International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 9, September- 2014

Design and Comparative Analysis of Connecting Rod using Finite Element Method

Mr. Shahrukh Shamim
Material Science and Metallurgical Engineering,
Maulana Azad National Institute of Technology,
Bhopal, India

Abstract— **Connecting rod is a major link which connects the piston to the crankshaft and is responsible for transferring the power from the piston to the crankshaft. In this paper Finite element analysis of connecting rod used in single cylinder four stroke petrol engines is taken for the study. Static stress analysis is conducted on connecting rod made up of two different materials viz. E-glass/Epoxy and Aluminium composite reinforced with Carbon nanotubes. Modelling and comparative analysis of connecting rod is carried out in commercially used FEM software ANSYS 14.0. Static structural analysis was done by fixing the piston end and applying load at the crank end of the connecting rod. Output parameters in static stress analysis are von-Mises stress, Shear stress, total deformation and equivalent elastic strain for the given loading conditions.**

Keywords—Connecting Rod; Finite Element Method; Static Stress Analysis; ANSYS 14.0

I. INTRODUCTION

The connecting rod is a major link which connects the piston to the crankshaft and is responsible for transferring the piston thrust to the crankshaft so that the reciprocating motion of piston is converted in to rotary motion by crankshaft. There are distinct types of materials and production methods used in the manufacturing of connecting rods. Connecting rod undergoes high amount of compressive loading due to combustion as well as tensile loading due to inertia. High cyclic load of order 108 to 109 acts on the connecting rod [1]. The tensile and compressive stresses are produced due to gas pressure, and bending stresses are produced due to centrifugal effect & eccentricity. So, the design of the connecting rod is generally of I-section in order to provide minimum weight and maximum rigidity [2]. Therefore, durability of connecting rod is of critical importance.

Abhinav Gautam et al. [1] performed static stress analysis of connecting rod made up of SS 304 used in Cummins NTA 885 BC engine. The model is developed in CATIA V5 software and imported to ANSYS WORKBENCH 14.0 software. They presented the results of the material and reported that the area close to root of the smaller end of connecting rod is very prone to failure, may be due to higher crushing load due to gudgeon pin assembly. B. Anusha et al. [3] performed static analysis on a connecting rod of Hero Honda Splendor using ANSYS. They modelled the connecting rod in Solid modelling software i.e. PRO/E (creo-parametric) and imported it in ANSYS Workbench software. They reported from the static analysis that the stress is maximum at the piston end of the connecting rod and it is less than the material allowable limit of stress. Mohammed et al. [4] describe the cause of failure of connecting rod by finite element approach with help of MSC Patran software and also performed the metallographic study of buckled connecting rod. They reported that due to some casting defects, porosity, crack initiated and propagated on the fractured surface. Suraj Pal et al. [5] carried out Finite Element analysis of the connecting rod of a Hero Honda Splendor using FEA tool ANSYS Workbench. Model of connecting rod is developed using Cad software Pro/E Wildfire 4.0. Fatigue and Static stress analysis is done to determine the von Misses stress, shear stress, elastic strain and total deformation. They concluded that stress is highest at the piston end so the material is increased in the stressed potion to reduce stress.

Dr. S B Jaju et al. [6], performed modelling and analysis of connecting rod of four stoke single cylinder engine for optimization of cost and material. They focused their study on two subjects, first static load stress analysis of the connecting rod, and second optimization for weight. Ram Bansal et al. [7] performed the dynamic analysis of the connecting rod. In his work, model of connecting rod of single cylinder- four stroke diesel engine is modelled in CATIA V5R18 software and analysis is done on ANSYS workbench.

II. PROBLEM FORMULATION

The objective of the present work is to develop solid model of connecting rod of single cylinder-four stroke diesel engine using ANSYS Design 14.0 software. Static structural analysis is performed using FEA software ANSYS 14.0. The materials used in this study are E-glass/Epoxy and Aluminium Nano composite reinforced with Carbon nanotubes. Output parameters of static structural analysis are von Mises stress, total deformation, shear stress and equivalent elastic strain.

III. MATERIALS FOR CONNECTING ROD

The materials which are widely used in manufacturing connecting rod are carbon steel, Aluminium reinforced with Boron carbide [8]. Connecting rods for automotive applications are typically manufactured by forging from either wrought steel or powder metal. The materials used in the present study are E-glass/Epoxy and Aluminium composite reinforced with Carbon nanotubes. Compositions of these materials are listed in table 1a&b.

Table 1a: Composition of E-Glass/Epoxy [9]

Components	E-Glass	Epoxy
Composition (Vol.%)	40%	60%

Table 1b: Composition of Al-CNTs composite [10]

Components	Aluminium	Carbon Nanotubes
Composition (Vol.%)	Al (purity 99%, 200 mesh), 98%	MWCNTs, 2 Vol.%

Material's mechanical Properties are illustrated in table 2a&b.

Table 2: Selected Mechanical properties of Al-MWCNT composite used in this study [10]

Parameters	Al-MWCNTs composite
Density	2.35 g/cc
Young's Modulus, E	89 GPa
Poisson's Ratio	0.33
Ultimate Tensile Strength	435 MPa
Behaviour	Isotropic

Table 2b: Mechanical Properties of E-Glass/Epoxy [11]

Tensile modulus along X direction	34000 Mpa
Tensile modulus along Y direction	6530 Mpa
Tensile modulus along Z direction	6530 Mpa `
Tensile strength	900 Mpa
Compressive strength	450 Mpa
Shear modulus along XY direction	2433 Mpa
Shear modulus along YZ direction	1698 Mpa
Shear modulus along ZX direction	2433 Mpa
Poisons ratio along XY direction	0.217
Poisons ratio along YZ direction	0.366
Poisons ratio along ZX direction	0.217
Density of the material	2.6 g/cc
Behaviour	Orthotropic

IV. FINITE ELEMENT ANALYSIS OF CONNECTING ROD

A. *Geometrical Parameters and Modelling*

Connecting rod of four stroke single cylinder engine is selected for the present investigation. According to the dimensions the model of the connecting rod is developed by using ANSYS Design 14.0. The modelled connecting rod is shown in fig 1. Various geometric parameters are listed in table 3.

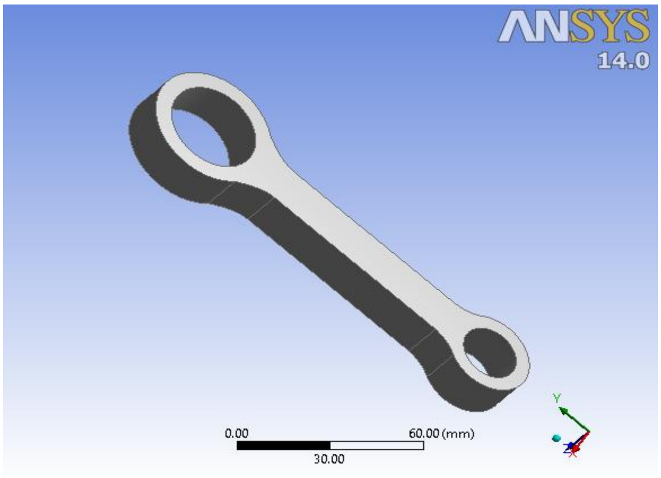


Fig. 1: Model of connecting rod

Table 3: Geometrical parameters of Connecting rod

Parameters	Values
Length of connecting rod	120 mm
Outer Diameter of Crank end	44 mm
Inner Diameter of Crank end	32 mm
Outer Diameter of Piston end	30.1 mm
Inner Diameter of Piston end	22 mm
Width of connecting rod	15.6 mm
Depth of connecting rod	16 mm

B. *Meshing*

Mesh Generation is one of the most critical aspects of engineering simulations. Meshing involves division of the entire model into small pieces called elements. Finite element mesh is generated using Tetrahedron element (TET10). In present case, number and type of elements taken is shown in table 3. Meshed Surface is shown in Fig. 2

Table 3: Total number of Nodes and Elements

Entity	Size
Number of Nodes	35528
Number of Elements	21724

C. *Load Distribution on Connecting Rod*

For static structural analysis, load is applied at the crank end and fixed support is given at the piston end. The analysis is carried out under axial loads. Here the axial load applied is 9500 N. The comparisons are done for optimization purpose.

ISSN: 2278-0181
Vol. 3 Issue 9, September- 2014

V. RESULTS AND DISCUSSIONS

A. Static Structural Analysis of Connecting Rod

Static structural analysis of connecting rod under tensile loading is carried out using ANSYS 14.0 software.

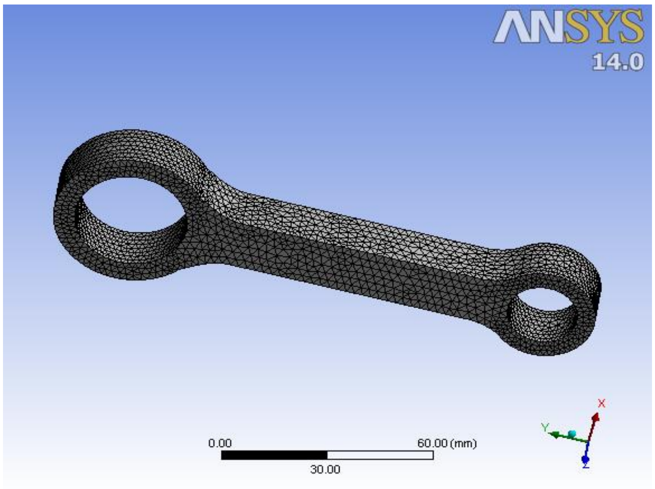
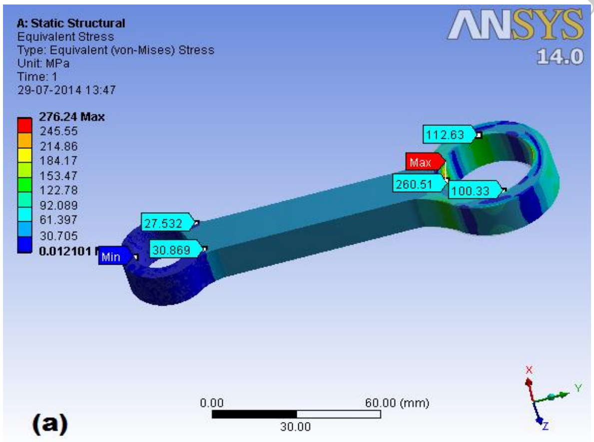
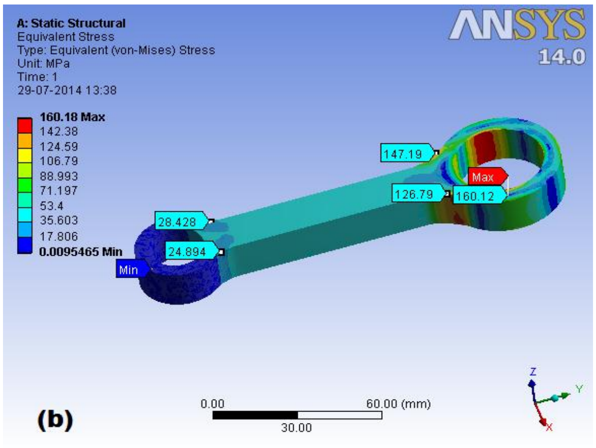
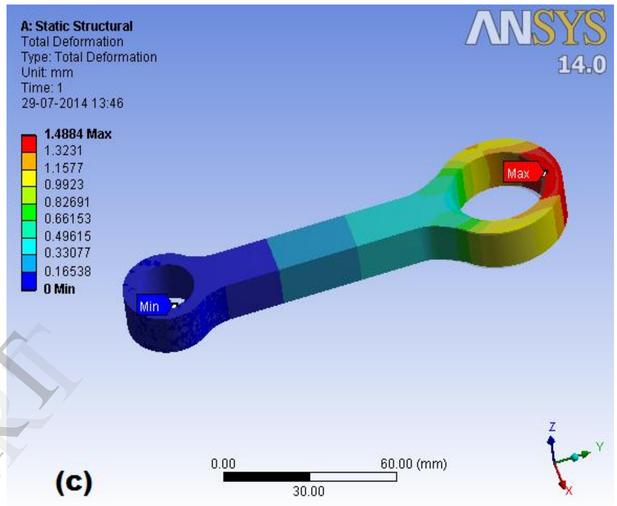


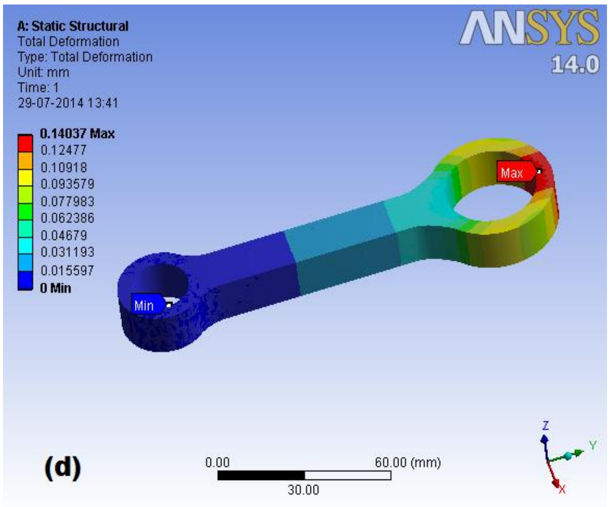
Fig 2: Meshed model of connecting rod

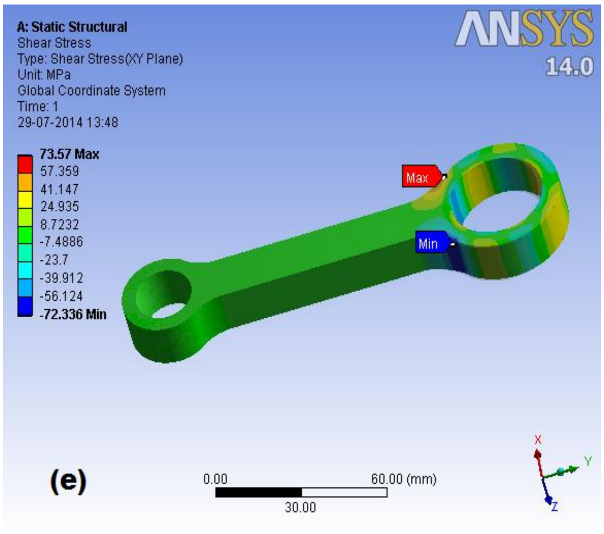
Maximum von Mises stress, total deformation, shear stress and equivalent elastic strain of E-Glass/Epoxy and Al-CNTs composite are the output parameters of this analysis and are shown in figure 3a-h.

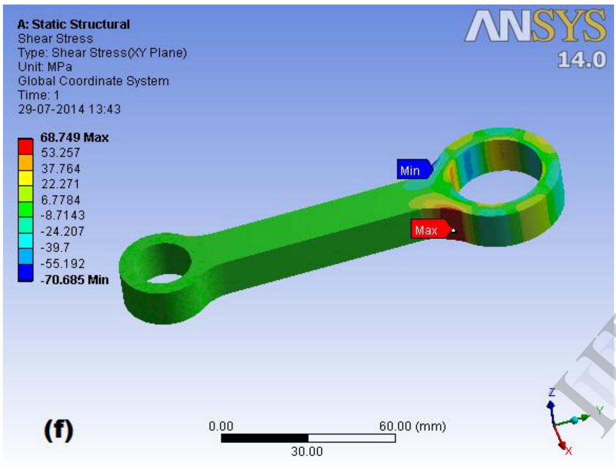


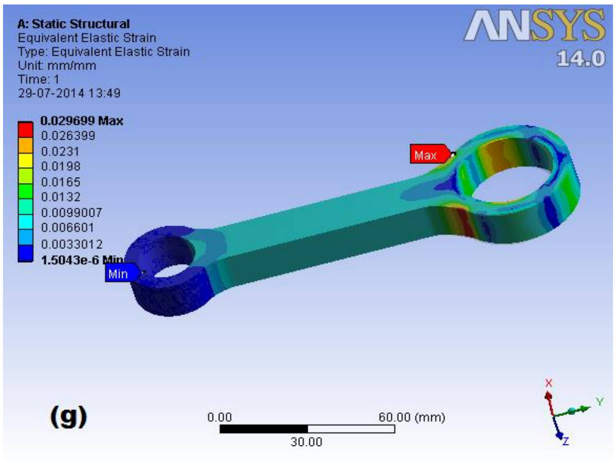












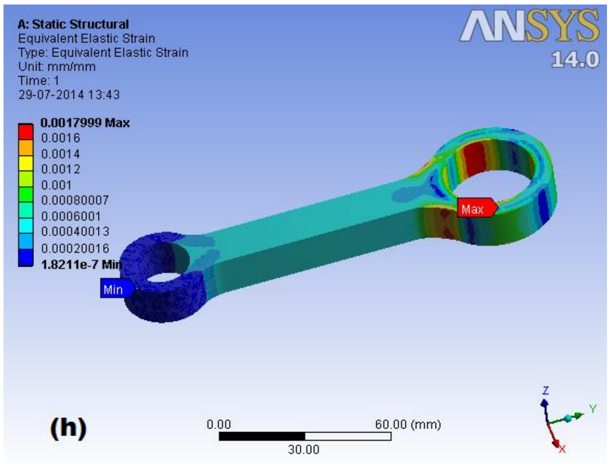


Fig.: 3(a&b) von-Mises stress in E-Glass/Epoxy and in Al-CNTs composite respectively, 3(c&d) Total deformations in E-Glass/Epoxy and in Al-CNTs composite, 3(e&f) Shear stress in E-Glass/Epoxy and AL-CNTs composite and 3(g&h) Equivalent elastic strain in E-Glass/Epoxy and in Al-CNTs composite

The FEA results for static analysis i.e. von-Mises stress, Total Deformation, shear stress and Elastic Strain are shown in Figure 3 (a-h) respectively. From the above results, comparison for the two materials is done. Table 4 shows the results of static structural analysis.

Table 4: Results of static structural analysis

Parameters	FEA results of E-Glass/Epoxy	FEA results of Al- 2 vol.%CNTs composite	Variation
Equivalent von-mises stress	276.24 MPa	160.18 MPa	42.0%
Total deformation	1.48 mm	0.140 mm	90.5%
Shear stress	73.57 MPa	68.74 MPa	6.5%
Elastic strain	$2.96*10^{-2}$ mm/mm	$1.799*10^{-3}$ mm/mm	93.9%

Weight comparison of E-Glass/Epoxy and Al-2 vol% CNTs composite used as connecting rod’s material is shown in table 5.

Table 5: Weight comparison

Material	Weight
E-Glass/Epoxy	0.102 Kg
Al- 2 vol.% CNTs composite	0.0929 Kg

CONCLUSION

Solid modelling of connecting rod for four stroke single cylinder has been done using FEA tool ANSYS 14 workbench. On the basis of this study following conclusion has been made:

- Maximum von-Mises stress for E-Glass/Epoxy and Al-2 vol.% CNTs composite is 276.24 MPa and 160.18 Mpa respectively.
- Connecting rod made from Al- 2 vol.% CNTs has less weight than that of E-Glass/Epoxy. Comparing the results obtained from the analysis for two different materials it has been found that the stress induced in the Al- 2 vol.%CNTs composite is less than the E-Glass/Epoxy

REFERENCES

[1] AbhinavGautam, K Priya Ajit, Static Stress Analysis of Connecting Rod Using Finite Element Approach, IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE), Volume 10, Issue 1 (Nov. - Dec. 2013), PP 47-51.

[2] P. G. Charkha, S. B. Jaju (2009), "Analysis & Optimization of Connecting Rod" Second International Conference on Emerging Trends in Engineering and Technology, ICETET-09.

[3] 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.

[4] Mohammed et al. 'Failure Analysis of a Fractured Connecting Rod'. Journal of Asian scientific research, 2012, vol. 2 issue 11, pp737-741.

[5] Suraj Pal et al., "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.

[6] Dr S B jaju, P G Chakra, "Modeling and Optimization of Connecting rod of Four Stroke Single Cylinder Engine for Optimization of cost and material".

[7] Ram Bansal, "Dynamic Simulation of a Connecting Rod Made of Aluminium Alloy Using Finite Element Analysis Approach Method" IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE), Jan. - Feb. 2013 Vol 5, Issue 2, pp01-05.

[8] K. Sudershn 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.

[9] Suhas et al., "Investigation on different Compositions of E-Glass/Epoxy Composite and its application in Leaf Spring", IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE), Volume 11, Issue 1 Ver. V (Feb. 2014), PP 74-80.

[10] Lin Jiang, Zhiqiang Li, Genlian Fan, Linlin Cao, Di Zhang, The use of flake powder metallurgy to produce carbon nanotube (CNT)/aluminium composites with a homogenous CNT distribution, Carbon 50 (2012) 1993–1998.

[11] U. S. RAMAKANTH et al, "DESIGN AND ANALYSIS OF AUTOMOTIVE MULTI-LEAF SPRINGS USING COMPOSITE MATERIALS", International Journal of Mechanical Production Engineering Research and Development (IJMPERD) ISSN 2249-6890 Vol. 3, Issue 1, Mar 2013, 155-162