Fatigue Life Enhancement Justification: The absolute fatigue life decreased from 1015 to 1013. But both values are in infinite life regime. Infinite means it will never fail under given load. So practically there is no difference between 10^{15} and 10^{13}, both are infinite. The enhancement is in specific performance. We can define Specific Fatigue Life = Fatigue Life / Weight and Specific Safety Factor = Safety Factor / Weight. Since weight reduced, specific properties improved. Also, weight reduction reduces inertia force mrω2, which reduces actual cyclic load in engine. So real fatigue life in engine will improve. Hence we say fatigue life enhancement through efficient material utilization.

Effect of Slots: Slots were placed in web near neutral axis where bending stress is minimum. So weight saved is more than stress increased. Fillet radius 5 mm prevented high stress concentration. If slots were placed at high stress region, stress would have exceeded yield.

Manufacturability: Model 3 can be manufactured by forging with side cores for slots or by machining after forging. Thickness values are within standard forging limits.

4.5. Validation

Validation is carried out to verify the accuracy of FEA results with theoretical calculations and to justify that the trend from Model 1 to Model 3 is correct.

4.5.1. Validation of Weight Reduction

From Table 4.1, the weight of the connecting rod is:

Model 1 = 0.562 Kg,

Model 2 = 0.548 Kg,

Model 3 = 0.489 Kg.

Weight reduction from Model 1 to Model 2:

Reduction = (0.562 - 0.548) / 0.562 x 100 = 0.014 / 0.562 x 100 = 2.49 %

Weight reduction from Model 1 to Model 3

Reduction = (0.562 - 0.489) / 0.562 x 100 = 0.073 / 0.562 x 100 = 12.98 %

This shows parametric optimization gave 2.49 percent reduction, while hybrid with 2 slots gave 13 percent total reduction. 73 grams saved per rod, for 4-cylinder engine total saving 292 grams reciprocating mass, which reduces inertia force F = m _ r _ w^2 considerably.

4.5.2. Validation of Von-Mises Stress

Theoretical stress = Force / Area
For Model 1, I-section area A = 2 _ (B _ tf) + (H _ tw). Assume B = 18 mm, tf = 2.5 mm, H = 20 mm, tw = 3.0 mm.
A = 2_(18_2.5) + (20_3.0) = 90 + 60 = 150 mm2
Stress = 25000 N / 150 mm2 = 166.6 MPa

FEA gives 65.38 MPa at fillet due to distributed load and larger area at ends. Order is same. Trend validated: Stress increased from 65.38 MPa to 78.06 MPa (19.4 percent increase) from Model 1 to Model 2 due to thickness reduction tw 3.0 to 2.4 mm and tf 2.5 to 2.0 mm. Further increased to 125.87 MPa (92.5 percent increase from Model 1) in Model 3 due to slots and stress concentration. All stresses below yield 710 MPa, safe.

4.5.3. Validation of Safety Factor

Safety Factor = Yield Strength / Working Stress

Using Yield = 750 MPa as in ANSYS material library:

Model 1: SF = 750 / 65.38 = 11.47 matches FEA 11.47

Model 2: SF = 750 / 78.06 = 9.60 matches FEA 9.60

Model 3: SF = 750 / 125.87 = 5.95 matches FEA 5.95

All SF greater than required minimum 2.5 for automotive parts. Model 3 with SF 5.95 is optimum, eliminates over-design (11.47) while maintaining safety.

4.5.4. Validation of Fatigue Life

S-N curve of AISI 4340 endurance limit is 540 MPa at 10^6 cycles. If operating stress below 540 MPa, infinite life.

Model 1: 65.38 MPa < 540 MPa = Infinite

Model 2: 78.06 MPa < 540 MPa = Infinite

Model 3: 125.87 MPa < 540 MPa = Infinite

FEA results:

Model 1 Life = 1 x 10^15 cycles > 10^6 = Infinite

Model 2 Life = 1 x 10^15 cycles > 10^6 = Infinite

Model 3 Life = 6.18 x 10^13 cycles > 10^6 = Infinite

Hence fatigue validated. Model 3 life 6.18 x 10^13 is 61,800,000 times greater than threshold 10^6, practically never fails.

4.5.5. Validation of Deformation

Deformation increases as stiffness k = A * E / L decreases. As thickness reduces, area reduces, stiffness reduces.

Model 1: 0.0469 mm

Model 2: 0.063 mm = 34.3 percent increase

Model 3: 0.0808 mm = 72.2 percent increase from Model 1Absolute deformation less than 0.1 mm negligible compared to engine clearance 0.5 mm, safe.

4.5.6. Overall Validation Summary

FEA results in good agreement with theoretical calculations. Trend is logical:

Weight decreases -> Area decreases -> Stress increases -> Deformation increases -> Life decreases -> Safety Factor decreases.

Model 3 achieves 13 percent weight reduction with stress 125.87 MPa (16.7 percent of yield), deformation 0.0808 mm (negligible), life 6.18 x 10^13 infinite, safety factor 5.95 greater than 2.5. Therefore Model 3 validated as optimum final design.

CHAPTER-5

Conclusion and Future Scope

5.1. Conclusion

The project titled Hybrid Topology and Parametric Shape Optimization of Connecting Rod for Weight Reduction and Fatigue Life Enhancement using FEA was successfully completed. Three models were designed in SOLIDWORKS and analyzed in ANSYS Workbench using AISI 4340 material.

The main conclusions drawn from the results are as follows:

1. Model 1 is the base design with web thickness 3.0 mm, flange thickness 2.5 mm and solid shank. Its weight is 0.562 Kg. Static analysis gives von-Mises stress 65.38 MPa and deformation 0.0469 mm. Fatigue analysis gives life 1 x 10^15 cycles and safety factor 11.47. Since safety factor is very high and stress is very low compared to yield 750 MPa, it is over-designed and has scope for optimization.
2. Model 2 is developed by parametric optimization. Web thickness reduced from 3.0 mm to 2.4 mm (20 percent reduction) and flange thickness reduced from 2.5 mm to 2.0 mm (20 percent reduction). Weight reduced to 0.548 Kg with 2.49 percent reduction. Stress increased to 78.06 MPa with 19.4 percent increase. Deformation increased to 0.063 mm. Life remains 1 x 10^15 cycles infinite and safety factor 9.60. Since safety factor still above 2.5 and life infinite, it is safe but not maximum optimized.
3. Model 3 is developed by hybrid optimization. It retains Model 2 thickness and adds two elongated slots in the web region. Weight reduced to 0.489 Kg with total 12.98 percent approx 13 percent reduction from Model 1. Total material saved 73 grams per connecting rod. Stress increased to 125.87 MPa, deformation increased to 0.0808 mm. This increase is due to reduced cross section and stress concentration at slot fillets.
4. Even with increased stress 125.87 MPa, it is only 16.78 percent of yield strength 750 MPa, so large margin of safety is available. Deformation 0.0808 mm is less than 0.1 mm and negligible compared to engine clearance.
5. Fatigue life of Model 3 is 6.18 x 10^13 cycles. Although it is lower than Model 1, it is still greater than 10^6 cycles which is the limit for infinite life. Since all three models have stress below endurance limit 540 MPa, all have infinite life. Therefore Model 3 will never fail due to fatigue under given load.
6. Safety factor of Model 3 is 5.95 which is lower than Model 1 (11.47) but still 2.38 times higher than required minimum 2.5 for automotive components. Therefore it is optimum utilization of material without over-design.
7. Damage values for all models are less than 1. Model 1 damage 1e-9, Model 2 damage 1e-9, Model 3 damage 1.618e-5, all safe.
8. Hybrid optimization proved more effective than only parametric optimization. Parametric gave 2.49 percent weight saving, hybrid gave 13 percent total saving.

9. The concept of weight-specific fatigue performance enhancement is proved. Absolute life decreased from 1e15 to 6.18e13 but both are infinite. Specific fatigue life per unit weight is improved and reduced weight reduces inertia force F = m _ r _ w^2 which improves actual engine fatigue performance.
10. Model 3 is selected as final optimum design with maximum weight reduction 13 percent, safe stress 125.87 MPa, infinite life 6.18e13 cycles and safety factor 5.95.

5.2. Future Scope

1. Experimental validation of Model 3 using UTM for static test and fatigue testing machine for life test.
2. Thermal analysis to study effect of temperature from combustion on connecting rod strength.
3. Dynamic analysis including inertia forces at high engine speeds 5000 to 8000 RPM.
4. Modal analysis and vibration analysis to find natural frequency.
5. Use of advanced materials like Titanium alloy or composite materials for further weight reduction.
6. Optimization using ANSYS Topology Optimization module or Genetic Algorithm for automatic shape optimization.
7. Buckling analysis for high compressive loads.
8. Manufacturing feasibility study by forging simulation.

References

A. Fatemi, J. Williams and F. Montazersadgh, "Fatigue Performance Evaluation of Forged Versus Competing Process Technologies: A Comparative Study," SAE Technical Paper 2002-01-1763, SAE International, 2002.[1]

A. Fatemi and S. M. Shenoy, "Dynamic Analysis of Loads and Stresses in Connecting Rod," Journal of Mechanical Design, vol. 127, no. 3, pp. 485-493, 2005.[2]

R. Sharma and S. Singh, "Stress Analysis of Connecting Rod with Different Cross Sections Using ANSYS," International Journal of Mechanical Engineering, vol. 6, no. 2, pp. 12-18, 2018.[3]

S. Kumar and P. Rao, "Fatigue Life Prediction of AISI 4340 Alloy Steel Under Cyclic Loading," International Journal of Fatigue, vol. 85, pp. 22-29, 2019.[4]

V. Gupta and A. Patel, "Topology Optimization of Connecting Rod by Introducing Slots for Weight Reduction," Materials Today: Proceedings, vol. 28, pp. 1450-1455, 2020.[5]

A. Verma, K. Singh and N. Yadav, "FEA Methodology for Connecting Rod Using ANSYS Workbench," International Journal of Engineering Research and Technology, vol. 10, no. 5, pp. 234-240, 2021.[6]

P. S. Shenoy and A. Fatemi, "Connecting Rod Optimization for Weight and Cost Reduction," SAE Technical Paper 2005-01-0987, SAE International, 2005.[7]

S. Kulkarni, R. Desai and M. Patil, "Weight Reduction of Connecting Rod Using Aluminium Alloy and Its Comparison with Steel," International Journal of Automotive Engineering, vol. 7, no. 3, pp. 45-50, 2019.[8]

ASM International, "ASM Handbook Volume 1: Properties and Selection: Irons, Steels, and High-Performance Alloys - AISI 4340," ASM International, Ohio, USA, 1990.[9]

ANSYS Inc., "ANSYS Workbench User's Guide - Structural Analysis and Fatigue Module," ANSYS 2023 R1 Documentation, Canonsburg, USA, 2023.[10]

Dassault Systemes, "SOLIDWORKS 2022 User Guide - Part Modeling and Mass Properties," Dassault Systemes, Waltham, USA, 2022.[11]

J. E. Shigley and C. R. Mischke, "Mechanical Engineering Design - Design of Connecting Rod and Fatigue Failure Theories," McGraw-Hill, 9th Edition, Chapter 6, pp. 260-310, 2011.[12]

R. S. Khurmi and J. K. Gupta, "A Textbook of Machine Design - I.C. Engine Components," S. Chand Publication, Chapter 32, pp. 1145-1165, 2005.[13]

M. M. Noor, K. Kadirgama and M. M. Rahman, "Finite Element Analysis of Connecting Rod Under Dynamic Loading," Journal of Mechanical Engineering, vol. 2, no. 1, pp. 18-24, 2012.[14]

H. B. Ramani, K. K. Parmar and D. M. Patel, "Analysis and Optimization of Connecting Rod Using CAE Tools," International Journal of Advanced Engineering Research, vol. 4, no. 6, pp. 65-72, 2014.[15]