Design and Analysis of Connecting Rod with Mass Optimization

Apurv Mukund Dubey[1] Lakhan Ashokji Mohta[2]
[1,2]B. Tech. Student
[1,2]Department of Mechanical Engineering
[1,2]Government College of Engineering, Amravati, Amravati, Maharashtra, India

Abstract— Connecting rod is the intermediate link between the piston and the crank. It is responsible to transmit the push and pull from the piston pin to crank pin, thus converting the reciprocating motion of the piston to rotary motion of the crank. The main idea of this study is to do analysis of connecting rod and get idea of stress producing during compressive and tensile loading. Later it gives idea about weight reduction opportunities for a production steel connecting rod. This has entailed performing a detailed load analysis. Therefore, this study contain two subjects, first, load and stress analysis of the connecting rod, and second, optimization for weight reduction. In the first part of the study, loads acting on the connecting rod and find out stresses. In second part the existing design is optimized and mass reduction is achieved.

Key words: FEA, FEM, CAD, Optimization

I. Introduction

A. *Single Slider Crank chain*

A single slider crank chain is a modification of basic four bar chain. It consists of one sliding and three turning pairs. It is usually found in reciprocating steam engine mechanism. This mechanism convert rotary motion into reciprocating motion and vice versa.

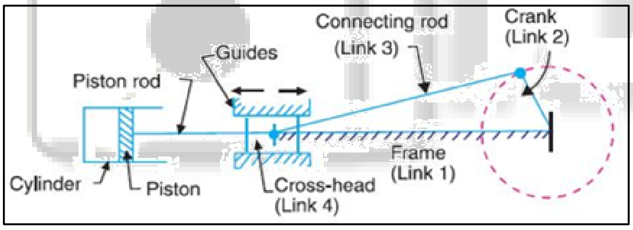


Fig. 1: Single slider crank mechanism

In single slider crank chain as shown in Fig 1.1 the link 1 and link 2 ,and link 2 and link 3, and links 3 and 4, form three turning pairs while the link 4 & 1 form sliding pair.

The link 1 corresponds to frame of engine which is fixed. The link 2 corresponds to crank; link 3 corresponds to connecting rod and link 4 corresponds to cross head. As the crank rotates the cross head reciprocates in the guides and thus piston reciprocates in the cylinder.

B. *Introduction to Connecting Rod*

Connecting rod is the intermediate link between the piston and the crank. And is responsible to transmit the push and pull from the piston pin to crank pin, thus converting the reciprocating motion of the piston to rotary motion of the crank. Connecting rod, automotives should be lighter and lighter, should consume less fuel and at the same time they should provide comfort and safety to passengers, that unfortunately leads to increase in weight of the vehicle. This tendency in vehicle construction led the invention and implementation of quite new materials which are light and meet design requirements. Lighter connecting rods help to decrease lead caused by forces of inertia in engine as it does not require big balancing weight on crankshaft.

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. The fact that forgings produce blow-hole-free and better rods gives them an advantage over cast rods.

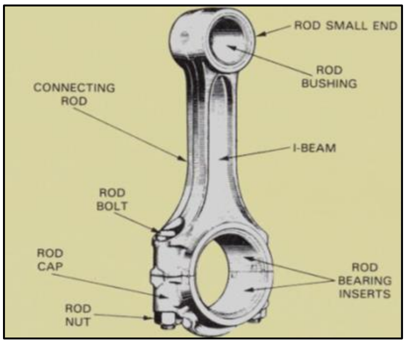


Fig. 2: Connecting rod

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 aluminium (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.

Connecting rods are highly dynamically loaded components used for power transmission in combustion engines. The optimization of connecting rod had already started as early year 1983 by Webster and his team. However, each day consumers are looking for the best from the best. That’s why the optimization is really important especially in automotive industry. Optimization of the component is to make the less time to produce the product that is stronger, lighter and less cost. The design and weight of the connecting rod influence on car performance. Hence, it is effect on the car manufacture credibility. Change in the structural design and also material will be significant increments in weight and performance of the engine.

Due to its large volume production, it is only logical that optimization of the connecting rod for its weight or volume will result in large-scale savings. It can also achieve the objective of reducing the weight of the engine component, thus reducing inertia loads, reducing engine weight and improving engine performance and fuel economy.

C. *Introduction to CATIA*

CATIA (Computer Aided Three-Dimensional Interactive Application) commonly referred to as a 3D Product Lifecycle Management software suite. CATIA supports

All rights reserved by *www.ijsrd.com*

multiple stages of product development, including conceptualization, design (CAD), engineering (CAE) and manufacturing (CAM). CATIA facilitates collaborative engineering across disciplines around its 3DEXPERIENCE platform, including surfacing and space design, mechanical engineering and system engineering.

D. Introduction to ANSYS

ANSYS is general-purpose Finite Element Modeling and Analysis software package. Finite Element Analysis is a numerical method of deconstructing a complex system into very small pieces (of user-designated size) called elements. The software implements equations that govern the behavior of these elements and solves them all; creating a comprehensive explanation of how the system acts as a whole. This type of analysis is typically used for the design and optimization of a system far too complex to analyze by hand. ANSYS provides a cost-effective way to explore the performance of products or processes in a virtual environment. This type of product development is termed virtual prototyping. With virtual prototyping techniques, users can iterate various scenarios to optimize the product long before the manufacturing is started. This enables a reduction in the level of risk, and in the cost of ineffective designs. ANSYS 14.0 is used for having the best computational results.

II. LITERATURE REVIEW

The connecting rod is subjected to a complex state of loading. It undergoes high cyclic loads of the order of 108 to 109 cycles, which range from high compressive loads due to combustion, to high tensile loads due to inertia. Therefore, durability of this component is of critical importance. Due to these factors, the connecting rod has been the topic of research for different aspects such as production technology, materials, performance simulation, fatigue, etc. For the current study, it was necessary to investigate finite element modeling techniques, optimization techniques, developments in production technology, new materials, fatigue modeling, and manufacturing cost analysis. This brief literature survey reviews some of these aspects.

Webster et al. (1983) performed three dimensional finite element analysis of a high-speed diesel engine connecting rod. For this analysis they used the maximum compressive load which was measured experimentally, and the maximum tensile load which is essentially the inertia load of the piston assembly mass. The load distributions on the piston pin end and crank end were determined experimentally. They modeled the connecting rod cap separately, and also modeled the bolt pretension using beam elements and multi point constraint equations.

Kuldeep B, Arun L.R, Mohammed Faheem [3] in which they replaced connecting rod by aluminum based composite material reinforced with silicon carbide and fly ash. And it also describes the modelling and analysis of connecting rod. FEA analysis was carried out by considering two materials. The parameters like von miss stress, von misses strain and displacements were obtained from ANSYS software. Compared to the former material the new material found to have less weight and better stiffness. It resulted in reduction of 43.48% of weight, with 75% reduction in displacement.

Mr. J.D.Ramani, Prof. Sunil Shukla, Dr. Pushpendra Kumar Sharma (2014) [4] studied the various loads acting on connecting rod and then modeled the connecting rod in Pro-E. They performed the analysis of connecting rod using ANSYS 14 under the action of compressive as well as tensile loads. In later part of analysis they gave a brief idea of mass optimization of connecting rod.

M.S. Shaari, M.M. Rahman, M.M. Noor, K. Kadirgama and A.K. Amirruddin [5] designed connecting rod of internal combustion engine using the topology optimization. The objectives of this paper are to develop structural modeling, finite element analyze and the optimization of the connecting rod for robust design. The structure of connecting rod was modeled utilized SOLIDWORKS software. Finite element modeling and analysis were performed using MSC/ PATRAN and MSC /NASTRAN software. Linear static analysis was carried out to obtain the stress/strain state results. The mesh convergence analysis was performed to select the best mesh or the analysis. The topology optimization technique is used to achieve the objectives of optimization which is to reduce the weight of the connecting rod. From the FEA analysis Result s, TET10 predicted higher maximum stress than TET4 and maximum principal stress captured the maximum stress. The crank end is suggested to be redesign based on the topology optimization results. The optimized connecting rod is 11.7% lighter and predicted low maximum stress compare to initial design. For future research, the optimization should cover on material optimization to increase the strength of the connecting rod.

K. Sudershan Kumar, Dr. k. Tirupathi Reddy, Syed Altaf Hussan "Modeling and analysis of two Wheeler connecting rod" [6] in this project connecting rod is replaced by Aluminum reinforced with Boron carbide for Suzuki GS150R motorbike. A 2D drawing is drafted from the calculations. A parametric model of connecting rod is modelled using PRO-E 4.0 software. Analysis is carried out by using ANSYS software. Finite element analysis of connecting rod is done by considering two materials, viz.. Aluminum Reinforced with Boron Carbide and Aluminum 360. 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. Compared to carbon steel, aluminum boron carbide and aluminum 360, Aluminum boron carbide is found to have working factor of safety is nearer to theoretical factor of safety, 33.17% to reduce the weight, to increase the stiffness by 48.55% and to reduce the stress by10.35% and most stiffer.

Vivek.c.pathade, Bhumeshwar Patle, Ajay N. Ingale "Stress Analysis of I.C. Engine Connecting Rod by FEM" [7] dealt with the stress analysis of connecting rod by finite element method using pro-e wild fire 4.0 and ansys work bench 11.0 software. And concluded that the stress induced in the small end of the connecting rod are greater than the stresses induced at the bigger end, therefore the chances of failure of the connecting rod may be at the fillet section of both end.

Om Parkash, Vikas Gupta, Vinod Mittal "Optimizing the Design of Connecting Rod under Static and Fatigue Loading" [8] main objective of this work is to re-

All rights reserved by *www.ijsrd.com*

optimize the existing design of connecting rod of universal tractor (U650) by changing some of the design variables .The existing design performs modelling and evaluates critical regions in the connecting rod under fatigue loading. In the present work,

The model is developed, analyzed and designed using CATIA 19, PRO-E and ANSYS workbench v12. Optimization of connecting rod is done under same boundary and loading conditions for variation in the few stress and fatigue parameters i.e. stresses, weight, life, damage, bi-axiality indication and safety factor. The allowable numbers of cycles under fully reversed fatigue loading are increased and assumed up to a maximum limit of 109 cycles. Stress concentration coefficient is varied to obtain the maximum cycles condition. The critical regions under both static and fatigue analysis are identified and improved. The connecting rod is modelled and optimized for the reduced weight, improved life and manufacturability. The results obtained from performed analysis can be used to modify the design of existing connecting rod, so that better performance i.e. reduced inertia, fatigue life and manufacturability can be obtained under varying static and fatigue conditions.

In a published SAE case study (1997), a replacement connecting rod with 14% weight savings was designed by removing material from areas that showed high factor of safety. Factor of safety with respect to fatigue strength was obtained by performing FEA with applied loads including bolt tightening load, piston pin interference load, compressive gas load and tensile inertia load. The study lays down certain guidelines regarding the use of the fatigue limit of the material and its reduction by a certain factor to account for the as-forged surface. The study also indicates that buckling and bending stiffness are important design factors that must be taken into account during the design process. On the basis of the stress and strain measurements performed on the connecting rod, close agreement was found with loads predicted by inertia theory. The study also concludes that stresses due to bending loads are substantial and should always be taken into account during any design exercise.

III. SYSTEM DEVELOPMENT

A. Material of connecting rod

Connecting rod is manufactured by drop forging process or by casting process. Generally for high strength and, material saving drop forging process is more commonly used. Various materials such as wrought steel forged steel, cast iron, aluminum, titanium, powdered metal etc. Most commonly use material among these is forged steel. The material properties of forged steel are as follows.

Sr.no	Property	Value
1	Young's Modulus of Elasticity (GPa)	201
2	Poisson's ratio	0.33
3	Density (kg/m^3)	7850
4	Yield stress (MPa)	550

Table 1: Properties of Forged steel

B. Detailed Dimensions of Connecting Rod

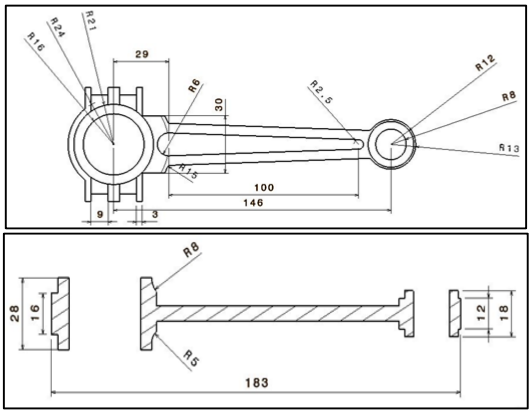


Fig. 3: Detailed dimensions of connecting rod

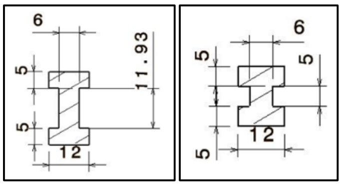


Fig. 4: Cross section of connecting rod

C. CAD Model Development

Detailed dimensions of connecting rod are shown in Fig.3 and Fig.4. CAD modeling software is dedicated for the specialized job of 3D-modelling. The model of the connecting rod includes many complicated parts. . It is very essential that the developed computer model is exactly a replica of the physical specimen. The connecting rod has been modelled in CATIA V5 R20 as shown in Fig.5. Connecting rod modelled in Part Design Workbench. Two modules of CATIA V5 R20 are used to develop the model of connecting rod.

– Sketcher Module.
– Part Module.

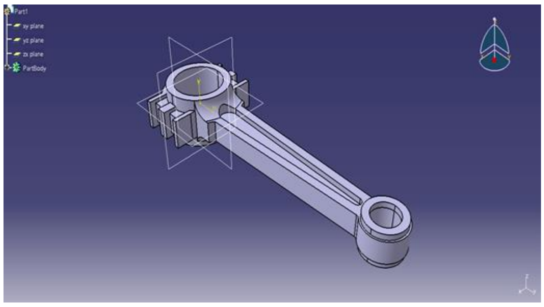


Fig. 5: Developed CAD model of connecting rod

IV. FEA OF CONNECTING ROD

Element analysis (FEA) has become common place in recent years, and is now the basis of a multibillion dollar per year industry. Numerical solutions to even very complicated stress problems can now be obtained routinely using FEA, and the method is so important that even introductory treatments of mechanics of materials such as these modules should outline its principal features.

FEA is having a large number of advantages such as the results can be visualized easily, the time of design is less, number of prototypes to be produced decreased, testing of prototypes has become easy, optimum design is achieved etc.

All rights reserved by *www.ijsrd.com*

A. Static Analysis of connecting rod

ANSYS 14 Mechanical APDL model was used for the analysis of connecting rod. CAD model of connecting rod was imported in ANSYS. Later Structural analysis was performed using following general procedure:

1) The CAD model was imported in ANSYS workbench
2) 10 node SOLID 187 element was selected for meshing
3) Material properties were defined.
4) Model was meshed with element size as 2 mm
5) Boundary conditions were applied
6) Solution was done
7) Results were analysed

1) Element Type

10 node SOLID 187 as shown in Fig.6. was the element selected for meshing the CAD model of connecting rod. SOLID187 element is a higher order 3-D, 10-node element. SOLID187 has quadratic displacement behaviour and is well suited to modelling irregular meshes. The element is defined by 10 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, hyper elasticity, creep, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyper elastic materials. The minimum edge length of the element was given as 2mm. This resulted in 55871 elements in the finite element model.

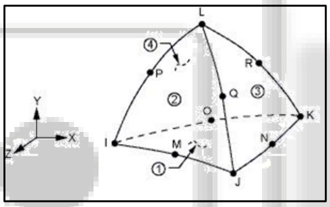


Fig. 6: Geometry of SOLID187

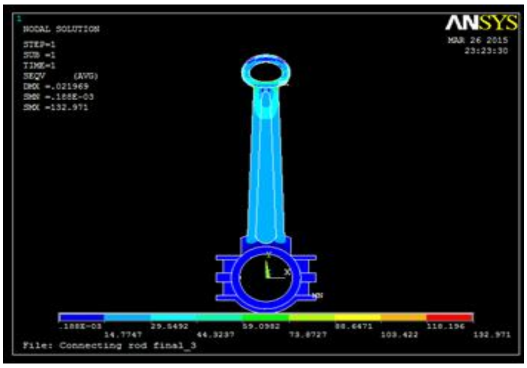
2) Material Properties

As discussed earlier the material selected for analysis is forged steel and material properties given in Table I are used.

3) Boundary conditions

Boundary conditions mean the constraints and forces applied to the FE model. In this case the analysis is carried out for compression and tension so different boundary conditions are applied to the FE model in both the cases.

In compression and tension maximum gas pressure of 16 MPa is applied on the connecting rod. For compressive loading 16 MPa pressure is applied at the small end and big end is kept fixed. For tensile loading 16 MPa pressure is applied at big end and small end is kept fixed. The results obtained are as shown in Fig.7 and Fig.8 and are summarized in Table 2.



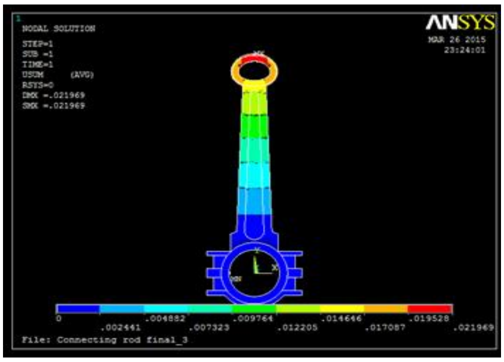
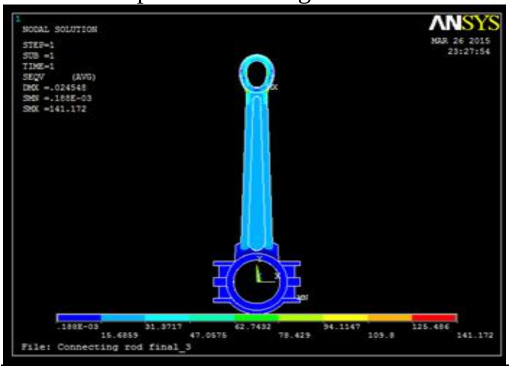


Fig. 7: Stresses and displacement of connecting rod in compressive loading condition



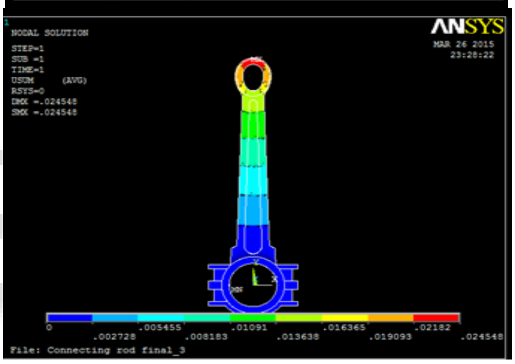


Fig. 8: Stresses and displacement of connecting rod in tensile loading condition

Sr. No.	Loading condition	Load applied (MPa)	Maximum Stress (N/mm^2)	Maximum Displacement (mm)
1	Compressive	16	132.971	0.021969
2	Tensile	16	141.172	0.024548

Table 2: Results of static analysis of connecting rod

V. OPTIMIZATION

Optimization means an act, process or methodology of making something (a design, system) as fully perfect, functional, or effective as possible. As stated earlier in the objectives of the project one of the objectives was optimization of the design for mass reduction. Generally optimization is done in terms of mass reduction or weight reduction due to following two reasons:

- As mass reduces material saving takes place and thus cost of production reduces
- As mass reduces the inertia forces acting on the rod reduces

In the optimization on current design the second aspect is used i.e. shape optimization. Some of the material is removed from the shank by giving a slight chamfer on its lateral edges. Due addition of this chamfer the mass of

All rights reserved by *www.ijsrd.com*

connecting rod is reduced by 9.89 grams. More amount of mass can be reduced by using composites of aluminum which are lighter in weight and have same strength as forged steel. The CAD model of optimized design of connecting rod is shown in Fig.9. The cross section of optimized design is shown in the Fig.10.

The main objective of optimization is to reduce mass and increase the strength of the existing design. In this project after optimizing the design it was found that the stresses induced are also reduced. Thus optimizing the design does not alter its safety and hence optimized design is also safe.

Similar boundary conditions were applied to the optimized design and the results obtained are summarized in Table III.

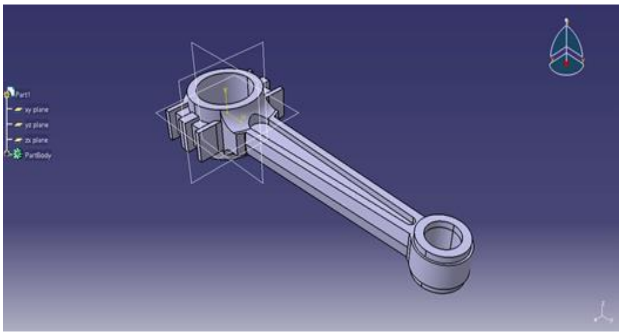


Fig. 9: CAD model Optimized design of connecting rod

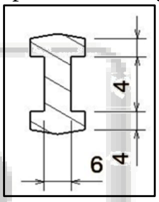


Fig. 10: Cross section of connecting rod

Sr. No.	Loading condition	Load applied (MPa)	Maximum Stress (N/mm^2)	Maximum Displacement (mm)
1	Compressive	16	129.791	0.023419
2	Tensile	16	149.86	0.02599

Table 3: Summary for optimized design of connecting rod

VI. CONCLUSION

The design of a standard connecting rod was modeled in CATIA and then static analysis was carried out with the help of ANSYS 14. Static analysis was carried out for compressive and tensile loading conditions. So following conclusions are derived:

- The stresses were found to be less than permissible value and hence the design is safe in both compression as well as in tensile loading.
- Optimization is done and mass is reduced by 9.89 grams.
- The stresses induced in optimized design are found to be less than the original stress value. Thus strength is increased.

REFERENCES

[1] http://www1.ansys.com/customer/content/documentation/121/ans_tut.pdf

[2] Nitin S. Gokhale, Sanjay S. Deshpande, Sanjeev V.Bedekar, Anand N Thite "Practical Finite Element Analysis" First Edition.

[3] Kuldeep B, Arun L.R, Mohammed Faheem, " Analysis and optimization of connecting rod using ALFASiC Composite material", International Journal of Innovative Research in Science, Engineering and Technology, June 2013.

[4] Mr. J.D.Ramani, Prof. Sunil Shukla, Dr. Pushpendra Kumar Sharma, "FE-Analysis of Connecting Rod of I.C.Engine by Using ANSYS for Material Optimization", Mr. J.D.Ramani et al Int. Journal of Engineering Research and Applications ISSN : 2248-9622, Vol. 4, Issue 3(Version 1), March 2014, pp.216-220.

[5] M.S. Shaari, M.M. Rahman, M.M. Noor, K. Kadirgama and A.K. Amirruddin, "Design of connecting rod of internal combustion engine: A Topology Optimization Research" National Conference in Mechanical Engineering Research and Postgraduate Studies (2nd NCMER 2010) 3-4 December 2010, Faculty of Mechanical Engineering, UMP Pekan, Kuantan, Pahang, Malaysia; pp. 155-166 ISBN: 978-967-0120-04-1.

[6] K. Sudershan Kumar, Dr. k. Tirupathi Reddy, Syed Altaf Hussan "Modeling and analysis of two Wheeler connecting rod", International Journal of Modern Engineering Research, Vol -2, Issue- 5, pp-3367-3371, Sep-Oct-2012.

[7] Vivek.c.pathade, Bhumeshwar Patle, Ajay N. Ingale "Stress Analysis of I.C. Engine Connecting Rod by FEM", International Journal of Engineering and Innovative Technology, Vol-1, Issue-3, pp-12-15, March2012.

[8] Om Parkash, Vikas Gupta, Vinod Mittal "Optimizing the Design of Connecting Rod under Static and Fatigue Loading", International Journal of Research in Management, Science & Technology (E-ISSN: 2321-3264) Vol. 1; No. 1, June 2013.

All rights reserved by *www.ijsrd.com*