ICEST-2K24 [International Conference on Engineering, Science and Technology]
In Association with
International Journal of Scientific Research in Science, Engineering and Technology
Print ISSN - 2395-1990 | Online ISSN : 2394-4099 (www.ijsrset.com)

Design and Development of Connecting Rod by Using Analysis Software

Kautkar Nitin Uttamrao, Vikas Bhosale, Rohan Kshirsagar, Vikram Surve, Kisan Shelake, Abhijit Dhavale
Department of Mechanical Engineering, SVERI College of Engineering Pandharpur, Solapur, Maharashtra, India

ABSTRACT
The connecting rod is a crucial component in internal combustion engines, converting the reciprocating motion of the piston into the rotational motion of the crankshaft. This study aims to analyze the structural integrity and performance of a connecting rod using ANSYS, a sophisticated engineering simulation software. The primary focus is on evaluating the stress distribution, Eigen value of buckling, deformation and factor of safety under static loading conditions. This study focuses on the structural analysis of an engine connecting rod using ANSYS software. The analysis includes defining boundary conditions, such as fixed support at the crankshaft end and applying axial loads at the piston pin end. Finite Element Analysis (FEA) is used to evaluate stress distribution, deformation and potential failure points under operating conditions. The study aims to optimize the design for improved strength and durability while minimizing weight. The results provide insights into material selection and geometric modifications to enhance the connecting rod's performance.
Keywords – Connecting rod, Analysis, Finite Element Analysis (FEA)

I. INTRODUCTION

A connecting rod also called a 'con rod', is the part of a piston engine which connects the piston to the crankshaft. Together with the crank, the connecting rod converts the reciprocating motion of the piston into the rotation of the crankshaft. The material used for the connecting rod it must be light and strong enough to withstand stress and twisting force. The connecting rod is made of nickel, high grade alloy steel, chrome and chrome vanadium steel [1]. For small engine the material used is aluminium & structural steel. The upper end of the connecting rod has a hole for the piston pin. The lower end of the connecting rod is split so that the connecting rod can be installed on the crankshaft. Connecting rods are expected to withstand significant cyclic and inertial loads that are elevated by directional changes at the end of each stroke [2].
Failure of rod bolts, incorrect tightening, or poor maintenance can all lead to connecting rod failure. Failures are typically observed in competitive driving events. Since connecting rods are made with a high level of safety in mind, these kinds of failures are rarely common [3]. There are two possible combinations for the connecting rod's cross-section. I-beam and H-beam or the two in combination. High power engines employ H-beams because they can handle a lot of stress and anxiety without flexing. I-beams are powerful, light, and able to

handle tremendous pressure. The study H-beam profile exhibits reduced von mises stresses up to 12.3% (average: 15.7%) and more stability, up to 43.1%, than the I-beam profile. However, because it is less expensive and easier to make, I-beam is utilised more frequently [4]. According to a study, the structural steel connecting rod has the highest maximum stress and the Al-alloy connecting rod has the highest maximum deformation.
Finite Element methods are used to determine the system's mechanical properties. The component's meshing mode is created by FEM, and it is subsequently examined. Mesh creation in 2D is rather simple, and enhancing its quality is a straightforward process. Since fatigue failure is more likely in automobiles, aircraft, and other machinery, precise design and production are also crucial to reducing this component as much as possible [5].
Failure of the buckled load In certain research publications, the connecting rod is also analysed under conditions of sudden piston locking, which results in a larger impact load on the connecting rod and is referred to as Hydro-lock failure (cause: excess fuel air mixture or water leakage into the cylinder).
The purpose of this effort is to determine the ideal connecting rod diameters that can withstand engine conditions for a daily driver's car. This connecting rod's dimension is computed using empirical formulas for safe design that have already been established. High strength alloy structural steel is used for the connecting rod. ANSYS software is used to analyse connecting rods by applying normal stress.
The goal is achieved since the simulation's findings demonstrate that the connecting rod is safe to use in the designated engine and has a respectable safety factor that will avert any accidents.

II. METHEDOLOGY

The connecting rod's geometry was modelled using CAD software and then Imported into ANSYS for analysis. Material properties, such as Young's modulus, Poisson's ratio and density were assigned based on commonly used materials like structural steel and aluminium alloys. Boundary conditions were applied to simulate the real-life constraints and loads experienced. During engine operation a finite element analysis (FEA) was performed to investigate the stress Concentration areas, total deformation and eigen value of buckling.
Conducting a connecting rod analysis using ANSYS software involves several steps to ensure a comprehensive and accurate simulation. Here’s a step-by-step methodology:

Step 1: Define the Problem and Objectives

Objective: To analyze the stress, strain and deformation of a connecting rod under static loading conditions.
Parameters: Material properties, geometric dimensions, boundary conditions, and loading conditions.

Step 2: Create or Import Geometry

CAD Model: Create the 3D model of the connecting rod using CAD software or import an existing CAD file into ANSYS. Ensure the model is clean with no errors in geometry. Simplify the model if necessary to reduce computational load without losing critical features.

Step 3: Define Material Properties

Material Selection: Assign the material properties to the connecting rod.

Step 4: Mesh the Geometry

Meshing: Discretize the geometry into finite elements. Use an appropriate element type (e.g., tetrahedral, hexahedral). Ensure mesh quality with an appropriate element size to capture stress concentrations.
Perform a mesh convergence study to determine the optimal mesh size.

Step 5: Apply Boundary Conditions and Loads

Boundary Conditions: At one end connecting rod is fixed and at pin end axial load or pressure is applied .

Apply fixed supports where the rod is attached to other components.

Step 6: Set Up the Analysis Type

Analysis Type: Choose the type of analysis to be performed. Static Structural Analysis: For determining stress and deformation under static loading.

Step 7: Solve the Problem

Solver Settings: Configure the solver settings and initiate the solution process. Monitor convergence criteria and solver progress.

III.FORCES AFFECTING THE CONNECTING ROD

The forces operating on the connecting rod are as follows:

(i) Gas pressure exerted on the piston.

(ii) Force from the reciprocating mass and the connecting rod's inertia.

(iii) Force generated by friction between the piston and its rings.

IV.CALCULATES FORCES

Force due to gas pressure Maximum force due to gas pressure, Fa = (π x d2 x Pe) / 4 = (3.14 x 1012 x 3.15) / 4 = 24,740N

V. RESULTS AND ANALYSIS

Result of buckling analysis by simulation of ansys software we get 5 numbers of modes in the form of total deformation. These 5 numbers of modes of deformation includes various forces this are tensile and compressive force, bending force, shear force, twisting force, etc.

By applying 40 pa axial load on the pin end of the con rod we get 5 modes of deformation and load multiplier under static loading condition. It shows minimum, maximum and average deformation of the connecting rod.

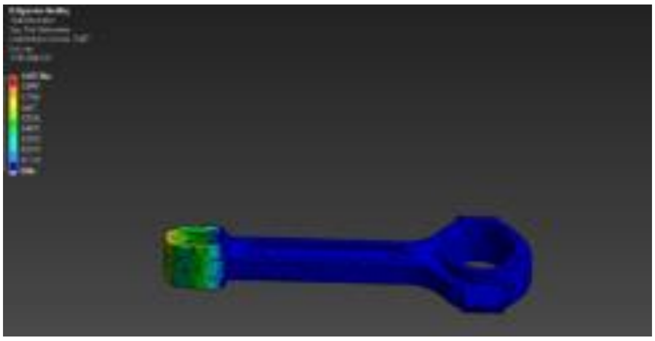

Fig. 1. Tensile force and deformation

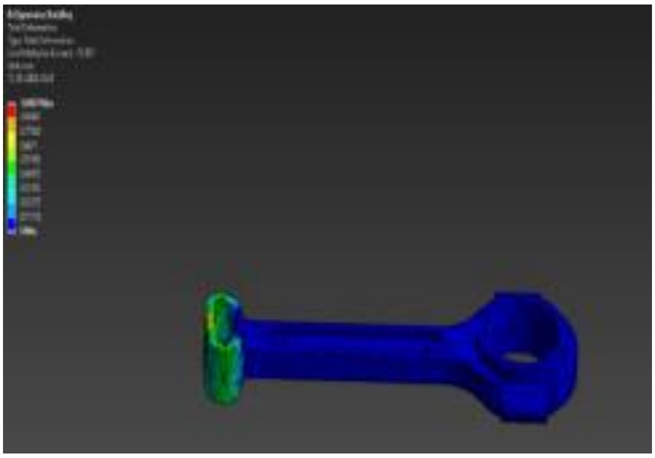

Fig. 2. Compressive force and its deformation

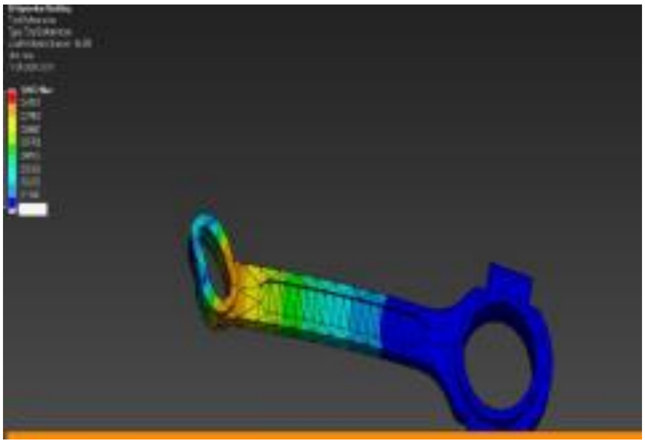

Fig. 3. Shear force and its deformation

The foundation of FEA is the domain's decomposition into a finite number of subdomains, or elements, for which the variational or weighted residual approach is used to generate the systematic approximation solutions.By breaking the domains up into pieces and describing the unknown field variable in terms of the presumptive approximation function within each element, FEA effectively simplifies the problem to one with a finite number of unknowns. These functions, which are also known as interpolation functions, are specified in terms of the field variable values at particular places, or nodes. Nodes are linked to neighbouring elements and are typically found at the edges of elements.

A Workbench programme called ANSYS Mechanical can carry out several engineering simulations, such as thermo electric, magnetostatic, thermal, vibration, and stress simulations. The model includes 20929 nodes and 11628 elements. A typical simulation consists of building up the model and the loads put to it, then solving for the model's reaction to the loads.

VI. RESULT AND DISCUSSION

From graph 6, it can be inferred that the most deformed material is aluminium 7075 T6, which is followed in order by aluminium 2024 T6, titanium, and carbon steel.Aluminium 7075 T6, titanium alloy, and carbon steel all have nearly equal von mises stresses, however aluminium 2025 T6 has the lowest von mises stresses.

Table 1 Comparison of values

Material	Max. Deform (mm)	Max. Von Mises Stress (Mpa)	Weight (Kg)	Min. Factor of Safety
Aluminium 2025-T6	0.39546	356.19	0.3759	0.9698
Aluminium 7085-T6	0.39931	357.8	0.37985	1.5000
Carbon steel :43CrMo4	0.1528	357.8	1.0366	1.5477
Ti-6Al-8v	0.26075	358.01	0.59307	2.4659

It can be observed that titanium has the greatest factor of safety, preceded by carbon steel, aluminium 7075 T6 and then aluminium 2024 T6.

VII. CONCLUSION

It is evident from the studies above that the best alloy to use for constructing connecting rods is titanium alloy. because it has the least amount of distortion and the highest Factor of Safety in relation to weight.

Despite having a lesser safety factor than titanium alloy, aluminium alloy 7075-T6 is the second choice for manufacturing since the connecting rod weighs significantly less than titanium alloy.

Additionally, it is evident that the aluminium alloy 2025-T6 is ineffective due to its low safety factor and potential for product failure. Furthermore, even though carbon steel has a higher safety factor, it is ineffective because it makes the product heavier.

VIII. REFERENCES

[1]. Balasubramaniam, B., Svoboda, M., and Bauer, W., 1991, “Structural optimization of I.C. engines subjected to mechanical and thermal loads,” Computer Methods in Applied Mechanics and Engineering, Vol. 89, pp. 337- 360.

[2]. Bhandari, V. B., 1994, “Design of Machine Elements,” Tata McGraw-Hill. Clark, J. P., Field III, F. R., and Nallicheri, N. V., 1989, “Engine state-of-the-art a competitive assessment of steel, cost estimates and performance analysis,” Research Report BR 89-1, Automotive Applications Committee, American Iron and Steel Institute.

[3]. Puran Singh, Debashis Pramanik, Ran Vijay Singh, “International Journal of Automotive Engineering and Technologies”, Vol. 4, Issue 4, pp. 245– 253,2015.

[4]. Mohammad Ranjbarkohan, Mohammad Reza Asadi and Behnam Nilforooshan Dardashti, Australian Journal of Basic and Applied Sciences, 5(12): 2084-2089, 2011.

[5]. Mohammad Reza Asadi Asad Abad, Mohammad Ranjbarkohan and Behnam Nilforooshan Dardashti Australian Journal of Basic and Applied Sciences, 5(12): 1830-1838, 2011.