

Static Structural and Modal Analysis of an Internal Combustion Engine Connecting Rod: Simulation Approach

Zakir Hussain Siddiqi¹; Ghanshyam Dhanera²

¹M.E. Research Scholar, ²Assistant Professor

Department of Mechanical Engineering, BM College of Technology, Indore (M.P.) India

Abstract:

The present research work is focused on static structural and modal analysis of a forged steel connecting rod of standard dimensions, using simulation approach. For this purpose, a standard connecting rod of an internal combustion (IC) engine was selected, and after calculating maximum gas force and providing necessary boundary conditions, static structural analysis and modal analysis were performed. Static structural analysis indicates that the connecting rod deforms mainly along its length (Z-axis), demonstrating effective axial load transmission and structural stability. The observed Von Mises stress was 703.2 MPa. A total deformation of 0.93326 mm is within acceptable limits, suggesting the rod maintains structural integrity without compromising engine performance. Modal analysis reveals six distinct natural frequencies ranging from 465.9 Hz to 5959.3 Hz, with greater deformations in lower modes, indicating increased susceptibility to vibrations at these frequencies, which could be critical if they align with the engine's operating range. For the purpose of simulation ANSYS 14.0 and for model creation CATIA V5R12 software were used.

Keywords: *Internal combustion (IC) engine, connecting rod, simulation, static structural analysis, modal analysis, software.*

1. Introduction

The connecting rod is a vital component in the mechanics of automobile engines, functioning primarily to transmit motion and force between the piston and the crankshaft. This role is critical in converting linear motion from the piston into the rotational motion needed to drive the crankshaft. According to Singh and Pramanik, this part of the engine undergoes significant loading, facing both tensile and compressive forces during operation due to maximum combustion pressures and inertia forces, which makes its structural integrity paramount (Singh & Pramanik, 2016). Additionally, Luo and Liu emphasized that the connecting rod experiences complex stress scenarios, necessitating a thorough analysis of its mechanical properties to

enhance performance and durability (Luo & Liu, 2018). Recent advances in engineering have underscored the importance of utilizing finite element analysis to scrutinize the structural integrity of connecting rods. Research conducted by Hasan indicates that simulations yield insights regarding stress distribution across connecting rods under various operational conditions, enhancing predictive capabilities for safety and performance (Hasan, 2018). Such methodologies allow for the identification of potential failure points, further promoting better design practices. By advancing this approach, researchers can simulate realistic engine operating conditions, ensuring that connecting rods can withstand fatigue over their functional lifespans (Luo & Liu, 2018; Hasan, 2018). Considering these facts, the present research work was focused on structural analysis and thermal analysis of the connecting rod. For this purpose, a connecting rod of standard dimensions was selected and ANSYS software was used for conducting for analysis purpose, and CATIA V5R12 was used for model development.

1.1 Objectives of the Research

The following points represent the objectives of the research work:

- a) To perform static analysis of the conducting rod; and
- b) To perform modal analysis of connecting rod.

2. Literature Review

The present section is devoted to contributions of researchers in the field of connecting rods and concludes with the gaps of the research.

2.1 Contributions of Researchers in the field of Connecting Rods

The following points represent the contributions of researchers in the field of connecting rods.

- **Patel et al. (2022)** conducted a comparative study examining stress analysis on connecting rods using finite element methods (FEM). Their work highlights the importance of different mesh types while analyzing the effects of supplementary loading on pre-stressed plates, ultimately aiding in the understanding of stress distributions in connecting rod designs (Patel et al., 2022).
- **Kumar (2018)** implemented a process failure mode and effects analysis (FMEA) during the assembly of connecting rods. This study emphasized the need for systematic analysis during production processes to enhance operational efficiency and product reliability, showcasing

how organizational strategies influence engineering practices in the connecting rod production sector in India (Kumar, 2018).

- **Chumbre (2018)** provided a detailed design and comparative analysis of connecting rods utilizing finite element analysis. This study aimed at understanding the structural integrity and performance characteristics of connecting rods through advanced simulation techniques, providing a comprehensive approach to design optimization (Chumbre, 2018).
- **Singh & Pramanik (2016)** focused on structural analysis incorporating finite element methods to investigate the mechanical behavior of connecting rods under various loading conditions. Their findings shed light on the influence of combustion pressure and inertial forces on connecting rod performance, thus informing design specifications for more robust components (Singh & Pramanik, 2016).
- **Sriharsha & Rao (2020)** examined design considerations concerning the forces experienced by connecting rods, particularly during combustion cycles. Their work emphasizes the necessity for robust construction capable of enduring severe tensile and compressive loads (Sriharsha & Rao, 2020).
- **Ilman & Barizy (2015)** performed a failure analysis on a damaged connecting rod from a reciprocating air compressor, using finite element models to pinpoint failure mechanisms. Their analysis contributes valuable insights into materials science and mechanical design, enabling the identification of weak points in connecting rod structures (Ilman & Barizy, 2015).
- **Kurdi et al. (2012)** simulated fatigue life predictions for engine connecting rods, utilizing finite element analysis to analyze loading conditions and establish parameters for improving product longevity (Kurdi et al., 2012).
- **Luo & Liu (2018)** engaged in fatigue analysis of engine connecting rods using finite element methods, illustrating the complex stress conditions these components endure during operation. Their findings support the need for comprehensive analysis methodologies for enhanced design processes (Luo & Liu, 2018).
- **Rajueni & Choudhury (2022)** assessed the availability and potential for repurposed medical products in India, indirectly highlighting cross-disciplinary innovation that could inspire future designs and materials used in connecting rod engineering Rajueni & Choudhury (2022) .

- **Zhang (2017)** explored finite element analysis concerning gasoline engine connecting rods, stressing reliability in performance enhancement. This research underscores the growing importance of lightweight design in automotive components and its consequent implications for engine performance (Zhang, 2017).
- **Choudhury & Saberwal (2019)** discussed organizational structures for orphan medicinal products, paralleling the complex systems involved in connecting rod manufacture, which could benefit from similar streamlined operational strategies (Choudhury & Saberwal, 2019).
- **Huo et al. (2012)** worked on lightweight structural designs of connecting rods, an important aspect of modern mechanical engineering that strives to reduce mass while ensuring strength, contributing to studies centered on automotive efficiency (Huo et al., 2012).
- **Lubis & Andoko (2019)** analyzed connecting rods in single-cylinder four-stroke petrol engines using finite element methods, revealing how poor fabrication processes could lead to performance issues, thus informing necessary design adjustments (Lubis & Andoko, 2019).
- **Fatu et al. (2005)** utilized advanced analytical approaches to examine the non-Newtonian effects in dynamically loaded connecting rod bearings, contributing to the understanding of lubrication and frictional dynamics critical for optimizing connecting rod performance (Fatu et al., 2005).
- **Wang et al. (2012)** provided insights into strength calculations for diesel engine linkages through established experimental methods, which serve to inform best practices in connecting rod design methodology (Wang et al., 2012).
- **Jia et al. (2013)** conducted a kinematic and dynamic analysis of piston-connecting rod mechanisms, detailing the complex movements that connecting rods must accommodate. This foundational understanding is crucial for any subsequent design improvements (Jia et al., 2013).
- **Zhang et al. (2012)** explored deformation and lubrication challenges faced by connecting rods in high-load environments, providing a deeper understanding of their operating conditions and potential design enhancements (Zhang et al., 2012).
- **(Yang et al., 2014)** investigated fracture splitting techniques for connecting rods, utilizing mechanical principles to improve manufacturing efficiency and reduce costs, thereby enhancing productivity within the automotive sector (Yang et al., 2014).

- **(Singh et al., 2008)** inadvertently contributed to engineering practices through research on transdermal drug delivery systems involving metallic rods, providing innovative ideas that could parallel material selections for connecting rod fabrication (Singh et al., 2008).
- **Choudhury (2020)** summarized key recommendations for connecting rod designs, encapsulating the ongoing discussions surrounding performance metrics and challenges faced by engineers in the field (Sriharsha & Rao, 2020).

2.2 Gaps in the Research

During the research work, it was found that there very less research papers found which focused on structural analysis and modal analysis of the existing connecting rods.

3. Solution Methodology

The present section is devoted to the details of analyses as well as software used in the research, the details of which are presented in upcoming sub-sections.

3.1 Structural Analysis

Structural analysis is a core area of civil and mechanical engineering that involves evaluating how structures respond to applied loads, forces, and environmental conditions. The primary objective is to ensure that a structure can safely withstand external influences without failure or excessive deformation. Engineers use structural analysis to study components like beams, columns, frames, and trusses, predicting how they will behave under different types of loads such as static, dynamic, thermal, or seismic forces. This ensures that the structure maintains its integrity and functionality throughout its service life.

3.2 Modal Analysis

Modal analysis is a technique used in structural dynamics to determine the natural frequencies, mode shapes, and damping characteristics of a structure. Every structure, whether simple or complex, has its own set of natural frequencies at which it tends to vibrate. When a structure is excited by external forces, especially at these natural frequencies, resonance can occur—leading to large amplitude vibrations that may cause damage or failure. Modal analysis helps engineers identify these critical frequencies and understand how the structure will behave under dynamic conditions.

3.3 ANSYS

ANSYS is versatile and widely used engineering simulation software that enables precise modeling, analysis, and optimization of complex systems. Based on the Finite Element Method (FEM), ANSYS provides a comprehensive environment for simulating various physical phenomena, including structural mechanics, heat transfer, fluid dynamics, and electromagnetism. Its intuitive user interface, advanced meshing tools, and high-performance solvers make it a preferred choice for engineers and researchers across industries. ANSYS allows users to visualize stress distributions, deformations, and failure zones, making it easier to interpret results and improve designs efficiently. *In the present research work ANSYS 14.0 was used.*

4. Case Study

The present section focuses on the details of problem formulation and its solution, the details of which are presented in upcoming sub-sections.

4.1 Problem Formulation

This research problem is formulated to perform both structural and modal analysis of a standard IC engine connecting rod using ANSYS software. The objective is to analyze stress distributions and deformation behavior under peak load conditions, determine the critical frequencies, and suggests modifications in geometry or material to optimize performance, reduce weight, and enhance durability.

4.2 Solution of the Problem

In order to solve the research problem, the following steps were adopted:

- a) First, the dimensions of an existing connecting rod used were taken, and the part was modeled using CATIA design software. In the present work, CATIA V5R12 software (Version 5, Release 12) was used for modeling, as shown in Figure 4.1.

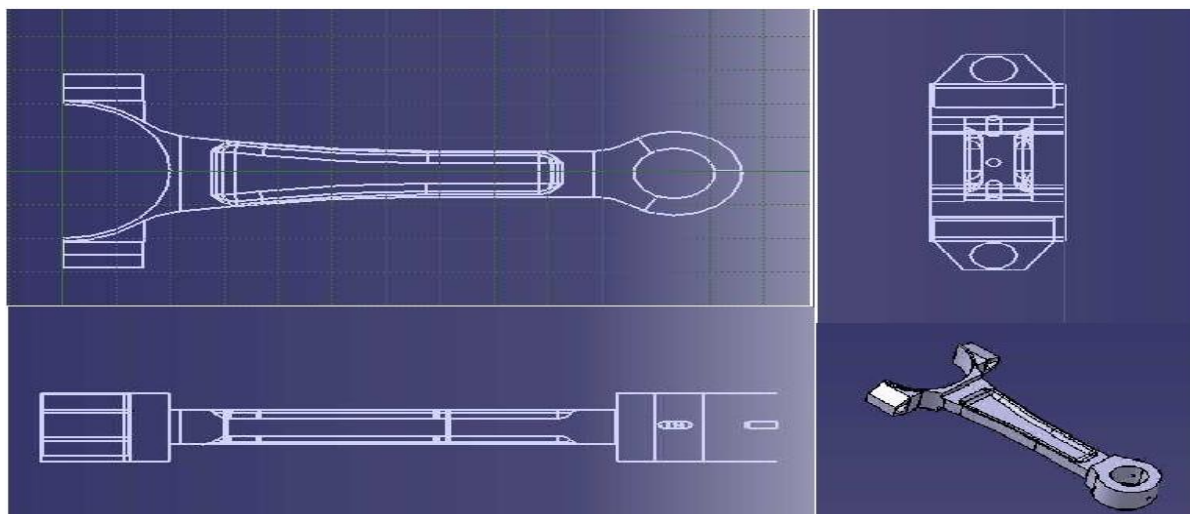


Figure 4.1: All the four views of connecting rod modeled in CATIA V5R12

b) In the next step, the model was imported to the ANSYS software, as shown below.

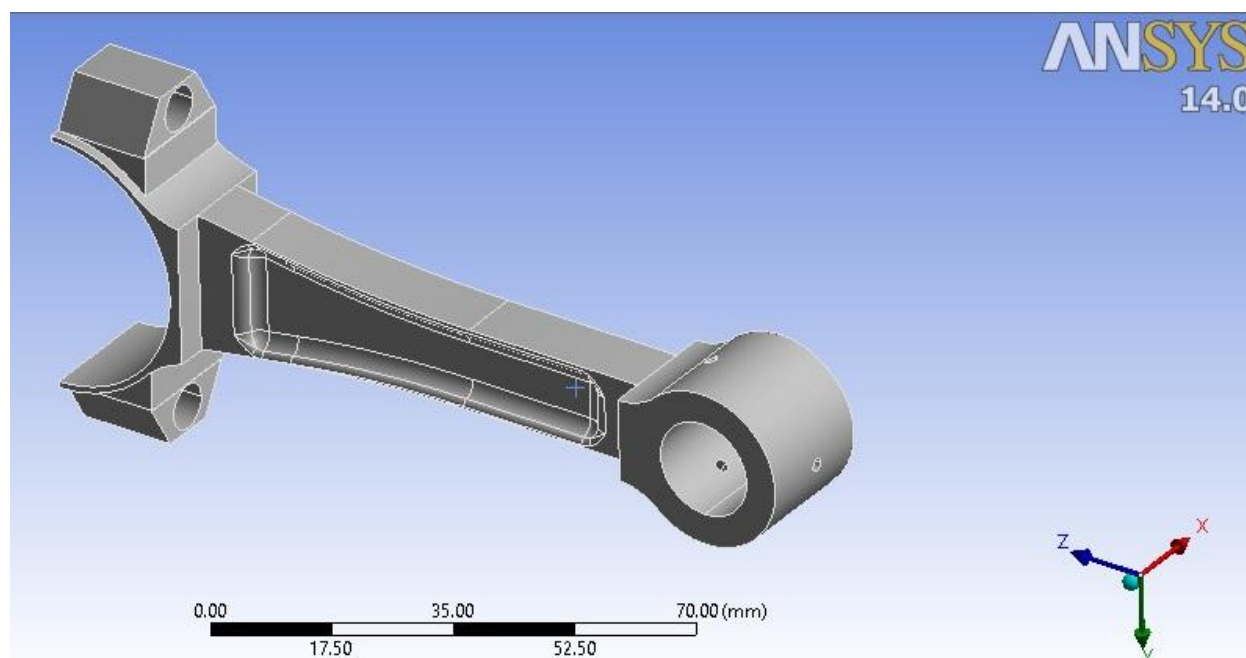


Figure 4.2: Imported model in ANSYS Software

In the next step, the material, forged steel was assigned to the imported model, the properties of which are presented as follows.

Table 4.1: Properties of Forged Steel

S.No	Mechanical Property	Value for forged steel
1.	Density (gm/cc)	7.7
2.	Average Hardness (HRB)	101
3.	Modulus of Elasticity (GPa)	221
4.	Yield Strength (MPa)	625
5.	Ultimate Strength (MPa)	625
6.	Poison's Ratio	0.29

- c) In the next step of the research, meshing of the model was performed, for the purpose of making it deformable.

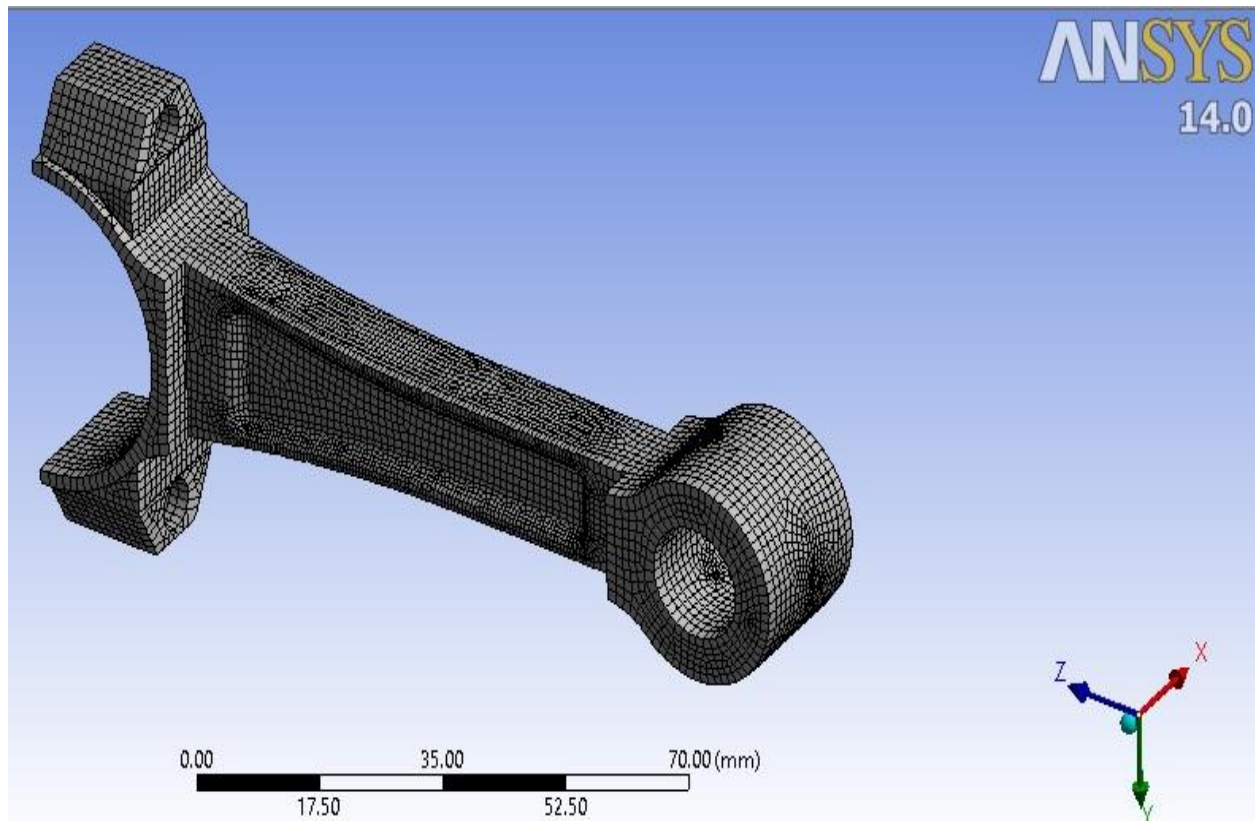


Figure 4.3: Meshed Model of Connecting Rod

Table given below presents the details of meshing.

Table 4.2: Details of Meshing

S. No	Entity	Details
1	Element type	Default
2	No. of nodes	51.3k
3	No. of cells	3152.1k

d) In the next step, the following boundary conditions were applied to the model.

- The big end is fixed;
- Load is applied on smaller end; and
- Maximum force will occur at zero degrees of crank angle or when the piston is at Top Dead Center (TDC).

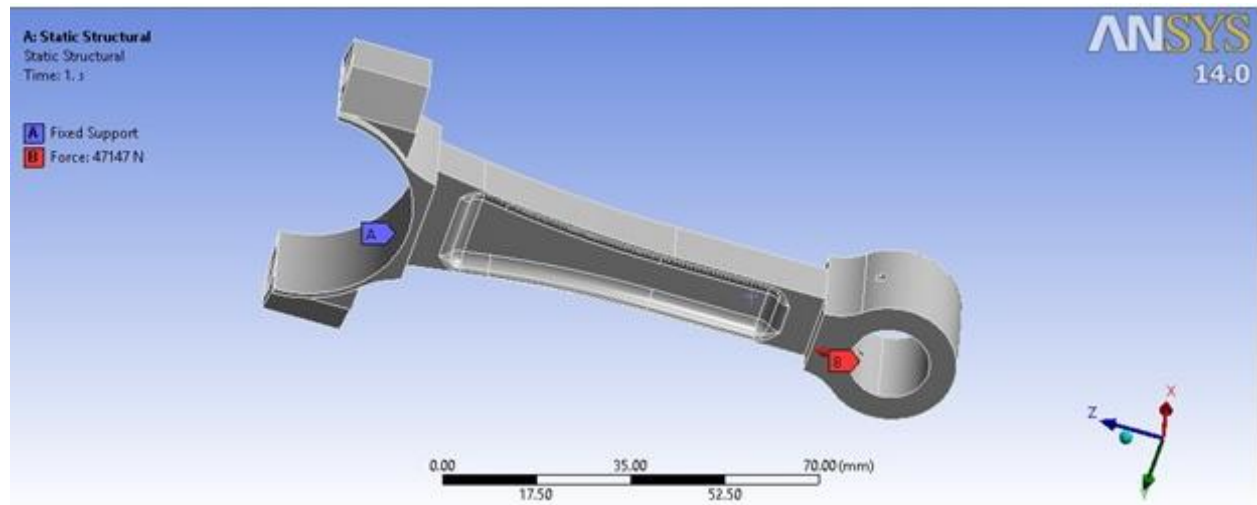


Figure 4.4: Application of Boundary Conditions on the Model

e) In the next step, the static structural analysis and modal analysis was carried out. For the purpose of calculations of gas force, the following engine specifications were used.

Table 4.3: Engine Specifications used for Calculations of Gas Force

S.No	Input Parameter	Value
1.	Diameter of Piston (d)	65.8 mm
2.	Displacement volume (Vs)	796 cc
3.	Number of cylinder (K)	3
4.	Swept volume per cylinder	0.0002653 m ³
5.	Length of connecting rod (l)	125 mm
6.	Torque (T)	69 N-m @ 3500 rpm
7.	Speed (N)	3500 rpm
8.	F _{max}	= 10. F _{imep}
9.	Mechanical efficiency	85 percent

f) In the next step of the research, gas force was calculated using the following equations.

Brake Power

$$B.P = \frac{2\pi NT}{60 \times 1000} \text{ KW} = 8.42 \text{ KW}$$

Indicated Power

$$I.P = \frac{B.P}{\eta_m} = 9.912 \text{ KW}$$

Mean Effective Pressure:

$$P_{imep} = \frac{I.P \times 60 \times 1000 \times 2}{L \times A \times n \times K} = 1.28 \text{ MPa}$$

Maximum Gas Pressure:

$$P_{max} = 10 \times P_{imep} = 12.80 \text{ MPa}$$

Maximum Gas Force:

$$F_{max} = P_{max} \times A = 47147.72 \text{ N}$$

g) In the next step, using the value of $F_{\max}$ structural analysis was conducted, and later, the modal analysis was also conducted.

5. Results and Discussion

The present section presents the details of results obtained and their associated discussion, as presented in upcoming sub-sections.

5.1 Results

Figure 5.1 to Figure 5.5 presents the results of the research work.

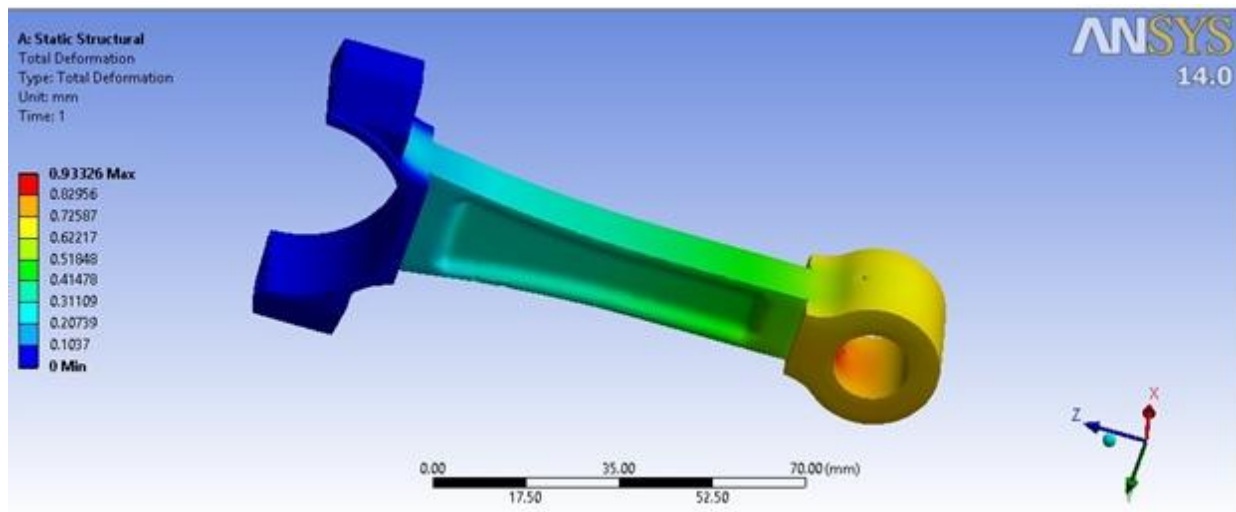


Figure 5.1: Total Deformation of connecting rod

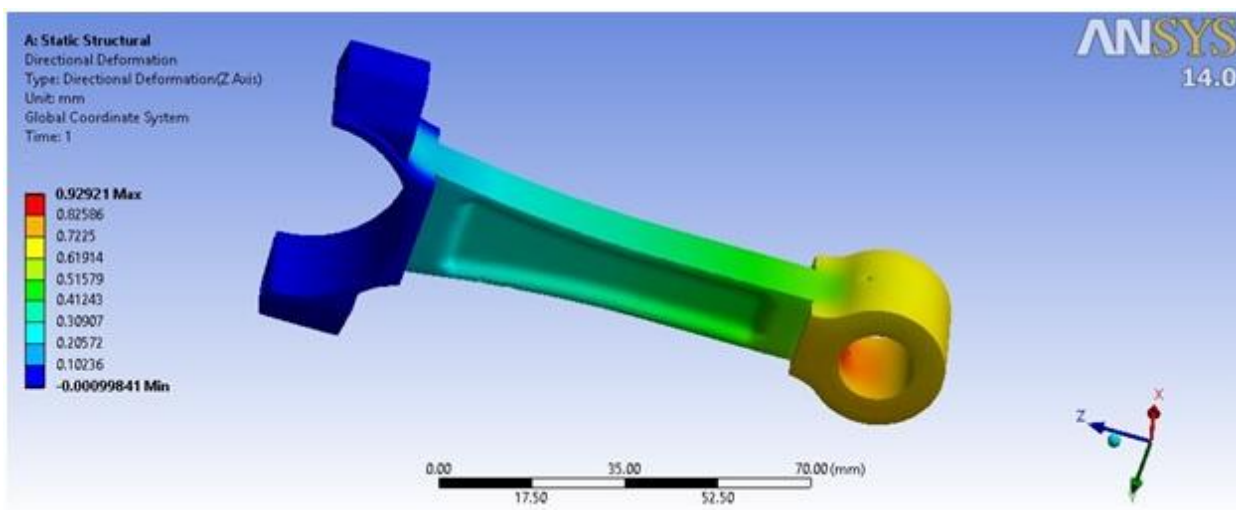


Figure 5.2: Directional Deformation (Z-Axis) of Connecting rod

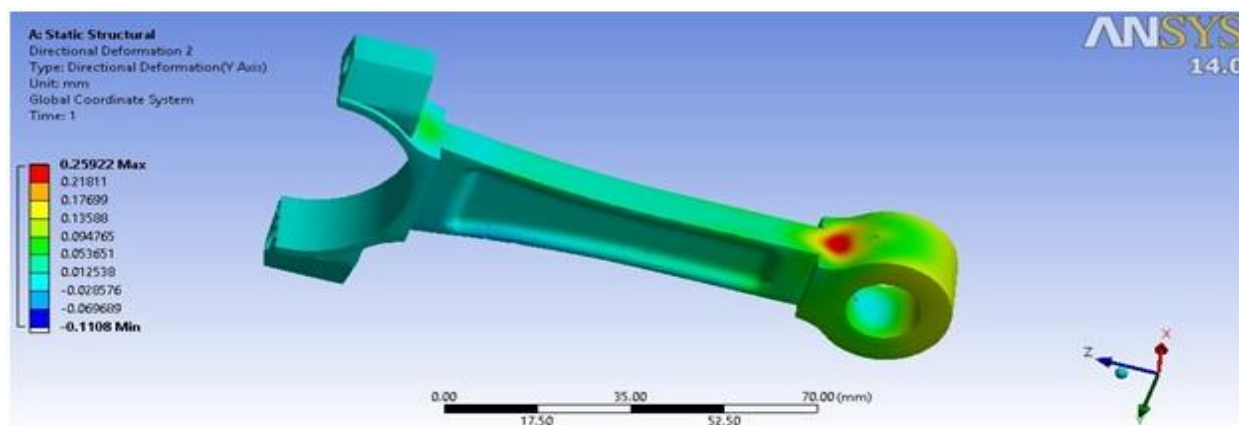


Figure 5.3: Directional Deformation (Y-Axis) of Connecting rod

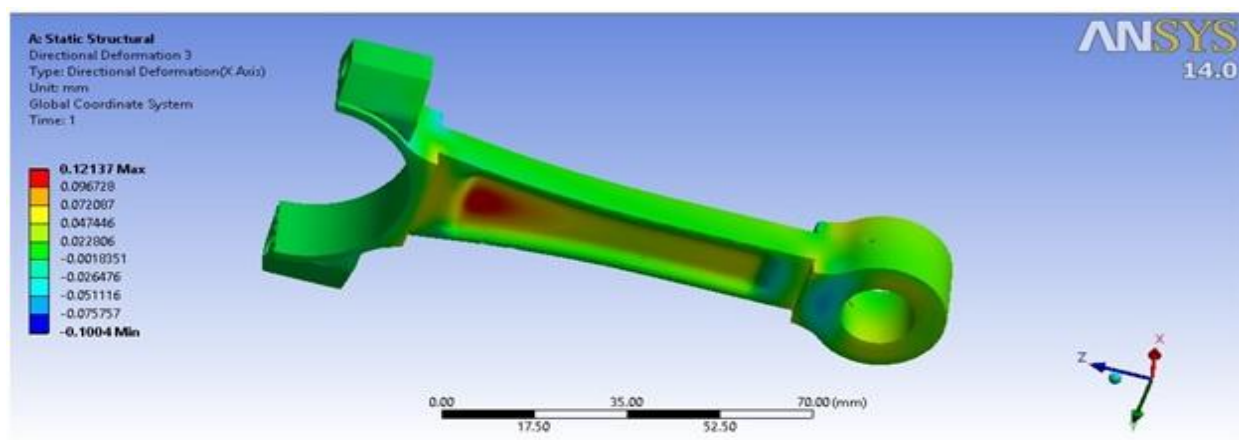


Figure 5.4: Directional Deformation (X-Axis) of Connecting rod

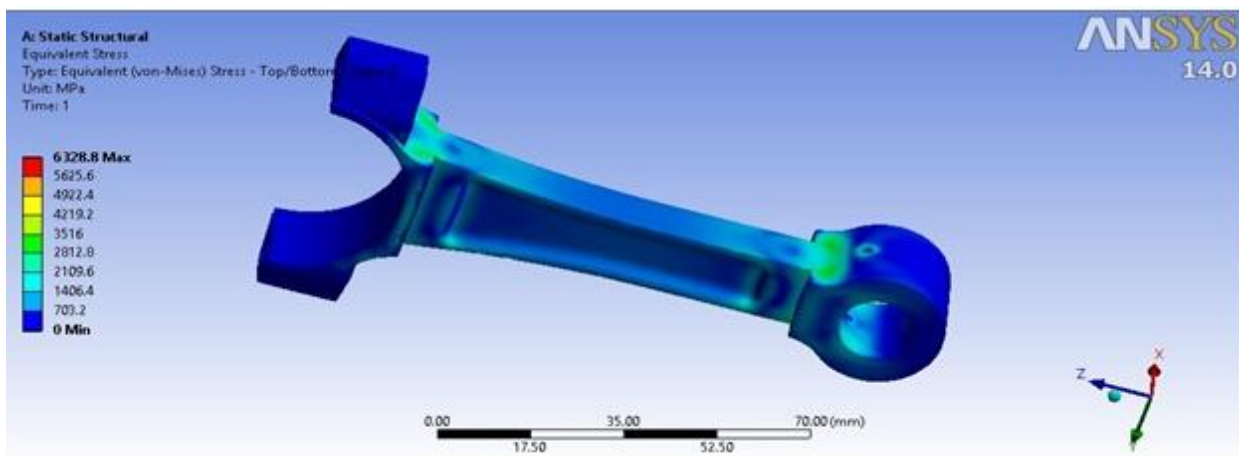


Figure 5.5: Equivalent Stress of Connecting rod

Table 5.1 presents the details of results obtained from static structural analysis.

Table 5.1: Results from Static Structural Analysis

S.No.	Parameter	Value
1.	Total Deformation	0.93326 mm
2.	Directional Deformation (Z Axis)	0.92921 mm
3.	Directional Deformation (Y Axis)	0.25922 mm
4.	Directional Deformation (X Axis)	0.12137 mm
5.	Equivalent Stress	703.2 Mpa

Figure 5.6 to 5.11 presents the results of modal analysis of connecting rod.

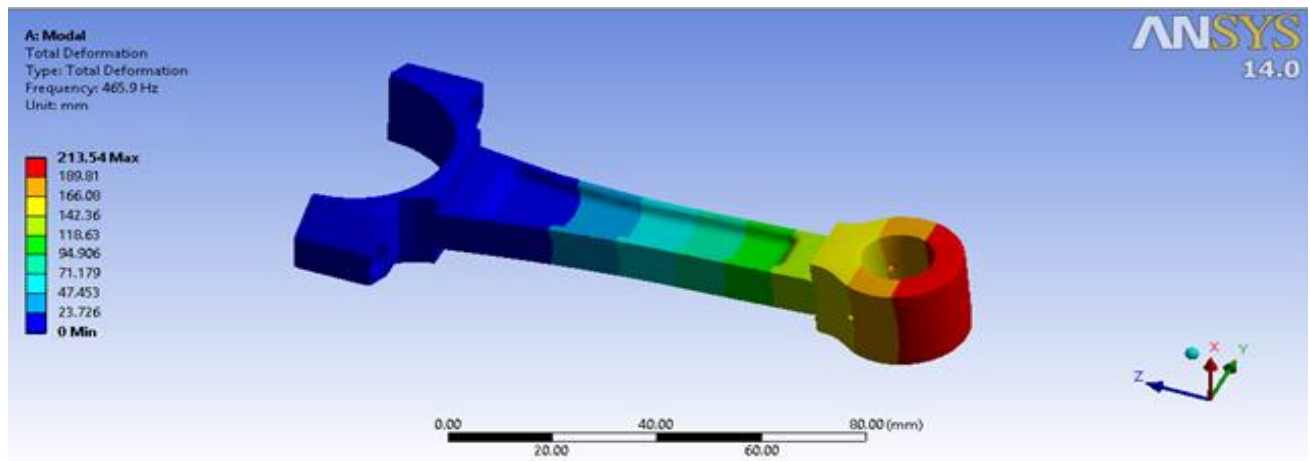


Figure 5.6: Total deformation for 1st mode shape

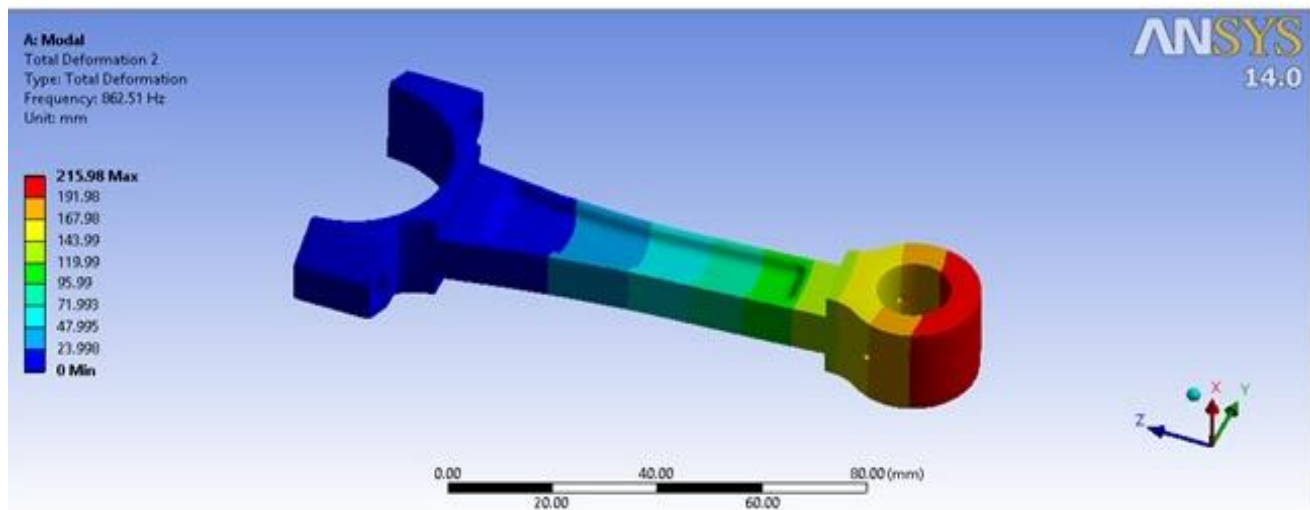


Figure 5.7: Total deformation for 2nd mode shape

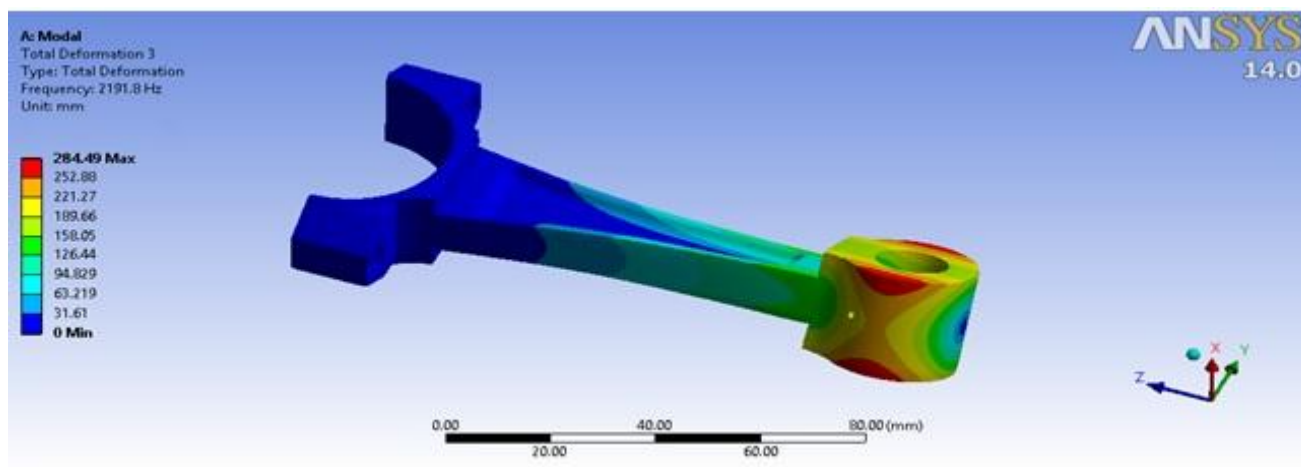


Figure 5.8: Total deformation for 3rd mode shape

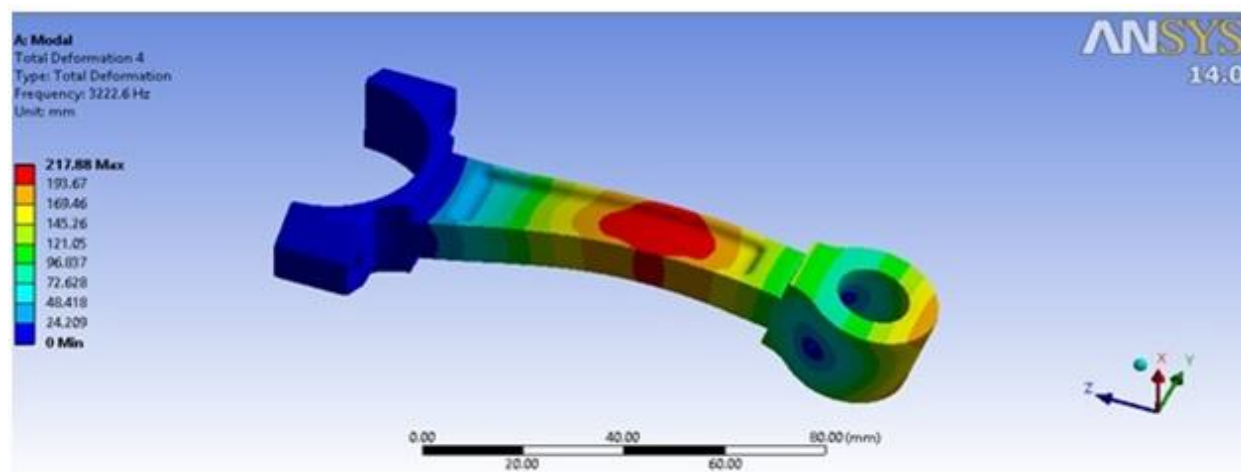


Figure 5.9: Total deformation for 4th mode shape

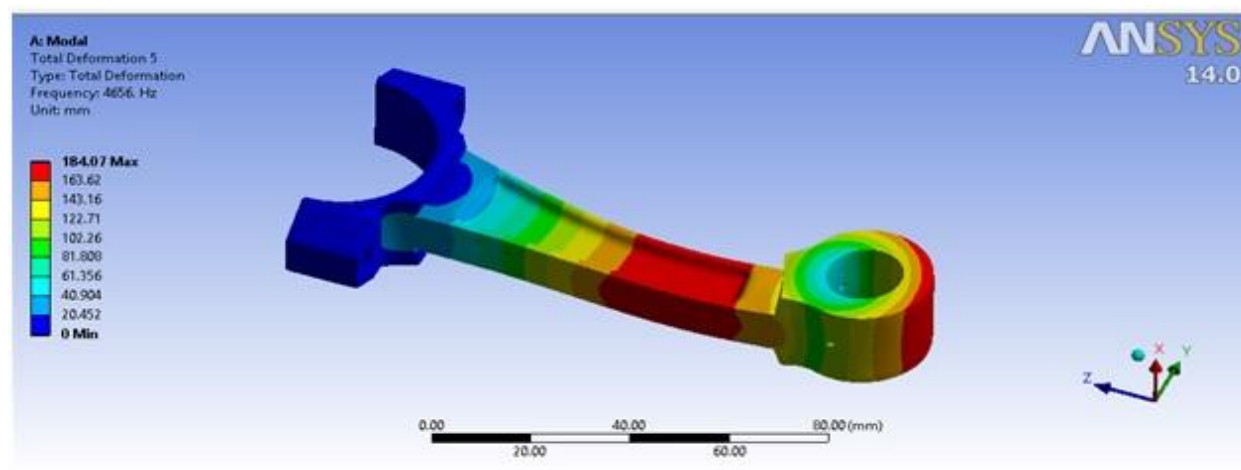


Figure 5.10: Total deformation for 5th mode shape

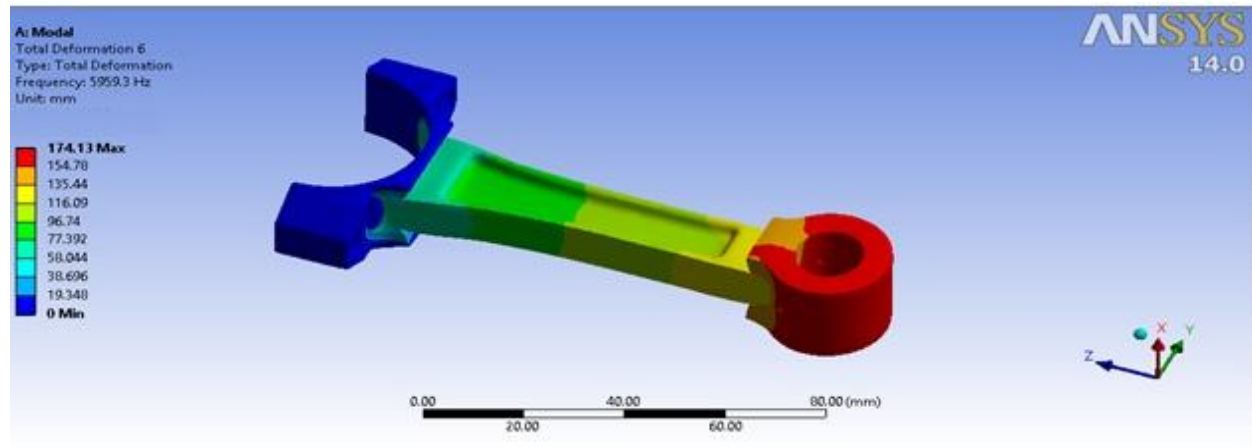


Figure 5.11: Total deformation for 6th mode shape

Table 5.2 presents the details of results obtained from modal analysis.

Table 5.2: Results of Modal Analysis

S.No	Mode Shape Number	Max. Total Deformation (mm)	Natural Frequency (Hz)
1.	1	213.54	465.9
2.	2	215.98	862.51
3.	3	284.49	2191.8
4.	4	217.88	3222.6
5.	5	184.07	4656
6.	6	174.13	5959.3

5.2 Discussion

The ANSYS static structural analysis of your connecting rod gives a good idea of how the part behaves under stress. The total deformation is around 0.93 mm, which means that's the maximum amount the rod moves or flexes when a load is applied. This isn't necessarily bad — all parts deform a little under stress — but the key is whether this amount of movement is acceptable in your design. If the connecting rod still functions well and doesn't interfere with other engine components at this level of deformation, then it's likely within safe limits.

Looking at the directional deformation in the Z-axis (which is usually the lengthwise direction of the rod), it's nearly the same as the total deformation — about **0.929 mm**. This tells us that almost all of the movement is happening along the rod's main axis. That makes sense, because the connecting rod is primarily designed to handle forces back and forth along its length, between the piston and the crankshaft. So far, so good — the rod is doing what it's supposed to do under load.

Now, when we look at the Y and X directions, the deformations are much smaller — 0.26 mm in Y and 0.12 mm in X. This means the rod isn't bending or twisting much sideways, which is exactly what we want. Too much movement in these directions could lead to vibration or even mechanical failure, so small values here are a good sign. It shows that the rod is stable and the load is being transferred along the right path.

The equivalent stress (also called Von Mises stress) comes out to about 703 MPa. This is a really important number because it tells us how close the rod is to failing. If you're using a material like forged steel, which typically has a yield strength higher than this — say 850 MPa or more — then you're in safe territory. But if your material has a lower yield strength, then you might want to rethink either the material or the design to prevent it from getting damaged under load.

The modal analysis looks at how the connecting rod vibrates naturally, without any external force continuously acting on it. The first mode shape shows a natural frequency of 465.9 Hz with a maximum deformation of 213.54 mm. This means that if the rod experiences a disturbance (like a sudden force or engine vibration) at around 466 Hz, it could start vibrating strongly in that first mode.

The second mode shape has a slightly higher deformation (215.98 mm) and a natural frequency of 862.51 Hz. Higher modes like this could represent bending in multiple directions or twisting, especially as the geometry of the part interacts with the frequency.

By the time we get to mode 3 and 4, the natural frequencies jump significantly to 2191.8 Hz and 3222.6 Hz, with deformations of 284.49 mm and 217.88 mm respectively. These modes are less likely to be excited under typical engine operating conditions but are still important to consider.

If any part of your system has components or harmonics that match these frequencies, there's a risk of resonance — which could damage the component over time.

Modes 5 and 6, with frequencies of 4656 Hz and 5959.3 Hz, show even less deformation (184.07 mm and 174.13 mm). These high-frequency modes tend to involve small, rapid vibrations, and while they might not cause large movements, they can still result in high local stresses — especially near joints, holes, or other stress concentration zones. These are worth noting for detailed design work or fatigue life studies.

6. Conclusions, Limitations and Future Scope of the Research Work

The present section tells about the conclusion, limitations and future scope of the research work, as presented in upcoming sections.

6.1 Conclusions

Following conclusions were obtained from the research work:

a) Conclusions from Static Structural Analysis

- The connecting rod primarily deforms along its length (Z-axis), with minimal deformation in lateral directions, indicating effective load transmission and structural stability under axial forces.
- The observed Von Mises stress of 703.2 MPa is significant and must be compared with the material's yield strength to ensure safety; if the stress exceeds allowable limits, a material or design change is necessary.
- The total deformation of 0.93326 mm is within acceptable limits, suggesting that the connecting rod maintains its structural integrity under load without excessive displacement that could affect engine performance or cause interference with adjacent components.

b) Conclusions form Modal Analysis

- The connecting rod exhibits distinct natural frequencies across six mode shapes, starting from 465.9 Hz up to 5959.3 Hz, with maximum theoretical deformations highlighting how the rod may vibrate in different patterns.

- Most deformation occurs in the lower modes, indicating that the connecting rod is more susceptible to vibration at lower frequencies — especially if those frequencies align with the engine's operating range.

6.2 Limitations and Future Scope of the Research

Following are the limitations of research work:

- a) The study assumes that the material behaves perfectly elastically, which isn't always the case in real life — especially under high stress or temperature.
- b) Only one static load condition was considered, but in an actual engine, the connecting rod faces rapidly changing and repeated loads.
- c) The way the connecting rod was held or supported in the simulation might not fully match how it's mounted in a real engine, which could affect the accuracy of the results.
- d) Since the findings are based purely on simulation, and no physical testing was done, it's hard to say how closely the results match real-world behavior.

The following points represent the future scope of the research work:

- a) Future work could include fatigue and thermal analysis to understand how the connecting rod performs over time and under heat — just like it would inside an actual engine.
- b) Trying out different materials or tweaking the design could help make the rod stronger, lighter, and better at handling vibrations.
- c) Doing real-world tests on a prototype could help confirm whether the simulation results hold up, and also improve the accuracy of future models.
- d) Adding more advanced simulations — like studying how the rod reacts to vibrations or sudden forces over time — would give a more complete picture of its performance.

References

- Choudhury, M. and Saberwal, G. (2019). The work, goals, challenges, achievements, and recommendations of orphan medicinal product organizations in india: an interview-based study. *Orphanet Journal of Rare Diseases*, 14(1). <https://doi.org/10.1186/s13023-019-1224-0>
- Chumbre, V. (2018). Design and comparative analysis of connecting rod using finite element analysis. *International Journal for Research in Applied Science and Engineering Technology*, 6(4), 765-773. <https://doi.org/10.22214/ijraset.2018.4129>

- Fatu, A., Hajjam, M., & Bonneau, D. (2005). Analysis of non-newtonian and piezoviscous effects in dynamically loaded connecting-rod bearings. *Proceedings of the Institution of Mechanical Engineers Part J Journal of Engineering Tribology*, 219(3), 209-224. <https://doi.org/10.1243/135065005x33892>
- Hasan, N. (2018). Simulation of connecting rod using finite element analysis. *International Journal of Innovative Research in Computer Science & Technology*, 6(5), 113-116. <https://doi.org/10.21276/ijircst.2018.6.5.5>
- Huo, F., Li, J., Xu, Y., Wu, B., Han, Y., Li, P., ... & Zhang, Q. (2012). Structural lightweight design of engine connecting rod., 1139-1148. https://doi.org/10.1007/978-3-642-33738-3_18
- Ilman, M. and Barizy, R. (2015). Failure analysis and fatigue performance evaluation of a failed connecting rod of reciprocating air compressor. *Engineering Failure Analysis*, 56, 142-149. <https://doi.org/10.1016/j.engfailanal.2015.03.010>
- Jia, Z., Hao, C., Sun, J., & Liu, X. (2013). Kinematics and dynamics analysis of piston-connecting rod mechanism of internal combustion engine. *Applied Mechanics and Materials*, 470, 539-542. <https://doi.org/10.4028/www.scientific.net/amm.470.539>
- Kumar, G. (2018). Implementation of process fmea during assembling of a connecting rod. *International Journal for Research in Applied Science and Engineering Technology*, 6(4), 1087-1096. <https://doi.org/10.22214/ijraset.2018.4185>
- Kurdi, O., Norman, M., Yulianti, I., & Rashid, M. (2012). Simulation of fatigue life prediction and enhancement of connecting rod of car engine. *Advanced Materials Research*, 557-559, 2410-2414. <https://doi.org/10.4028/www.scientific.net/amr.557-559.2410>
- Lubis, D. and Andoko, A. (2019). Elastic linear analysis of connecting rods for single cylinder four stroke petrol engines using finite element method. *Journal of Mechanical Engineering Science and Technology (Jmest)*, 3(1), 42-50. <https://doi.org/10.17977/um016v3i12019p042>
- Luo, Q. and Liu, X. (2018). Fatigue analysis of engine connecting rod based on finite element method.. <https://doi.org/10.2991/mees-18.2018.48>
- Luo, Q. and Liu, X. (2018). Fatigue analysis of engine connecting rod based on finite element method.. <https://doi.org/10.2991/mees-18.2018.48>
- Patel, A., Yadav, A., & singh, R. (2022). Comparative study of stress analysis on connecting rod using fem. *International Journal of Engineering Research in Mechanical and Civil Engineering (Ijermce)*, 9(7), 25-29. <https://doi.org/10.36647/ijermce/09.07.a006>
- Rajueni, K. and Choudhury, M. (2022). Assessing the availability of repurposed orphan drugs in india.. <https://doi.org/10.1101/2022.12.22.22283870>
- Singh, G., Ghosh, B., Dave, K., & Somsekhar, V. (2008). Screening of venlafaxine hydrochloride for transdermal delivery: passive diffusion and iontophoresis. *Aaps Pharmscitech*, 9(3), 791-797. <https://doi.org/10.1208/s12249-008-9111-3>
- Singh, P. and Pramanik, D. (2016). Structural analysis of connecting rod using fea / sonlu elemanlar analizi ile rod bağlama yapısal analizi. *International Journal of Automotive Engineering and Technologies*, 4(4), 245. <https://doi.org/10.18245/ijaet.69316>

- Singh, P. and Pramanik, D. (2016). Structural analysis of connecting rod using fea / sonlu elemanlar analizi ile rod bağlama yapısal analizi. International Journal of Automotive Engineering and Technologies, 4(4), 245. <https://doi.org/10.18245/ijaet.69316>
- Sriharsha, B. and Rao, P. (2020). Design considerations for connecting rod. International Journal of Engineering and Advanced Technology, 9(3), 2368-2373. <https://doi.org/10.35940/ijeat.c5759.029320>
- Wang, G., Guo, L., Sun, W., & Yan, J. (2012). Strength calculation analysis of diesel engine linkage based on contact finite element method and experimental study. Advanced Materials Research, 569, 539-545. <https://doi.org/10.4028/www.scientific.net/amr.569.539>
- Yang, H., Kou, S., Gao, W., & Shen, Y. (2014). Investigation of crack line offset of fracture splitting connecting rod. Applied Mechanics and Materials, 472, 100-104. <https://doi.org/10.4028/www.scientific.net/amm.472.100>
- Zhang, J. (2017). Finite element analysis of ea 113 gasoline engine connecting rod. International Journal of Mechanical Engineering and Applications, 5(4), 208. <https://doi.org/10.11648/j.ijmea.20170504.14>
- Zhang, J., Zhang, S., & Zhang, Y. (2012). The analysis of the deformation and contact lubrication problem of hpd diesel engine connecting rod bearings based on the fft method and flexibility matrix method. Advanced Materials Research, 602-604, 2170-2173. <https://doi.org/10.4028/www.scientific.net/amr.602-604.2170>