Modeling & Analysis of Connecting Rod of Composite material using E-Glass, Epoxy, MWCNT & Aluminum

[1]Mr. Kamble Abhijit G., [2]Prof. Shitole J.S., [3]Prof. V. T. Rathod, [4]Prof. Valkunde B.R.

[1]Student, [2,3,4]Assistant Professor
Department of Mechanical Engineering
Dattakala Group of Institutions Faculty of Engineering, Maharashtra, India

***Abstract*: The main purpose of this article is to introduce the idea of designing connecting rods with minimal cost and the use of different materials such as E-Glass, epoxy, MWCNT, and aluminum. Here, we use these materials to improve various properties that are better than existing materials. Generally speaking, forged steel has a vast space for manufacturing connecting rods. However, the limitations of recovering forged steel with the help of these materials are very small. The overall result of the project is to provide an alternative to existing materials.**

***Keywords*: Design, Manufacturing, Composite material, Properties of material, connecting rod.**

1 INTRODUCTION

The connecting rod is a mechanical element used to transfer energy from the piston to the crank. It usually consists of two ends, namely big end and little end. The small end is used to connect the piston through the piston pin, and the big end is used to connect the camshaft. During each rotation of the crankshaft, the connecting rod is usually subjected to a large repetitive force: shear force due to the angle between the piston and the crank pivot, compression force (for example, the piston moves down), and when the piston moves The pull is upwards. These forces are proportional to the square of the engine speed (RPM). Connecting rod failure, often referred to as a "tie rod", is one of the most common causes of catastrophic engine failures in automobiles. The broken connecting rod is often passed through the side of the crankcase, causing the engine to be irreparable. Common causes of connecting rod failure are excessive engine speed leading to traction failure, impact force when the piston hits the valve (due to valve mechanism problems), connecting rod bearing failure (usually due to lubrication problems or incorrect connecting rod installation)..

2 LITERATURE REVIEW

Suraj Pal [1] "Design and Optimization Evaluation of Connecting Rod Parameters Based on Finite Element Method" This paper takes the finite element analysis of a single-cylinder four-stroke gasoline engine as the research object; using finite element technology can easily analyze the pillar structure system. First, use Cad software Pro/E Wildfire 4.0 to develop a suitable finite element model. Then, use ANSYS finite element analysis software for static analysis to determine the von Misses stress, shear stress, elastic strain, and total strain in the current design pillar. On the connecting rod, after completing the safety design work.gives 0.028 ha/hr. This design is sustainable for poor farmers.

G. Naga Malleshwara Rao [2] "Using ANSYS to design optimization and analysis of connecting rods" The main goal of this work is to explore opportunities to reduce the weight of I.C connecting rods. The engine passes inspection of various materials such as genetic steel, aluminum, titanium and cast iron. This is implied by performing a detailed load analysis.

K. Sudershan Kumar [3] "Two-wheel crank modeling and analysis" This article introduces crank modeling and analysis. In this project, the connecting rod of Suzuki GS150R motorcycle is replaced by boron carbide reinforced aluminum. Make 2D drawings based on calculations. The parametric pillar model is modeled using PRO-E 4.0 software. Use ANSYS software for analysis.

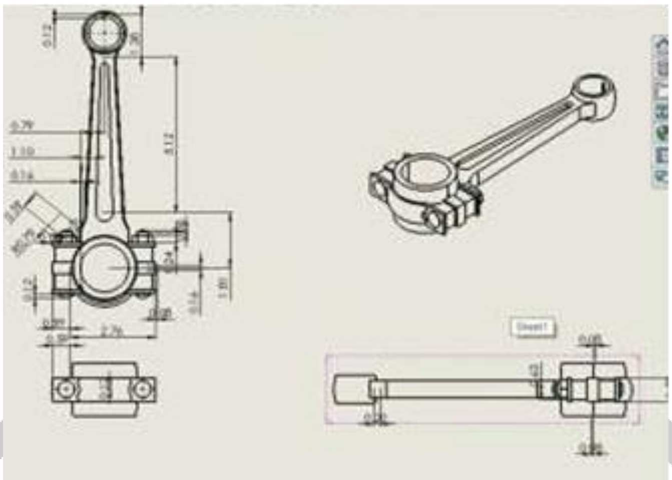
B. Anusha, C. Vijaya Bhaskar Reddy [4] "Using ANSYS to model and analyze the connecting rod of a two-wheeled vehicle" In this paper, a static analysis of the connecting rod of a 4-stroke gasoline engine and a single-vehicle connecting rod is carried out. Cylinder. The model is developed using powerful modeling software, namely PRO/E (Creoparametric). Perform additional finite element analysis to determine the von-misses stress, shear stress, and strain under the given loading conditions.

B. Anusha, Dr. C. Vijaya Bhaskar Reddy [5] "Using ANSYS to compare materials of two-wheel cranks" The model crank was imported into the analysis software, namely ANSYS. Perform static analysis to determine the transmitted stress, strain, shear stress, and total strain using the analysis software (ie ANSYS) under given loading conditions. In this analysis, two materials were selected and analyzed. The results of the two material software are compared and used to design the connecting rod.

Mr. HB Ramani [6] "Using ANSYS Software to Analyze Connecting Rods under Different Load Conditions" In this study, the load on the connecting rods was analyzed in detail, and then the finite element method was carried out in Ansys-13 medium. In different parts of the connecting rod, the total force applied to the connecting rod is calculated, then modeled, joined and loaded into ANSYS software. Determine the maximum stress of different parts of the connecting rod through analysis. The maximum pressure stress occurs between the end of the pin and the connecting rod and between the bearing cap and the connecting rod. The maximum tensile stress occurs in the lower half of the pin end and between the pin end and the connecting rod. The suggested results may help inspire changes in the connecting rod design.

Dr. NA Wankhede, Suchita Ingale [7], admitted trying to design and analyze a connecting rod used in a diesel engine in the context of the transverse bending force acting along its length during its cycle. Whiplash stress is the basis for evaluating the performance of various materials that can be used to make connecting rods. The traditional material used is steel, which was designed with CATIA V5 CAD tools, and then used ANSYS 14.5 workbench to analyze in the field of finite element analysis to detect the bending stress acting on it, and follow this procedure for different materials is 7075 aluminum , 6061 aluminum and high-strength carbon. .Mukesh K Singh & et.al are studied mechatronic concepts are incorporated in the development of a battery-assisted four-wheel weeder for weeding operation in wide row (more than 30 cm) crops.

Bansal [8] used FEA to dynamically simulate aluminum alloy connecting rods. In this analysis, the connecting rod is subjected to stress analysis and optimization under static load. Perform a static load analysis to determine the von-mises stress of the connecting rod.



3. POPOSED WORK:

In this project we are going to perform following activities,

- Existing material of connecting rod and their
- Properties
- Failure Analysis
- Suggest Alternative material for connecting rod and
- their properties
- Design of connecting rod by using analytical method
- Modeling of proposed connecting rod with suitable
- material by using Solid works
- Analysis of proposed model by using ANSYS.
- Operation was 15mm, transmission efficiency is 80%.

4. MODELING & ANALYSIS OF PROPOSED DESIGN

Solidworks is a 3D solid modeling software package that enables users to develop complete solid models for design and analysis in a simulation environment. At Solid works, we draw inspiration and try different designs to create 3D models. Designers, engineers, and other professionals use Solid works to produce simple and complex parts, assemblies, and engineering drawings.

The basis of FEA is to decompose the domain into a limited number of subdomains (elements), and construct the approximate solution of the system by applying weighted mutation or residual methods. In fact, FEA simplifies the problem to a limited number of unknowns by dividing the domain into elements and representing the unknown variables according to the approximate function assumed within each element. These functions (also called interpolation functions) are defined based on the value of the field variables at specific points (called nodes). Nodes are usually located at the cell boundary and connected to adjacent cells. ANSYS Mechanical is a workbench application that can perform various engineering simulations, including stress, heat, vibration, thermoelectric, and magnetostatic simulations. A typical simulation includes configuring the model and the load applied to it, and solving the response of the model to the load. The design has 20929 nodes and 11628 elements.

4. MATERIAL SELECTION

In the previous chapters, we have studied the characteristics and analysis of forged steel. Except for forged steel, few different alloys are used to produce connecting rods, such as alloy steel, aluminum alloy, powder metal, etc.
We will discuss and compare different materials in this chapter.

Properties	Forged Steel	Powder Metal	C-70 Alloy Steel
Young's Modulus	201	199	212
Yield Strength	700	588	574
Ultimate Tensile Strength	938	866	966
Strength Coefficient	1400	1379	1763
Strain Hardening	0.122	0.152	0.193

These are common materials used for connecting rods. But as we discussed in the previous chapters, these materials have limitations, so we will also choose different materials with similar properties and strengths. Taking into account the above characteristics, we will choose a composite material, the composition of which includes E-Glass, epoxy resin, fly ash and so on. These components can meet specifications and attributes in the required way.
The composition of the components will be determined after successful testing and composite analysis.
Several materials must be considered when using composite materials to make connecting rods. We will use E glass, epoxy resin, multi-walled carbon nanotubes and aluminium to produce connecting rods. So let's review the properties of these materials,
Material: E – Glass Fiber
Composition: 54% SiO_2 – 15% Al_2O_3 – 12% CaO

Property	Minimum Value	Maximum Value	Units (S.I.)
Atomic Volume (average)	0.0088	0.009	m^3/kmol
Density	2.55	2.6	Mg/m^3
Energy Content	100	120	MJ/kg
Bulk Modulus	43	50	GPa
Compressive Strength	4000	5000	MPa
Ductility	0.026	0.028	
Elastic Limit	2750	2875	MPa
Endurance Limit	2970	3110	MPa
Fracture Toughness	0.5	1	MPa.$m^{1/2}$
Hardness	3000	6000	MPa
Modulus of Rupture	3300	3450	MPa
Poisson's Ratio	0.21	0.23	
Shear Modulus	30	36	GPa
Tensile Strength	1950	2050	MPa
Young's Modulus	72	85	GPa
Glass Temperature	820	850	K
Maximum Service Temperature	620	630	K
Specific Heat	800	805	J/kg.K
Thermal Conductivity	1.2	1.35	W/m.K
Thermal Expansion	4.9	5.1	10^{-6}/K
Breakdown Potential	15	20	MV/m
Dielectri Constant	6.13	6.33	

Material: MWCNT (Multi Walled Carbon Nanotubes)

Properties of MWNTs

- **Electrical:** When properly integrated into a composite structure, MWNT has high conductivity; however, the outer wall itself has conductive properties, and the inner wall is not important for conductivity.
- **Morphology:** MWNT has a high aspect ratio, and its length is usually more than 100 times the diameter, and in some cases even higher. Its performance and application depend not only on its aspect ratio, but also on the degree of entanglement and straightness of the tube; this in turn is a function of the degree and size of defects in the tube.
- **Physical:** A single, defect-free MWNT has excellent tensile strength and can significantly increase its strength when integrated into composite materials such as thermoplastic or thermoset composites.
- **Thermal:** MWNT has a thermal stability above 600°C, which is due to the level of defects and to a certain degree of purity; as a residual catalyst in the product, it can also accelerate decomposition.
- **Chemical:** MWNT is an allotrope of sp2 hybrid carbon, similar to graphite and fullerene, so it has high chemical stability. However, nanotubes can be functionalized to increase the strength and dispersibility of the compound.

Material: Epoxy

- Low Shrinkage
- High strength
- Excellent adhesion to various substrates
- Effective electrical insulation
- Chemical and solvent resistance, and
- Low cost and low toxicity
- The tensile strength ranges from 90 to 120 MPa
- A tensile modulus ranging from 3100 to 3800 MPa
- Glass transition temperatures (Tg) that range from 150 to 220 ◦C

Material: Aluminum

Property	Value
Atomic Number	13
Atomic Weight (g/mol)	26.98
Crystal Structure	FCC
Melting Point (°C)	660.2
Boiling Point (°C)	2480
Mean Specific Heat (0-100°C)	0.219
Thermal Conductivity (0-100°C)	0.57
Co-Efficient of Linear Expansion (0-	23.5
Electrical Resistivity at 20°C (Ω.cm)	2.69
Density (g/cm^3)	2.6898
Modulus of Elasticity (GPa)	68.3
Poisson's Ratio	0.34

4. RESULT & DISCUSSION

The connecting rod is the main connecting rod connecting the piston and the crankshaft, and is responsible for transmitting power from the piston to the crankshaft. This project takes the finite element analysis of the connecting rod of a single-cylinder four-stroke gasoline engine as the research object. The static stress analysis will be performed on a connecting rod composed of two different materials, ie. E-glass/epoxy composite and aluminum reinforced with carbon nanotubes. The modeling and comparative analysis of the connecting rod are carried out in the commercial FEM ANSYS Workbench software. The static structural analysis will be performed by clamping the piston end and applying a load to the crankshaft end of the connecting rod. The output parameters in static stress analysis are von-Mises stress, shear stress, total strain and equivalent elastic strain under given loading conditions.
We will compare the recorded data tested with ANSYS as before. In this project, we will do a simple comparison and other related analysis to provide a detailed study of the topic.

CONCLUSION

FEA's ANSYS Workbench tool has been used to model the connecting rod of a single four-stroke cylinder. Based on this research, the following conclusions are drawn:

The maximum von-Mises stress of E-Glass / Epoxy and Al-2 CNT compounds should be greater than forged steel and powder metal.

The connecting rod made of Al-MWCNT will have a lighter weight than E-Glass / Epoxy.

This will be achieved after comparing the analysis results of two different materials. It is found that the induced stress in the Al-MWCNTs compound is less than E-Glass / Epoxy.

REFERENCES

[1] Suraj Pal, “Design Evaluation and Optimization of Connecting Rod Parameters Using FEM”, International Journal of Engineering and Management Research, Vol.-2, Issue-6, December 2012, ISSN No.: 2250-0758, Pages: 21-25.
[2] G. Naga Malleshwara Rao, “Design Optimization and Analysis of a Connecting Rod using ANSYS,” International Journal of Science and Research (IJSR), India Online ISSN: 2319-7064.
[3] K. Sudershan Kumar et al., “Modeling and Analysis of Two Wheeler Connecting Rod,” International Journal of Modern Engineering Research (IJMER), Vol.2, Issue.5, Sep-Oct. 2012, pp-3367-3371.
[4] B. Anusha, C.Vijaya Bhaskar Reddy, Modeling and Analysis of Two Wheeler Connecting Rod by Using Ansys, IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE), Volume 6, Issue 5 (May. - Jun. 2013), PP 83-87.
[5] Mr. H. B. Ramani, 2 Mr.Neeraj Kumar, 3 Mr. P. M. KasundraW. Li, D. Li and J. Ni, “Analysis of Connecting Rodunder Different Loading Condition Using Ansys Software” International Journal of Engineering Research & Technology (IJERT) Vol. 1 Issue 9, November- 2012 ISSN: 2278-0181
[6] Leela Krishna Vegi, Venu Gopal Vegi."Design And Analysis of Connecting Rod Using Forged steel “.International Journal of Scientific & Engineering Research, Volume 4, Issue 6, June-2013. 2081 ISSN 2229-5518
[7] Prof. Vivek C. Pathade1, Dr. Dilip S. Ingole, “Stress Analysis of I.C.Engine Connecting Rod by FEM and Photoelasticity” IOSR Journal of Mechanical and Civil Engineering (IOSRJMCE) e-ISSN: 2278-1684 Volume 6, Issue 1 (Mar. - Apr. 2013), PP 117-125.
[8] Kuldeep B, Arun L.R, Mohammed Faheem “Analysis and Optimization of Connecting Rod Using ALFASiC Composites”, ISSN: 2319- 875, International Journal of Innovative Research in Science, Engineering and Technology, Vol. 2, Issue 6, June 2013.
[9] Prof. N. P. Doshi, 1 Prof .N. K. Ingole “Analysis of Connecting Rod Using Analytical and Finite Element Method.” International Journal of Modern Engineering Research” (IJMER) Vol.3, Issue.1, Jan-Feb. 2013, ISSN: 2249-6645.
[10] Ram Bansal, “Dynamic simulation of connecting rod made of aluminum alloy using finite element analysis approach”, IOSR Journal of Mechanical and Civil Engineering, Volume 5, Issue 2, pp. 01-05, Jan. - Feb. 2013.