IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 6, Issue 9, March 2026
Impact Factor: 8.2

# Design and Structural Analysis of a Connecting Rod Using Maraging Steel

**Dr. M. Srinivasa Rao[1], A. Abinay[2], B. Suresh[3], A. Abhinav[4], M. Pavani[5]**
Associate Professor &COEMECH, ACE Engineering College, Ghatkesar, India [1]
Assistant Professor, MECH, ACE Engineering College, Ghatkesar, India[2]
Students, MECH, ACE Engineering College, Ghatkesar, India[3-5]

**Abstract:** *The connecting rod is a critical component of an internal combustion engine, responsible for transmitting forces between the piston and the crankshaft under highly dynamic loading conditions. This project focuses on the design and analysis of a connecting rod using maraging steel, an ultra-high-strength alloy known for its exceptional toughness, high fatigue resistance, and excellent dimensional stability. A three-dimensional (3D) model of the connecting rod is developed using computer-aided design software and analyzed through finite element analysis(FEA) to evaluate its structural behaviour under peak engine loading conditions. The analysis is carried out to determine the stress distribution, total deformation, and factor of safety of the connecting rod. The performance of the component is assessed to ensure that it can withstand the applied loads without failure. The results obtained from the simulation demonstrate that maraging steel provides superior strength and durability compared to conventional materials used for connecting rods. The analysis also helps in identifying critical stress regions and evaluating the structural integrity of the design. The study concludes that maraging steel is a highly suitable material for connecting rod applications in high-performance engines due to its high strength-to-weight ratio, excellent fatigue properties, and ability to withstand severe operating conditions. This work highlights the effectiveness of design and finite element analysis in improving the reliability and performance of engine components*

**Keywords:** Connecting rod, maraging steel, finite element analysis (FEA), stress analysis, total deformation, factor of safety, internal combustion engine, structural analysis, high-strength alloys, and mechanical design

## I. INTRODUCTION

The connecting rod is a vital component of an internal combustion engine that connects the piston to the crankshaft and plays an important role in the transmission of power. It transfers the force generated during the combustion of fuel in the cylinder from the piston to the crankshaft. Through this mechanism, the reciprocating motion of the piston is converted into the rotational motion of the crankshaft, which ultimately drives the engine and produces mechanical power. Because of its critical function in power transmission, the connecting rod must be designed with high precision and strength to withstand the complex forces acting on it during engine operation.

During the working cycle of an engine, the connecting rod experiences various types of stresses such as compressive stress during the power stroke, tensile stress during the exhaust and intake strokes, and bending stress due to inertia forces. These stresses occur repeatedly at very high speeds, which may lead to fatigue failure if the component is not properly designed. Therefore, the connecting rod must have sufficient strength, stiffness, and fatigue resistance to ensure safe and reliable engine performance.

Traditionally, connecting rods are manufactured using materials such as carbon steel, alloy steel, or aluminum alloys. However, with the increasing demand for higher engine performance and durability, advanced materials with superior mechanical properties are being considered. In this project, maraging steel is selected as the material for the connecting rod. Maraging steel is a high-strength alloy steel known for its excellent mechanical properties such as very high tensile strength, superior toughness, good fatigue resistance, and excellent dimensional stability. These properties make maraging steel highly suitable for components that operate under heavy loads and cyclic stresses.

To achieve an efficient and reliable design, modern engineering tools such as Computer-Aided Design (CAD) and Finite Element Analysis (FEA) are widely used. CAD software helps engineers create accurate three-dimensional models of mechanical components, while FEA software allows detailed analysis of stress, strain, and deformation under various loading conditions. These tools reduce the need for physical prototypes and help engineers evaluate the performance of a design before manufacturing.

In this project, the connecting rod is first modeled using CATIA, which is a powerful CAD software used for designing complex mechanical components. CATIA allows the creation of precise 3D models based on the required dimensions and design parameters. After the modeling process is completed, the 3D model is imported into ANSYS for structural analysis. ANSYS is a widely used FEA software that helps analyze how a component behaves under different loads and constraints.

Using ANSYS, the connecting rod is subjected to structural analysis to determine parameters such as total deformation, equivalent (von Mises) stress, and elastic strain. These parameters help in understanding how the connecting rod behaves when exposed to engine operating conditions. The results obtained from the analysis help identify the regions where maximum stress occurs and ensure that the stress values are within the allowable limits of the selected material.

The aim of this project is to design and analyze a connecting rod using maraging steel and evaluate its structural performance under applied loads using finite element analysis. By studying the stress distribution, strain, and deformation, it is possible to determine whether the designed connecting rod is safe and reliable for engine applications. This study also helps in improving the design efficiency and selecting suitable materials for high-performance engine components.

## II. METHODOLOGY AND MATERIAL

### 2.1 Overview of Methodology

The present study focuses on the design and structural analysis of a connecting rod made of Maraging Steel using Finite Element Analysis (FEA). The methodology adopted for this project involves several systematic steps to evaluate the structural performance of the connecting rod under operating conditions.

Initially, the geometrical model of the connecting rod is created using CAD software based on standard engine dimensions. The designed model is then imported into ANSYS Workbench for further analysis. Material properties corresponding to Maraging Steel, such as density, Young's modulus, Poisson's ratio, and yield strength, are assigned to the model.

In the next stage, meshing of the connecting rod is performed to divide the geometry into finite elements for accurate numerical analysis. Proper mesh refinement is ensured in critical regions such as the big end, small end, and shank area where stress concentration is expected.

After meshing, boundary conditions and loading conditions are applied to simulate the actual working conditions of the connecting rod. The crank end is constrained while the piston end is subjected to compressive load representing the gas pressure developed inside the engine cylinder.

Subsequently, static structural analysis is carried out to determine important parameters such as total deformation, equivalent (Von Mises) stress, and strain distribution. The obtained results help in identifying critical stress regions and evaluating the structural integrity of the connecting rod.

Finally, the results obtained from the analysis are interpreted and discussed to determine whether the connecting rod made from Maraging Steel can safely withstand the applied loads without failure.

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 6, Issue 9, March 2026
Impact Factor: 8.2

### 2.2. MECHANICAL PROPERTIES OF MARAGING STEEL

| `Property | Symbol / Unit | Typical Value |
|---|---|---|
| Density | ρ | 8.0 g/cm³ |
| Young's Modulus | E | 200 GPa |
| Ultimate Tensile Strength | σ | 2000–2400 MPa |
| Hardness | HRC | 50–55 |
| Poisson's Ratio | ν | 0.3 |
| Elongation at Break | % | 5–8 |
| Fatigue Strength | MPa | ~600–800 |
| Thermal Conductivity | W/m·K | 25 |
| Specific Heat | J/kg·K | 460 |

TABLE I

## III. MODEL OF CONNECTING ROD

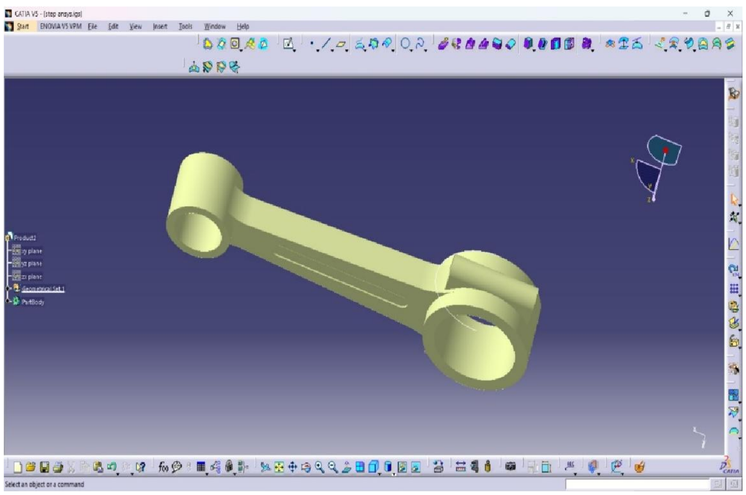

Fig. 1Model of Connecting Rod

The connecting rod is a critical component of an internal combustion engine that connects the piston to the crankshaft and transmits the combustion force. The design of the connecting rod must ensure sufficient strength and rigidity to withstand the compressive and tensile forces generated during engine operation.

• In this study, the connecting rod is designed based on standard engine design considerations. The geometry of the connecting rod consists of three main parts: the small end, shank, and big end. The small end is connected to the piston through the piston pin, while the big end is connected to the crankshaft. The shank portion is designed with an appropriate cross-section to provide high strength while minimizing weight.

• The design model of the connecting rod is developed using CAD software and later imported into ANSYS Workbench for structural analysis. During the design stage, important parameters such as dimensions, material properties, and loading conditions are considered to ensure accurate analysis.

• Maraging Steel is selected as the material for the connecting rod due to its excellent mechanical properties such as high strength, toughness, and good fatigue resistance. These properties make it suitable for high-performance engine components that are subjected to cyclic loading conditions.

• The designed connecting rod model is then used for finite element analysis (FEA) to evaluate parameters such as stress distribution, deformation, and strain under applied loads. The results obtained help in assessing the structural performance and reliability of the connecting rod design.

## IV. ANALYSIS

A rough answer to a wide range of engineering issues. Although it was originally designed to investigate stresses in complex aircraft structures, it has now been expanded and applied to the broader field of continuum mechanics. Engineering institutions and business are paying close attention to it because of its versatility and adaptability as an analysis tool. The finite element method has evolved into a formidable tool for solving a wide range of engineering problems numerically. Because complex issues may be modelled and released with relative ease, advances in computer technology and CAD systems have led to growing usage of FEM in research and industry. 5.2 Basic steps in the Finite Element Analysis : a) Pre processing phase: create and discretize the solution domain into finite elements i.e subdivide the real continuum into nodes and elements. • Assume a shape function to represent the physical behavior of an element; that is an approximate continuous function is assumed to represent the solution of an element. • Develop equations for all the elements in the mesh.. • These generally take form • [K][U] = [F] • Where [K]' is a square matrix, known as stiffness matrix • (U) is the vector of (unknown) nodal displacements or temperature • {F}' is the vector of applied nodal forces • Assemble the elemental equations to obtain the equations of the whole problem. Construct the global stiffness matrix. • Apply boundary conditions, initial conditions, and loading. 26 b) Solution Phase: Solve a set of linear or nonlinear algebraic equations simultaneously to obtain nodal results of primary degrees of freedom or unknowns, such as displacement values at different nodes in structural problem or temperature values at different nodes in heat transfer problem. c) Post processing phase: • Computation of any secondary unknowns or variables e.g., the gradient of the solution. • Interpretation of the results to check whether the solution makes sense. • Tabular and/or graphical presentation of the result.

### TYPES OF STRUCTURAL ANALYSIS

The types of structural analyses available in the ANSYS family of products are explained below. The primary unknowns (nodal degrees of freedom) calculated in a structural analysis are displacements. Other quantities, such as strains, stresses, and reaction forces, are then derived from the nodal displacements. Structural analyses is available in the ANSYS/Multi physics, ANSYS/Mechanical, ANSYS/Structural, and ANSYS/Linear Plus programs only. One can perform the following types of structural analyses 1) Static Analysis: Used to determine displacements, stresses, etc., under static loading conditions. It comprises of both linear and non-linear static analysis. Non linearity can include plasticity, stress stiffening, large deflection, large strain, hyper elasticity, contact surfaces, and creep. 2) Modal Analysis: Used to calculate the natural frequencies and mode shapes of a structure. Different mode extraction methods are available. 3) Harmonic Analysis: Used to determine the response of a structure to harmonically time-varying loads. 29 4) Transient Dynamic Analysis: Used to determine the response of a structure to arbitrarily time-varying loads. All non-linearity mentioned under Static Analysis above are allowed. 5) Spectrum Analysis: An extension of the modal analysis, used to calculate stresses and strains due to a response spectrum or a PSD input (random vibrations). 6) Buckling Analysis: Used to calculate the buckling loads and determine the buckling mode shape. Both linear (Eigen value) buckling and nonlinear buckling analyses are possible.

### PROCEDURE FOR PERFORMING STATIC STRUCTURAL ANALYSIS

**STEP 1:** Selection of analysis feature Open Ansys workbench and then select static structural analysis from left side tool bar Structural analysis interface.

**STEP 2: ENGINEERING DATA**

The data to be calculated is to be submitted in the module properties such as yield strength, young's modulus, Poissons's ratio, F.O.S are to be provided. Engineering data interface.

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 6, Issue 9, March 2026
Impact Factor: 8.2

**STEP 3: INSERTION OF GEOMETRY**

Right click on the geometry and then click on import geometry. Then close the present tab and again right click on the geometry then click on the modify designer tool. Import of Geometry.

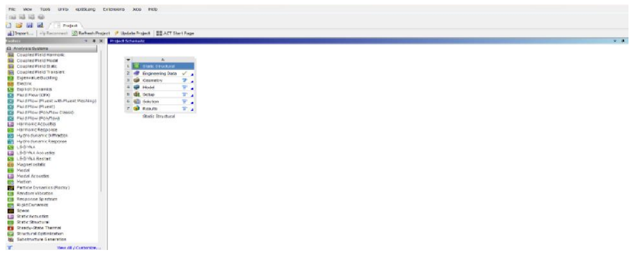

Fig.2. Introduction to Ansys

**STEP 5: MESHING OF THE MODEL**

Meshing is one of the most important steps in Finite Element Analysis. In this process, the entire geometry is divided into a large number of small elements called finite elements. These elements are connected at nodes and together form the mesh of the structure.

The accuracy of the simulation largely depends on the quality of the mesh. A finer mesh produces more accurate

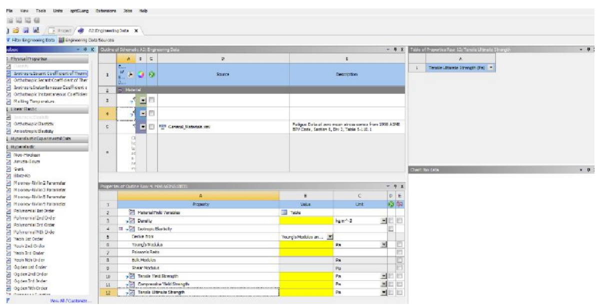

Fig.3. Adding Material Properties results but requires more computational time.

The meshing procedure followed in this project includes:

1. Opening the ANSYS Mechanical Environment.
2. Selecting the mesh option.
3. Defining the element size (typically between 2 mm to 5 mm for this model).
4. Generating the mesh automatically.

After meshing, the connecting rod model consists of thousands of elements that allow the solver to perform numerical calculations efficiently.

**STEP 6. APPLICATION OF BOUNDARY CONDITIONS**

To simulate real operating conditions of the engine, appropriate boundary conditions are applied to the connecting rod model.

Fixed Support

The big end of the connecting rod is connected to the crankshaft and is assumed to be constrained during analysis. Therefore, a fixed support is applied to the big end inner surface.

Load Application

The load applied on the connecting rod is due to the gas pressure acting on the piston during the combustion process. The load is calculated using the equation:

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 6, Issue 9, March 2026
Impact Factor: 8.2

The piston diameter used in this project is 85 mm and the maximum gas pressure is 7 MPa.
The piston area is calculated using:
Using this area, the force acting on the connecting rod is calculated and applied at the small end of the connecting rod in the form of axial force.

**STEP.7. SOLUTION AND SIMULATION**

Once the geometry, material properties, mesh, boundary conditions, and loads are defined, the next step is to solve the model.
ANSYS uses numerical techniques to solve the finite element equations and determine the structural response of the connecting rod. The simulation calculates the distribution of stresses, strains, and deformation throughout the model.
The solver processes the data and generates results based on the applied loading conditions.

**STEP.8. POST-PROCESSING AND RESULT EVALUATION**

After the simulation is completed, the results are evaluated using the post-processing tools available in ANSYS Mechanical.
The following results are obtained from the analysis:
Total Deformation
Total deformation represents the displacement experienced by the connecting rod due to applied loads.
Equivalent Stress (Von-Mises Stress)
Von-Mises stress is used to determine whether the material will yield under the applied load. It is one of the most important criteria used in structural analysis.
Equivalent Strain
Strain represents the deformation per unit length experienced by the material when subjected to stress.
These results are displayed in the form of colour contour plots, which help identify the regions of maximum stress concentration.

**STEP.9. VALIDATION OF RESULTS**

The simulation results obtained from ANSYS are compared with theoretical calculations to verify the accuracy of the analysis.
If the maximum stress obtained from the simulation is less than the yield strength of the material, the connecting rod design is considered safe.

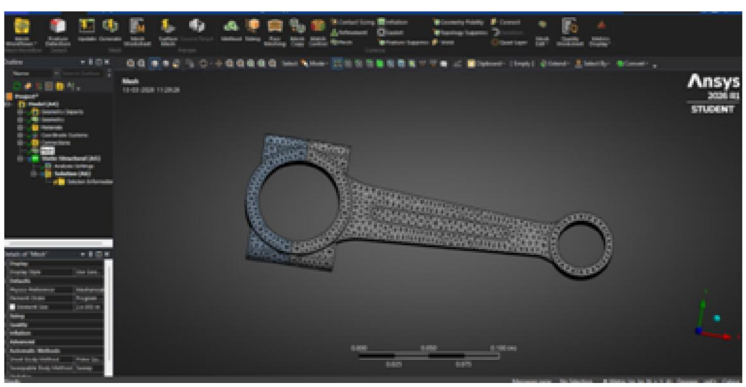


Fig.4. Meshing of Connecting rod



DOI: 10.48175/IJARSCT-32716

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 6, Issue 9, March 2026
Impact Factor: 8.2

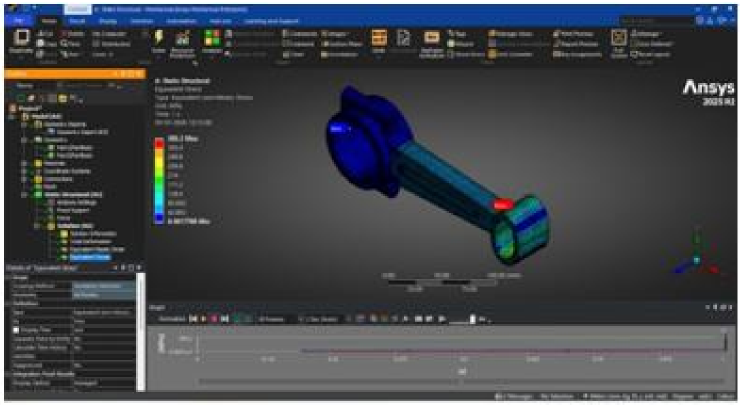

Fig.5.Equivalent stress

**Results Obtained from ANSYS Analysis**

The structural analysis of the connecting rod was carried out using the Finite Element Method in ANSYS. The Von-Mises stress distribution across the connecting rod was obtained after applying the calculated load and boundary conditions.

The simulation results show the following values:

Maximum Equivalent Stress=385.2MPa

Minimum Equivalent Stress = 0.0017 MPa

The maximum stress occurs in regions where the geometry experiences the highest load concentration, typically near the small end or fillet regions of the connecting rod. These areas are critical because stress concentration is usually higher there due to changes in geometry.

The minimum stress value is observed in regions where the load effect is minimal or where the structure remains relatively undeformed.

Material Consideration

YieldStrength≈2000MPa

Ultimate Tensile Strength ≈ 2100 MPa

Since the maximum Von-Mises stress developed in the connecting rod is significantly lower than the yield strength of maraging steel, the material will not experience permanent deformation under the given loading conditions.

This confirms that the connecting rod design is structurally safe and capable of handling the applied engine forces without failure.

The equivalent stress analysis performed using ANSYS demonstrates that the maximum stress generated in the connecting rod is 385.2 MPa, which is considerably lower than the yield strength of maraging steel (approximately 2000 MPa). The calculated Factor of Safety of 5.2 further confirms that the component operates well within safe limits.

Therefore, the connecting rod made of maraging steel is structurally stable and safe for the given operating conditions, and the design can be considered reliable for use in internal combustion engine applications.

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429 Volume 6, Issue 9, March 2026 Impact Factor: 8.2

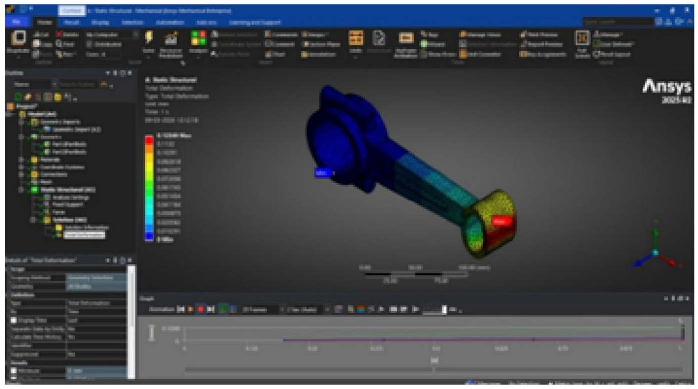

Fig.6.Total Deformation

Total deformation represents the overall displacement of the structure when an external load is applied. It shows how much the component changes its position or shape due to the applied forces. In structural analysis using ANSYS Workbench, total deformation is an important parameter used to evaluate the stiffness and rigidity of the component.

In this analysis, the deformation of the connecting rod was calculated after applying the load corresponding to the gas pressure acting on the piston. The deformation results obtained from the simulation are as follows:

Maximum Deformation=0.12349mm

Minimum Deformation = 0 mm

The maximum deformation occurs at the region where the load is directly applied, usually near the small end of the connecting rod, while the minimum deformation occurs at the constrained region where the fixed support is applied.

The obtained maximum deformation value of 0.12349 mm is very small when compared with the overall dimensions of the connecting rod. This indicates that the component experiences only a negligible displacement under the applied loading conditions.

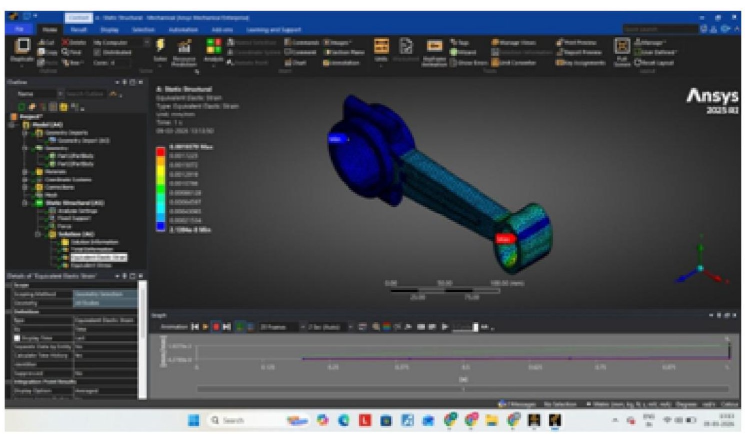

Fig.7.Equivalent Strain

Equivalent strain represents the deformation per unit length experienced by a material when it is subjected to external forces. It indicates how much the material stretches or compresses relative to its original dimensions. In structural simulations performed using ANSYS Workbench, equivalent strain is used to evaluate whether the material remains within its elastic limit or if it approaches plastic deformation.

During the analysis of the connecting rod, the strain distribution across the entire model was obtained after applying the calculated load and boundary conditions. The simulation results show the following values:

Maximum Strain=0.0019379mm/mm

Minimum Strain = $2.13 \times 10^{-8}$ mm/mm

The maximum strain occurs at regions where the stress concentration is higher, generally near the small end or fillet regions of the connecting rod, where the load transfer from the piston takes place. These areas experience greater deformation because they are directly affected by the applied force. On the other hand, the minimum strain occurs in regions where the load effect is very small or near the constrained areas where the structure is fixed.

The obtained maximum strain value of 0.0019379 mm/mm is relatively small and lies well within the elastic region of the material. The connecting rod in this study is assumed to be made of Maraging Steel, which is known for its high strength, toughness, and excellent resistance to deformation.

## V. CONCLUSION

The present work focused on the design and structural analysis of a connecting rod used in an internal combustion engine. The connecting rod model was created using CATIA and analyzed using ANSYS Workbench to study its structural behavior under the applied load conditions.

From the simulation results, the maximum equivalent (Von-Mises) stress obtained was 385.2 MPa, which is much lower than the yield strength of Maraging Steel (approximately 2000 MPa). The calculated factor of safety is about 5.2, indicating that the connecting rod is safe under the given loading conditions.

The maximum deformation observed was 0.12349 mm, which is very small, showing that the structure is rigid and stable. The maximum strain value of 0.0019379 mm/mm is also within the elastic limit of the material, meaning the component will return to its original shape after removing the load.

Therefore, the analysis confirms that the connecting rod design is structurally safe, reliable, and capable of withstanding the applied engine loads without failure.

## REFERENCES

**[1].** T. Faris, "Finite Element Analysis of Connecting Rod under Static Loading," International Journal of Mechanical Engineering.

**[2].** S. Kumar and P. Singh, "Fatigue Behavior of Steel and Composite Connecting Rods," Materials Today.

**[3].** A. Joseph, "Design Optimization of IC Engine Connecting Rod Using FEA," Journal of Engineering Design & Analysis.

**[4].** R. Anusha, "Mechanical Evaluation of Maraging Steel for High-Performance Applications," Journal of Materials Science Research.

**[5].** M. Patel, "Prediction of Mechanical Properties Using Machine Learning for Automotive Components," AI in Engineering Journal.

**[6].** D. Rao, "Stress and Vibration Analysis of Connecting Rod Using ANSYS," International Journal of Vehicle Engineering.

**[7].** R. Sharma, "Comparative Study of Conventional Steel and Maraging Steel Connecting Rods," International Journal of Manufacturing Science.