International Journal of Research
Available at https://journals.pen2print.org/index.php/ijr/
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 06 Issue 10
September 2019

Modeling and Structural Analysis of IC Engine Connecting Rod with Composite Material

R Charan Kumar[1], Shaik Abzal Basha[2], A Vidya Sagar[3]
[1]P.G. Scholar, [2]Guide, Assistant Professor,[3]Head of the Department
[1,2,3] BRANCH : *MACHINE DESIGN*
[1,2,3] Geethanjali College Of Engineering & Technology
Email : [1]rcharankumar92@gmail.com, [2]shaikabzalbasha777@gmail.com

Abstract

The automobile engine connecting rod is a high volume production, critical component. It connects reciprocating piston to rotating crankshaft, transmitting the thrust(power) of the piston to the crankshaft. Every vehicle that uses an internal combustion engine requires at least one connecting rod depending upon the number of cylinders in the engine.

Connecting rods for automotive applications are typically manufactured by forging from either wrought steel or powdered metal. They could also be cast. Here unwanted stresses found on crank and piston ends of connecting rod while transmitting power from piston to crankshaft.

The main objective of this study was to explore weight and cost reduction opportunities for a production forged wrought steel and Composite HM Carbon/Epoxy connecting rods. This has entailed performing a detailed load analysis. The most important factors that are concentrated are stress distribution and deflections. Connecting rod is designed by Unigraphics software and Ansys software is used for analysis of connecting rod.

INTRODUCTION

The automobile engine connecting rod is a high volume production, critical component. It connects reciprocating piston to rotating crankshaft, transmitting the thrust of the piston to the crankshaft. Every vehicle that uses an internal combustion engine requires at least one connecting rod depending upon the number of cylinders in the engine.

Connecting rods for automotive applications are typically manufactured by forging from either wrought steel or powdered metal. They could also be cast. However, castings could have blow- holes which are detrimental from durability and fatigue points of view.

Connecting Rod Evaluation

Since 1986, more than 500 million powders forged (P/F) connecting rods have been manufactured and installed in automobiles worldwide. The application of forging a preformed near-net shape consolidated from metal powder has been widely accepted since the early eighties and today is the preferred manufacturing technique for 60% of the connecting rods manufactured in North America. The remaining portion of the connecting rod market segment is produced by use of either conventional steel forging, or to a lesser extent, casting manufacturing processes.

The powder forged (P/F) rod is: fabricated by consolidating metal powders into a pre form that is sintered, reheated to forging temperature (or in some cases forged subsequently to sintering), fully densified by forging to final shape, fracturing of the rod cap end, and then machined (minimally) to final dimensions.

While the two competing forging processes are similar, there are a number of subtle differences between the two. The forged steel rod is fabricated by starting with a wrought steel billet, heating the billet and forging it in the material's plastic temperature range, fracturing or cutting the rod cap end, and then machining portions of the product to realize the final dimensional characteristics of the component.

All connecting rods function in internal combustion engine environments and are subjected to high rate cyclic loading requiring exacting tolerances and fits to mating components, such as the crankshaft and piston head.

Until the advent of the crackable P/F connecting rod cap end, all connecting rod cap ends were sawn or machined apart to enable inclusion of a bearing and attachment to the crankshaft.

The cost of sawing and machining the cap end to meet tolerances in finish and fit were a considerable portion of the manufacturing cost and sawing required that the internal diameter of the rod cap end account for the "kerf" of the sawn area in the shape of the "hole" (it was formed slightly out-of-round). A "crackable" rod cap end provides the advantages of lower cost to separate the cap end, the surfaces of the cracked ends mate better and more accurately when reassembled after machining and the tolerances of the cap end internal diameter can be closely held to a perfectly circumferential circle. From the conventional method of splitting connecting rods with twenty steps, the fracture splitting method reduced the process to ten steps, all essentially machining, honing or drilling steps.

Connecting rod manufacture is a high volume, price sensitive application with strict performance, design and durability requirements. Process or material improvements leading to lower costs result in large scale cost savings. Annual North American production is approximately 100 million rods.

The C-70 Story

A new steel, C-70, has been introduced from Europe as a crackable forging steel. Alloying elements in the material enable hardening of forged connecting rods when they undergo controlled cooling after forging. This material fractures in a fashion similar to powder forged materials.

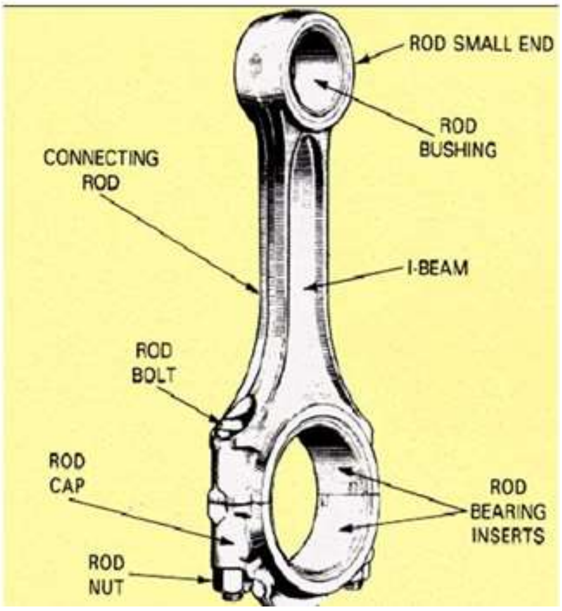
Recently the American Iron and Steel Institute's (AISI) Bar and Rod Market Development Group has promoted C-70 as an improved material over P/F alloys on the basis of optimization work and economic analysis performed by a candidate for a Master's of Science Degree in Mechanical Engineering at the University of Toledo. The thesis advisor was Dr. Ali Fatemi. The study investigated weight and cost reduction opportunities of steel forged connecting rods. Analysis focused on comparing and then optimizing a rod design using crackable forged steel (C- 70). Using finite element analysis (FEA) techniques, the author was able to reduce the weight by 10% and by using "crackable" C-70, reduce the costs by 25% (over current forged steel connecting rods) and ostensibly 15% less than a P/F rod with similar or better fatigue behavior. The study identified fatigue strength as the most significant design factor in the optimization process. The AISI funded study focused on using FEA analysis to show where and how the original connecting rod design configuration could be reconfigured to reduce weight and by using the "crackable" C- 70, to eliminate some cost considerations.

The objective of my project is to create a 3d model of connecting rod and perform finite element analysis of the connecting rod and observe high stressed locations.

Modification on the connecting rod is done to reduce the stresses and finite element analysis shall be carried on the modified model.

GEOMETRY OF THE CONNECTING ROD

A solid model of the connecting rod, as shown in Figure was generated using UNIGRAPHICS .



Literature Review

Leela Krishna Vegi1 , Venu Gopal Vegi2: Published on "Design And Analysis of Connecting Rod Using Forged steel". The connecting rod is the intermediate member between the piston and the Crankshaft. Its primary function is to transmit the push and pull from the piston pin to the crank pin, thus converting the reciprocating motion of the piston into rotary motion of the crank. This thesis describes designing and Analysis of connecting rod. Currently existing connecting rod is manufactured by using Carbon steel. In this drawing is drafted from the calculations.

Mohamed Abdusalam Hussin1 , Er. Prabhat Kumar Sinha2 , Dr. Arvind Saran Darbari3: DESIGN AND ANALYSIS OF CONNECTING ROD USING ALUMINIUM ALLOY 7068 T6, T6511. In a reciprocating piston engine, the connecting rod connects the piston to the crank or crankshaft. In modern automotive internal combustion engines, the connecting rods are most usually made of steel for production engines, but can be made of aluminum (for lightness and the ability to absorb high impact at the expense of durability) or titanium (for a combination of strength and lightness at the expense of affordability) for high performance engines, or of cast iron for applications such as motor scooters. The small end attaches to the piston pin, gudgeon pin (the usual British term) or wrist pin, which is currently most often press fit into the con rod but can swivel in the piston, a "floating wrist pin" design. The connecting rod is under tremendous stress from the reciprocating load represented by the piston, actually stretching and being compressed with every rotation, and the load increases to the third power with increasing engine speed. Failure of a connecting rod, usually called "throwing a rod" is one of the most common causes of catastrophic engine failure in cars, frequently putting the broken rod through the side of the crankcase and thereby rendering the engine irreparable; it can result from fatigue near a physical defect in the rod, lubrication failure in a bearing due to faulty maintenance or from failure of the rod bolts from a defect, improper tightening, or re-use of already used (stressed) bolts where not recommended. Despite their frequent occurrence on televised competitive automobile events, such failures are quite rare on production cars during normal daily driving. This is because production auto parts have a much larger factor of safety, and often more systematic quality control.

Dr. N. A. Wankhade1 , Suchita Ingale2: They had written on "Review on Design and Analysis of Connecting Rod Using Different Material". Connecting Rods are used practically generally used in all varieties of automobile engines acting as an intermediate link between the piston and the crankshaft of an engine of an automobile. It is responsible for transmission the up and down motion of the piston to the crankshaft of the engine, by converting the reciprocating motion of the piston to the rotary motion of crankshaft. While the one end, small end the connecting rod is connecting to the piston of the engine by the means of piston pin, the other end, the bigger end being connected to the crankshaft with lower end big end bearing by generally two bolts. Generally connecting rods are being made up of stainless steel and aluminium alloy through the forging process, as this method provides high productivity and that too with a lower production cost. Forces generated on the connected rod are generally by weight and combustion of fuel inside cylinder acts upon piston and then on the connecting rod, which results in both the bending and axial stresses. The present paper attempts to design and analyze the connecting rod used in a diesel engine in context of the lateral bending forces acting along its length during cycle of it The lateral bending stress are commonly called as whipping stress and this whipping stress forms the base of evaluation of performance of various materials that can be used for manufacturing of connecting rod. The conventional material used is steel which is designe using CAD tool which is CATIA V5 and subsequently analysed for bending stress acting on it in the arena of finite element analysis using ANSYS workbench 14.5 and this procedure is followed for different material which are aluminium 7075 , aluminium 6061 and High Strength Carbon.

Leela Krishna Vegi1 , Venu Gopal Vegi2: They written a paper on "Design And Analysis of Connecting Rod Using Forged steel". The connecting rod is the intermediate member between the piston and the Crankshaft. Its primary function is to transmit the push and pull from the piston pin to the crank pin, thus converting the reciprocating motion of the piston into rotary motion of the crank. This thesis describes designing and Analysis of connecting rod. Currently existing connecting rod is manufactured by using Carbon steel. In this drawing is drafted from the calculations. A parametric model of Connecting rod is modeled using CATIA V5 R19 software and to that model, analysis is carried out by using ANSYS 13.0 Software. Finite element analysis of connecting rod is done by considering the materials, viz... Forged steel.

The best combination of parameters like Von misses Stress and strain, Deformation, Factor of safety and weight reduction for two wheeler piston were done in ANSYS software. Forged steel has more factor of safety, reduce the weight, increase the stiffness and reduce the stress and stiffer than other material like carbon steel. With Fatigue analysis we can determine the lifetime of the connecting rod. Prateek Joshi1 , Mohammad UmairZaki2: Have published a journal on "FEM Analysis of Connecting Rod of different materials using ANSYS". Connecting Rods are practically generally used in all varieties of automobile engines. Acting as an intermediate link between the piston and the crankshaft of an engine. It is responsible for transmission of the up and down motion of the piston to the crankshaft of the engine, by converting the reciprocating motion of the piston to the rotary motion of crankshaft. Thus, this study aims to carry out for the load, strain and stress analysis of the crank end of the connecting rod of different materials. Based on which the High Strength Carbon Fiber connecting rod will be compared with connecting rod made up of Stainless Steel and Aluminum Alloy. The results can be used for optimization for weight reduction and for design modification of the connecting rod. Pro-E software is used for modeling and analyses are carried out in ANSYS software. The results archived can also help us identify the spot or section where chances of failure are high due to stress induced. Also the results obtained can be used to modify the existing designs so that better performance and longer life cycle can be archived.

PROBLEM DEFINITION & METHODOLOGY

Connecting rods for automotive applications are typically manufactured by forging from either wrought steel or Composite metal. The problem occurs in connect rod is going to fail under loading condition (power production) in Engine if proper material is not choose for manufacturing.

In this study NX-CAD software shall be used for 3Dmodeling of connecting rod, ANSYS software shall be used for the finite element analysis of connecting rod.

The methodology followed in my project is as follows:

- Create a 3D model of the connecting rod is developed by using NX-CAD software.

International Journal of Research
Available at https://journals.pen2print.org/index.php/ijr/
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 06 Issue 10
September 2019

- Perform structural analysis on connecting rod using ANSYS software and obtain the deflections and von misses stresses values produced.
- Structural analysis was performed by using steel, HM Carbon/Epoxy and E-Glass/Epoxy materials.
- Obtained results were tabulated and best material which had less stress during loading condition is proposed for manufacturing of connecting rod.

INTRODUCTION TO UNIGRAPHICS

Overview of Solid Modeling

The Unigraphics NX Modeling application provides a solid modeling system to enable rapid conceptual design. Engineers can incorporate their requirements and design restrictions by defining mathematical relationships between different parts of the design.

Design engineers can quickly perform conceptual and detailed designs using the Modeling feature and constraint based solid modeler. They can create and edit complex, realistic, solid models interactively, and with far less effort than more traditional wire frame and solid based systems. Feature Based solid modeling and editing capabilities allow designers to change and update solid bodies by directly editing the dimensions of a solid feature and/or by using other geometric editing and construction techniques.

Advantages of Solid Modeling

Solid Modeling raises the level of expression so that designs can be defined in terms of engineering features, rather than lower-level CAD geometry. Features are parametrically defined for dimension-driven editing based on size and position

Overview of Solid Modeling

The Unigraphics NX Modeling application provides a solid modeling system to enable rapid conceptual design. Engineers can incorporate their requirements and design restrictions by defining mathematical relationships between different parts of the design.

Design engineers can quickly perform conceptual and detailed designs using the Modeling feature and constraint based solid modeler. They can create and edit complex, realistic, solid models interactively, and with far less effort than more traditional wire frame and solid based systems. Feature Based solid modeling and editing capabilities allow designers to change and update solid bodies by directly editing the dimensions of a solid feature and/or by using other geometric editing and construction techniques.

Advantages of Solid Modeling

Solid Modeling raises the level of expression so that designs can be defined in terms of engineering features, rather than lower-level CAD geometry. Features are parametrically defined for dimension-driven editing based on size and position.

Features

Powerful built-in engineering-oriented form features-slots, holes, pads, bosses, pockets-capture design intent and increase productivity

Patterns of feature instances-rectangular and circular arrays-with displacement of individual features; all features in the pattern are associated with the master feature

Blending and Chamfering

- zero radius
- Ability to chamfer any edge
- Cliff-edge blends for designs that cannot accommodate complete blend radius but still require blends

Advanced Modeling Operations

- Profiles can be swept, extruded or revolved to form solids
- Extremely powerful hollow body command turns solids into thin-walled designs in seconds; inner wall topology will differ from the outer wall, if necessary
- Fixed and variable radius blends may overlap surrounding faces and extend to a

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 06 Issue 10
September 2019

Tapering for modeling manufactured near-net shape parts

- User-defined features for common design elements (Unigraphics NX/User-Defined Features is required to define them in advance

PROCEDURE OF 3D MODELING OF CONNECTING ROD

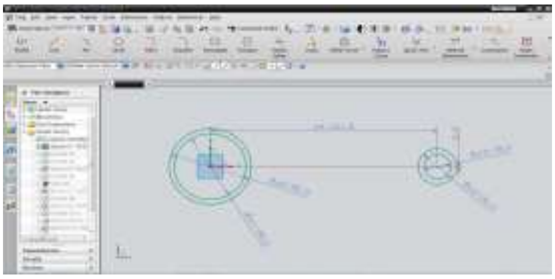


Fig.4.1 2d sketch of connecting rod

Fig 4.2 Extrude of connecting rod

Fig 4.3 Extrude of connecting rod

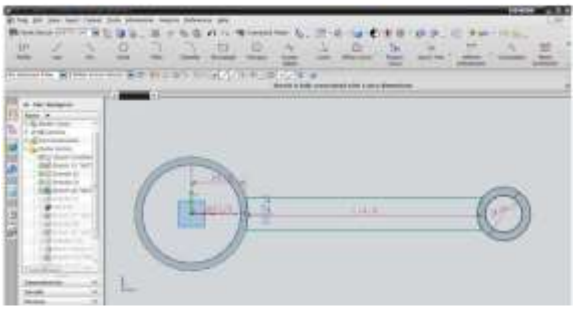


Fig 4.4 2d sketch of connecting rod

Fig 4.5: Extrude of connecting rod

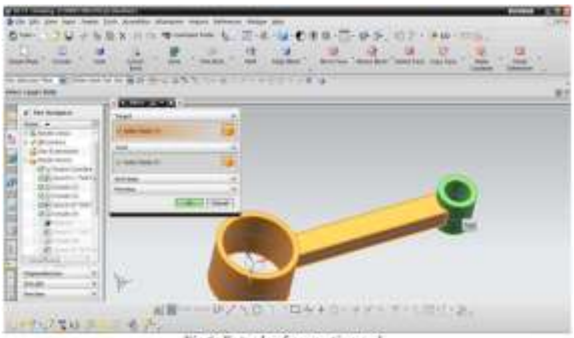


Fig.6: Extrude of connecting rod

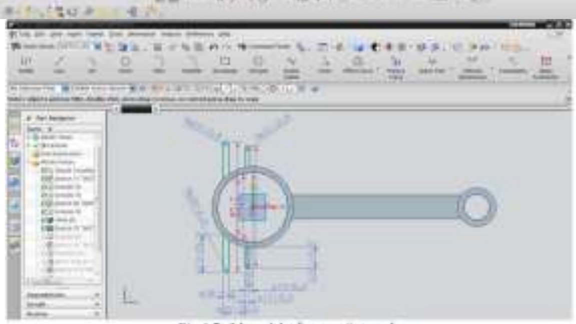


Fig 4.7: 2d model of connecting rod

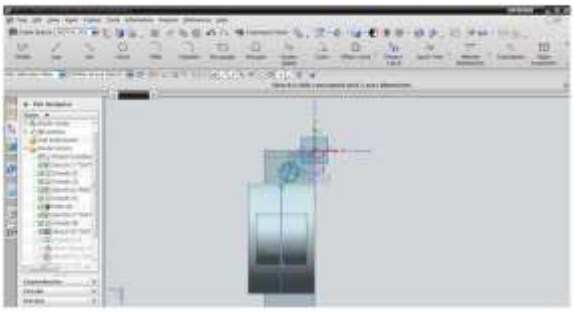


Fig 4.8: 2d model of connecting rod

International Journal of Research
Available at https://journals.pen2print.org/index.php/ijr/
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 06 Issue 10
September 2019

Fig4.9: Extrude of connecting rod

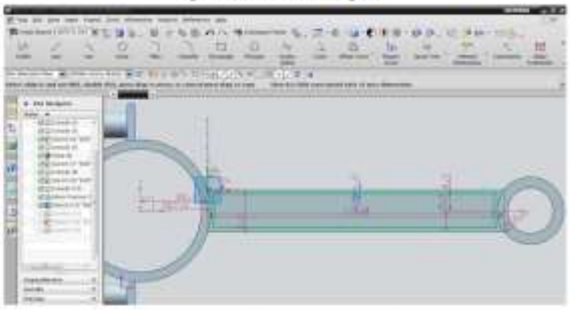


Fig 4.10 2d model of connecting rod

Fig4.11: Extrude of connecting rod

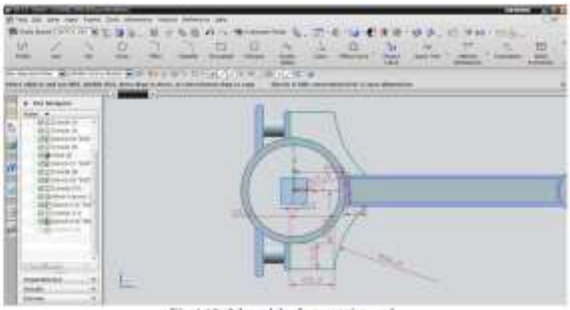


Fig 4.12: 2d model of connecting rod

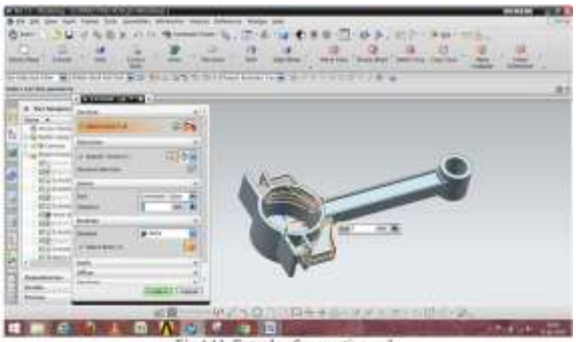


Fig 4.13: Extrude of connecting rod

Fig 4.14: final Design of connecting rod

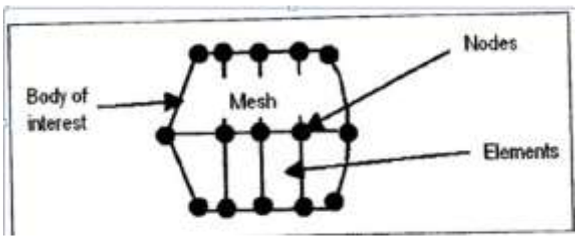
ANALYSIS METHOD

FINITE ELEMENT ANALYSIS (FEA)

Finite element analysis was first developed in the forties for use in structural analysis and it is numerical technique for solution of boundary – value problems. The accuracy of the FEA method can be improved by refining the mesh in the model using more elements and nodes.

The finite elements method is an apparently complex technique, but still the fundamental principles are relatively straight forward. As a typical example, consider a body in which the distribution of an unknown variable such as temperature or displacement is required.

The first step is dividing the actual geometry of the structure using a collection of discrete portions called finite elements which are joined together by shared nodes. The collection of nodes and finite elements is known as the mesh as shown in figure 4.2.



General, traditional hand calculations involve many conservative approximations. Beside, FEA

International Journal of Research
Available at https://journals.pen2print.org/index.php/ijr/
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 06 Issue 10
September 2019

solid model sometimes used to verify the validity of assumptions in the calculations, such as:

- Verify assumptions used in hand calculations
- Determine local stresses in a connection or part
- Provide more accurate stress or deflection information

Axial compressive load at the crank end

In this case axial compressive load of 26.7KN is applied at crank end

Fig5.2: Boundary conditions of the connecting rod for case-1 condition

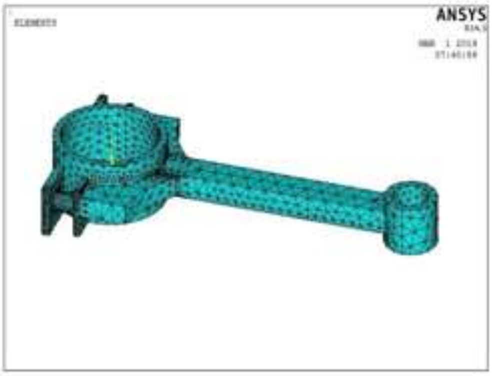
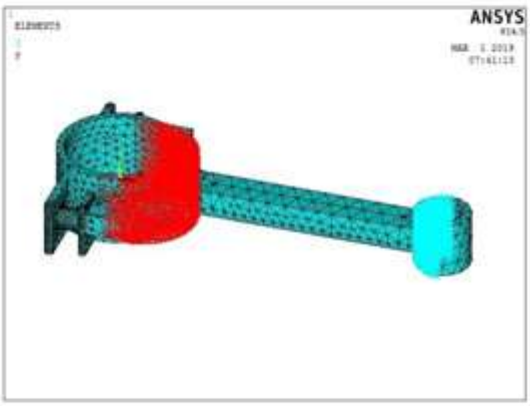


Fig5.3: Applied load condition of connecting rod

Since ansys does not have the option to apply parabolic load we have written a macro. To apply the parabolic load we have to read the macro.



Results:

Total Deflection and Deflection in X-dir

Fig5.4: Total deformation of connecting rod

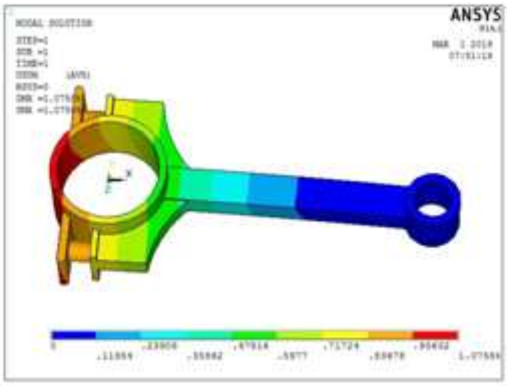
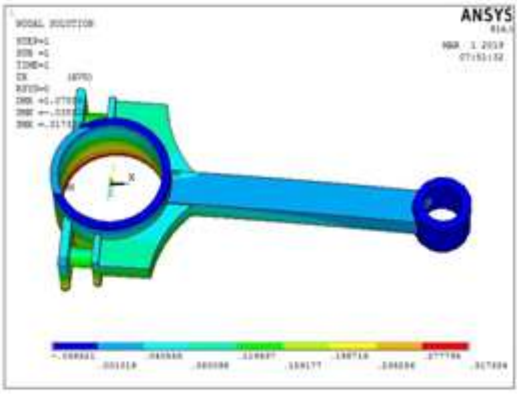


Fig5.5: Deformation in x direction of connecting rod



Deflection in Y-dir and Z-dir

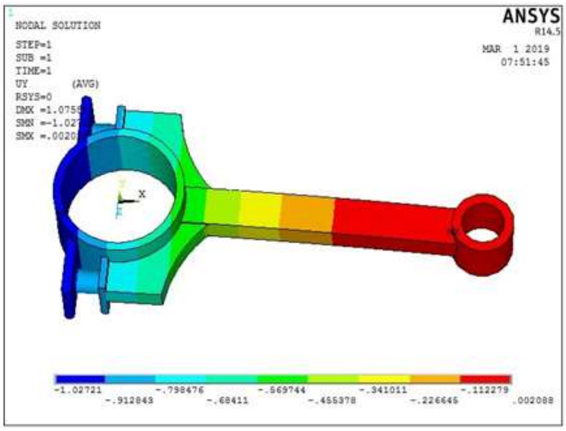


Fig5.6 Deformation in y direction of connecting rod

International Journal of Research
Available at https://journals.pen2print.org/index.php/ijr/
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 06 Issue 10
September 2019

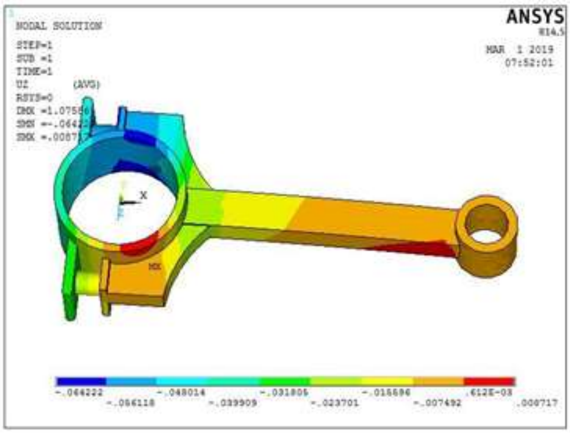


Fig5.7: Deformation in z direction of connecting rod

From the above results maximum total deflection observed is 1.07mm

VonMises Stress

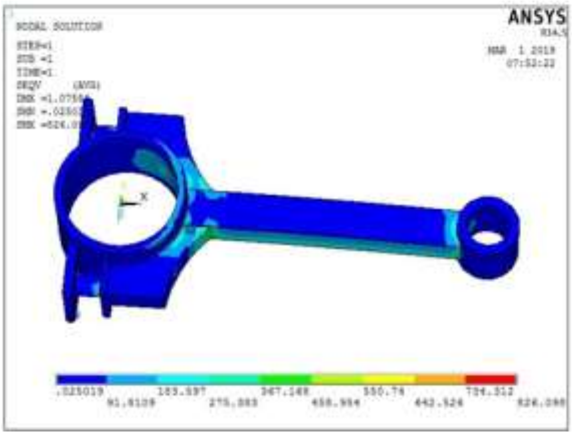


Fig5.8: VonMises stresses of connecting rod

From the results the maximum VonMises stress observed is 826Mpa

STRUCTURAL ANALYSIS PROCESS OF CONNECTING ROD USING HM CARBON/EPOXY COMPOSITE MATERIAL

Structural static analysis is performed on the connecting rod to identify the high stress regions and attempts are made to modify the connecting rod to reduce the stresses.

The crank and piston pin ends are assumed to have a sinusoidal distributed loading over the contact surface area, under tensile loading. This is based on experimental results.

In this study four finite element models were analyzed. FEA for both tensile and compressive loads were conducted. Two cases were analyzed for each case, one with load applied at the crank end and restrained at the piston pin end, and the other with load applied at the piston pin end and restrained at the crank end. In the analysis carried out, the axial load was 26.7 KN (6 kips) in both tension and compression.

The 3d model generated in UNIGRAPHICS is converted into parasolid. The parasolid file is imported into ansys and finite element analysis is carried out in Ansys.

Material properties

Material used for connecting rod is composite materials (hm carbon/epoxy):

Longitudinal Modulus (EZ): 190 GPa Transverse Modulus (EY): 7.7 GPa Shear modulus (Gxy): 4.2 GPa

Shear modulus (Gyz): 4.2 GPa Shear modulus (Gxz): 4.2 GPa Poisson's Ratio: 0.3

Density: 1600 Kg/m3

Yield strength: 800 Mpa

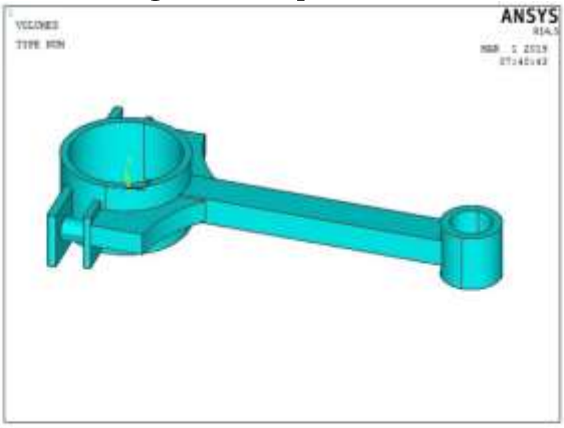


Fig 5.9: 3 3d model of the connecting rod imported into Ansys

Axial compressive load at the crank end

In this case axial compressive load of 26.7KN is applied at crank end with cosine distribution over 120° and piston pin end restrained over 120°

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 06 Issue 10
September 2019

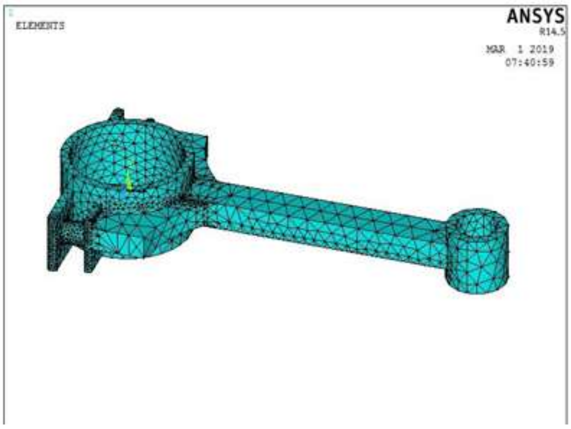


Fig5.10: Boundary conditions of the connecting rod for case-1 condition

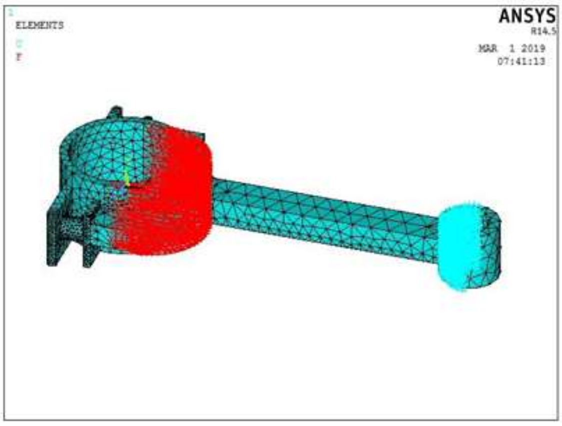


Fig5.11: Applied load condition of connecting rod

Since ansys does not have the option to apply parabolic load we have written a macro. To apply the parabolic load we have to read the macro.

Results:

Total Deflection and Deflection in X-dir

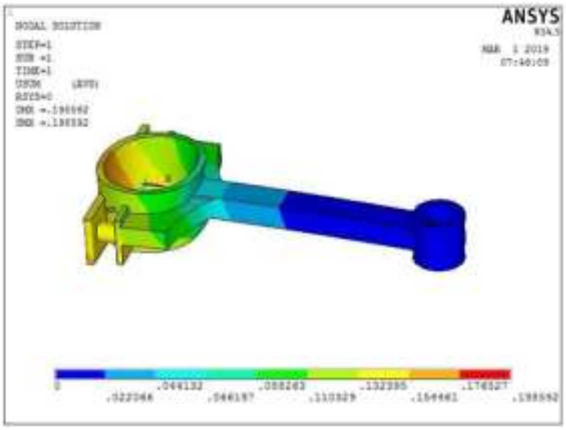


Fig5.12: Total deformation of connecting rod

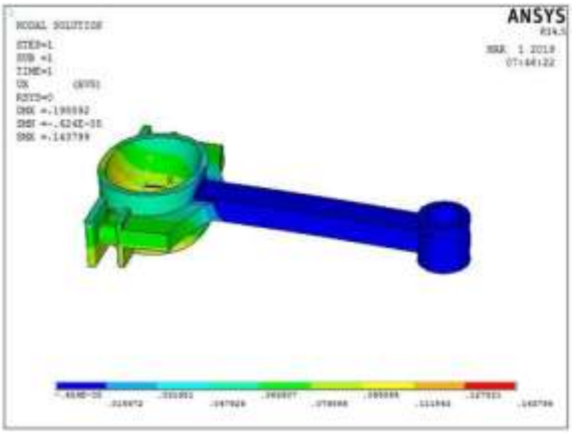


Fig5.13: Deformation in x direction of connecting rod

Deflection in Y-dir and Z-dir

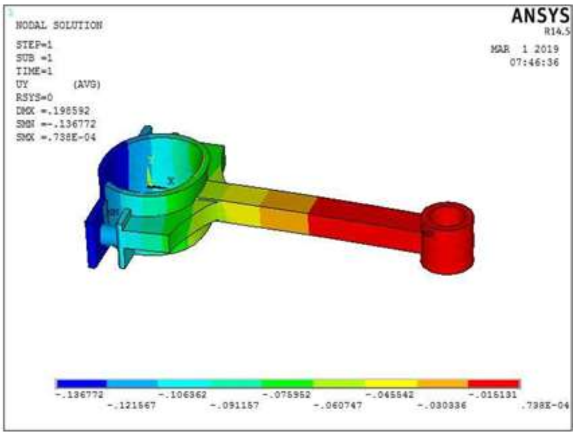


Fig5.14 Deformation in y direction of connecting rod

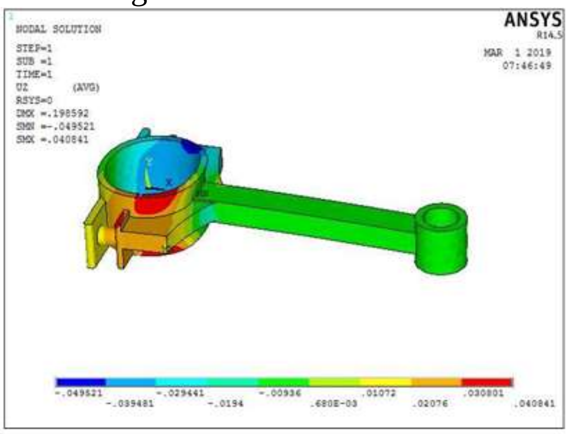


Fig5.15: Deformation in z direction of connecting rod From the above results maximum total deflection observed is 0.198mm

VonMises Stress:

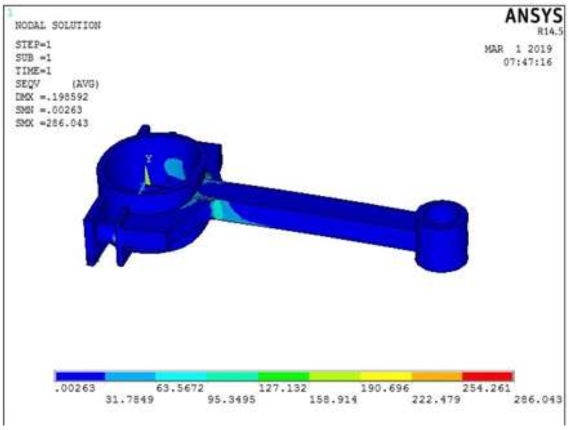


Fig5.16: VonMises stresses of connecting rod

From the results the maximum VonMises stress observed is 286Mpa

STRUCTURAL ANALYSIS PROCESS OF CONNECTING ROD USING E-GLASS/EPOXY COMPOSITE MATERIAL

Structural static analysis is performed on the connecting rod to identify the high stress regions and attempts are made to modify the connecting rod to reduce the stresses.

The crank and piston pin ends are assumed to have a sinusoidal distributed loading over the contact surface area, under tensile loading. This is based on experimental results.

In this study four finite element models were analyzed. FEA for both tensile and compressive loads were conducted. Two cases were analyzed for each case, one with load applied at the crank end and restrained at the piston pin end, and the other with load applied at the piston pin end and restrained at the crank end. In the analysis carried out, the axial load was 26.7 KN (6 kips) in both tension and compression.

The 3d model generated in UNIGRAPHICS is converted into parasolid. The parasolid file is imported into ansys and finite element analysis is carried out in Ansys.

Material properties

1. Material properties of composite materials (E-Glass/Epoxy):

Longitudinal Modulus (Ex): 50 GPa Transverse Modulus (EY): 12 GPa Shear modulus (Gxy): 5 .6GPa Shear modulus (Gyz): 5.6 GPa Shear modulus (Gxz): 5.6 GPa Poisson's Ratio: 0.33
Density: 2000 Kg/m3

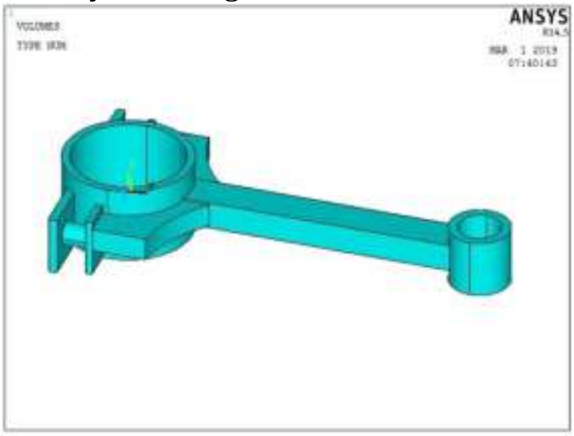


Fig 5.17: 3d model of the connecting rod imported into Ansys

Axial compressive load at the crank end

In this case axial compressive load of 26.7KN is applied at crank end with cosine distribution over 120º and piston pin end restrained over 120º

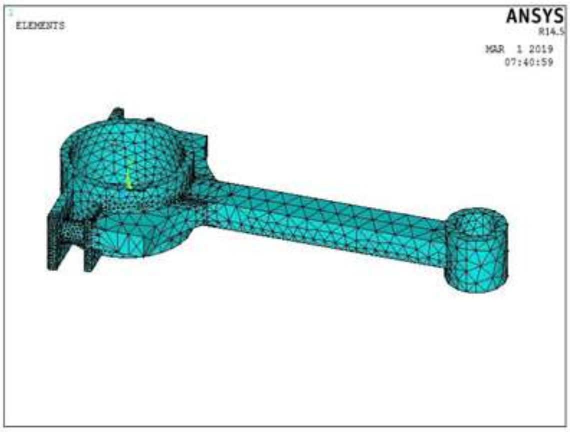


Fig5.18: Boundary conditions of the connecting rod for case-1 condition

International Journal of Research
Available at https://journals.pen2print.org/index.php/ijr/
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 06 Issue 10
September 2019

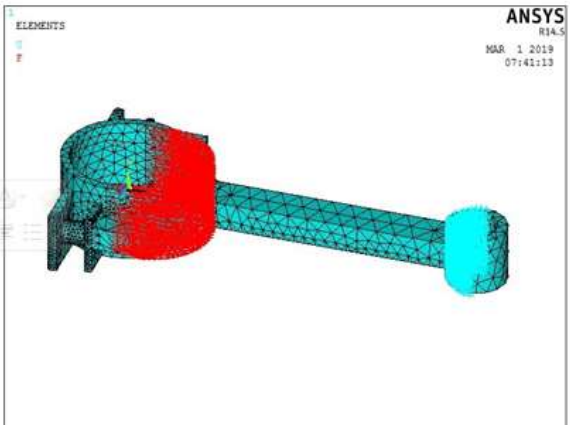
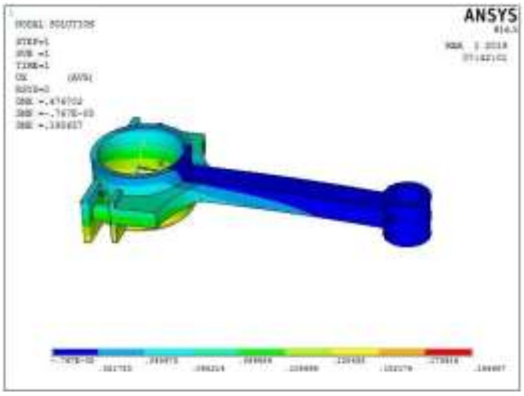


Fig5.19: Applied load condition of connecting rod

Since ansys does not have the option to apply parabolic load we have written a macro. To apply the parabolic load we have to read the macro



Results:

Total Deflection and Deflection in X-dir

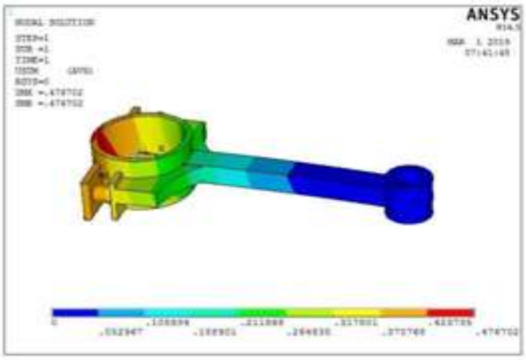


Fig5.20: Total deformation of connecting rod

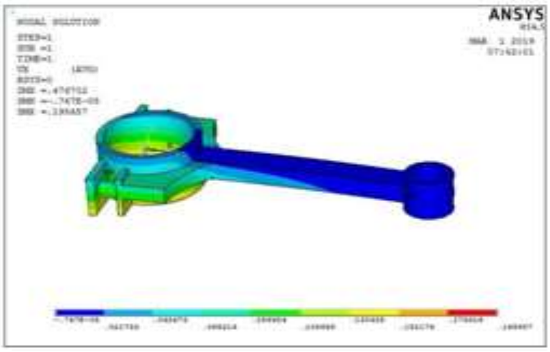
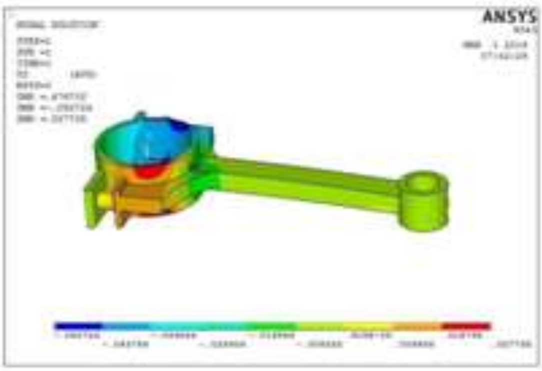


Fig5.21: Deformation in x direction of connecting rod

Deflection in Y-dir and Z-dir

Fig5.22 Deformation in y direction of connecting rod



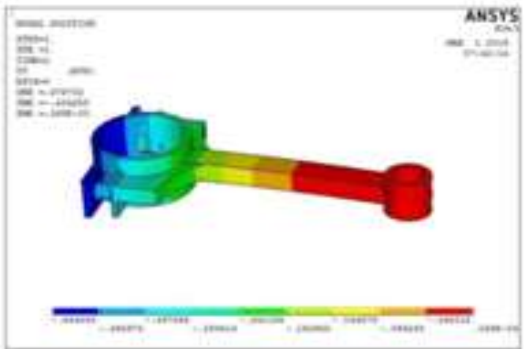
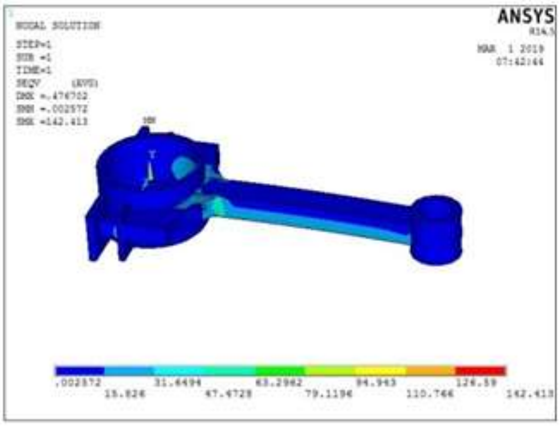


Fig5.23: Deformation in z direction of connecting rod

From the above results maximum total deflection observed is 0.476mm

VonMises Stress:

Fig5.24: VonMises stresses of connecting rod



From the results the maximum VonMises stress observed is 142.4Mpa

RESULTS AND CONCLUSION

The main aim of this project is to develop the Automobile connecting rod component with proper light weight, high strength composite material. In this project accurate design of connecting rod is been completed using NX-CAD software. After completion of design of connecting rod, analysis was estimated using Ansys software. Analysis of connecting rod is completed with different materials i.e., Steel, Composite HM Carbon/Epoxy and Composite E-Glass/Epoxy materials. From analysis results deformations and von misses stress results were find out for different materials. Results are mentioned in below table.

Types of result	Connecting rod using steel	Connecting rod using HM Carbon/Epoxy	Connecting rod using E-Glass/Epoxy
Deformation (mm)	1.07	0.198	0.476
Von misses stress (MPa)	826	286	142.4

Conclusion

Analysis of connecting rod is completed with different materials i.e., Steel, Composite HM Carbon/Epoxy and Composite E-Glass/Epoxy materials in this project.

Using steel

Von misses Stress found on connecting rod at loading condition was 826 MPa. This value is greater than yield strength of steel (280MPa). According to von misses stress theory connecting rod goes to fail using steel.

Using HM Carbon/Epoxy

Von misses Stress found on connecting rod at loading condition was 286 MPa. This value is less than ultimate strength of HM Carbon/Epoxy (800MPa). According to von misses stress theory connecting rod is safe using HM Carbon/Epoxy.

Using E-Glass/Epoxy

Von misses Stress found on connecting rod at loading condition was 142.4 MPa. This value is less than ultimate strength of E-Glass/Epoxy (900MPa). According to von misses stress theory connecting rod is safe using E-Glass /Epoxy.

References

[1]. Balasubramaniam, B., Svoboda, M., and Bauer, W.Structural optimization of I.C. engines subjected to mechanical and thermal loads. Comput. Meth. Appl.Mech. Engrg, 1991, 89, 337– 360.

[2]. Webster, W.D., Coffell, R., and Alfaro, D. A three dimensional finite element analysis of a high speed diesel engine connecting rod. SAE Technical Paper 831322, 1983.

[3]. Ishida, S., Hori, Y., Kinoshita, T., and Iwamoto, T.Development of technique to measure stress on connecting rod during firing operation. SAE Technical Paper 951797, 1995, pp. 1851– 1856.

[4]. Rice, R. C. (Ed.) SAE Fatigue design handbook,3rd edition, 1997 (Society of Automotive Engineers,Warrendale, PA).

[5]. Athavale, S. and Sajanpawar, P. R. Studies on some modelling aspects in the finite element analysis of small gasoline engine components. Proceedings of the small engine technology conference, Society of Automotive Engineers of Japan, Tokyo, 1991, pp. 379–389.

[6]. Pai, C. L. The shape optimization of a

connecting rod with fatigue life constraint. Int. J. Mater. Prod. Technol.1996, 11(5– 6), 357–370.
[7]. Shenoy, P. S. and Fatemi, A. Connecting rod optimization for weight and cost reduction. SAE Technical Paper 2005-01-0987, 2005