iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 6 **Issue:** IV **Month of publication:** April 2018

DOI: http://doi.org/10.22214/ijraset.2018.4129

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue IV, April 2018- Available at www.ijraset.com

Design and Comparative Analysis of Connecting Rod using Finite Element Analysis

Vinayak Chumbre[1], Vinayak Kallannavar[2], Anilkumar Shirahatti[4], Ratan Patil[2], Shirish M. Kerur[3]
[1]Student, Jain College of Engineering, Belgaum
[2]Assistant Professor, Jain College of Engineering, Belgaum
[2]Associate Professor, Jain College of Engineering, Belgaum
[3]Professor, Jain College of Engineering, Belgaum

Abstract*: Connecting rod is an intermediate link which connects the piston and the crankshaft in an internal combustion engine, the main work of connecting rod is to convert the linear motion of the piston (thrust force) into rotary motion of the crankshaft. In this study, an attempt has been made to analyze and understand the connecting rod structure using Finite Element Analysis method. An invariable model of connecting rod is modelled using NX 6.0 and on this model static structural analysis is carried out by using ANSYS14.5 simulation tool. Further analysis was carried out by considering different materials to understand the variations of equivalent von-mises stress, strain, total deformation and factor of safety.
Keywords*: Connecting rod, Finite element analysis, ANSYS Workbench, Static structural analysis.

I. INTRODUCTION

Connecting rod is an intermediate member which converts the linear motion of piston into rotary motion of crankshaft. Connecting rod consists of two ends, one is the smaller end which is connected to piston by means of piston pins or gudgeon pin whereas the bigger end is connected to the crankshaft by means of bearing. During operation thrust force is exerted on the piston by the combustion of the fuel, this force is to be transmitted to the crankshaft via connecting rod, due to this force the connecting rod undergoes tremendous amount of stress. Kumar et. al. [1] carried out finite element analysis of the four-stroke petrol engine connecting rod using ANSYS workbench in the aim of weight reduction& also reducing the inertia force on the connecting rod. Fatigue analysis is also carried out & results showed a reduction of 3.05% of weight of original connecting rod. Roy [2] made an attempt to understand various designs of connecting rod and analyse them to get an optimal design. The selected design had better results in various parameters and were under safe stress. It was observed that the variation in stress and strain result from the existing analysis was of 9.6% and 10.5% respectively. Shenoy et. al. [3] carried out dynamic analysis and optimization of connecting rod. The dynamic analysis was carried out keeping cost and weight reduction parameters as main focus points. Dupare et. al. [4] obtained the maximum weight reduction in a connecting rod, without affecting the main structural parameters. Vegi et. al. [5] proposed a new design and carried out analysis of connecting rod to study the variation in the results using different materials from carbon steel to forged steel. It is observed that change in material had no effect on equivalent stress for both the cases, but they reported that for forged steel material the factor of safety and stiffness have been improved, also the weight of forged steel was observed to be less than the carbon steel. Joshi et. al. [6] worked on the optimization of weight and designing of connecting rod by considering different materials such has high strength carbon fiber, stainless steel and aluminium alloy. They reported that the von-mises stress for carbon fiber is less compared to other materials. Kumar et. al. [7] carried out dynamic analysis on connecting rod using Bajaj pulsar (150cc) in order to reduce the weight and also reduce the moment of inertia. They reported that 42CrMo steel alloy is 11.67 % lighter material compared to 20CrMo and 6.42 % lighter when compared to 30CrMo steel. Bhargav et. al. [8] compared different materials by carrying out both static and dynamic analysis on a connecting rod. It is observed that the von-mises stress and weight of Al-MWCNT (Multi Wall Carbon Nano Tube) are less compared to Ti-6Al-4V, E-glass and carbon steel. Taware et. al. [9] carried out FEA analysis of connecting rod used in Hero Splendor motorbike and studied the effect of change in material from ASTM A216 GR WCB and Aluminium 360, then the results were compared and concluded that there is less deformation in ASTM A216 GR WCB which helps in long durability and also it is cheaper. Kuldeep et. al. [10] compared different material by carrying FEA analysis on connecting rod with the aim of weight reduction and increasing the stiffness. Ramakrishna et. al. [11] studied the effect of change in materials by carrying out FEA analysis on connecting rod, reduction in the weight of connecting rod was observed by replacing the material from 4340 alloy steel to AlSiC-9 and found that AlSiC-9 is 61.65% lighter compared to 4340 alloy steel connecting rod. In last few decades, many researchers [12-18] have worked on optimization of automotive vehicle components to enhance the performance characteristics but major emphasis is given on engine components.

In current study, the finite element analysis is carried out on the connecting rod to understand the design parameters like stress, strain, deformation etc. by considering different materials. Suitable dimensional changes are suggested based on simulation results, to the existing connecting rod design in order to optimise the structure.

II. FINITE ELEMENT ANALYSIS

Geometrical model of connecting rod was constructed using NX 6.0 modelling tool. The dimensions of connecting rod as shown in Fig. 1 are maintained same as that of parametric model used by Roy [2]. Structural steel is used as the specimen material for the simulations so that the FEA results can be compared with the previously published results. The material properties of the structural steel are listed in Table. I and the stress-strain curve for the same is plotted in Fig. 2.

Static structural analysis is performed on the parametric model of connecting rod using ANSYS 14.5 simulation tool. The geometric model was meshed to get 7758 number of tetrahedral elements and 14168 number of nodes as shown in Fig. 3 (a). During simulation crank end (larger diameter) is fixed and bearing compressive load of 4319 N was applied at the piston end (smaller diameter) as shown in Fig. 3 (b).

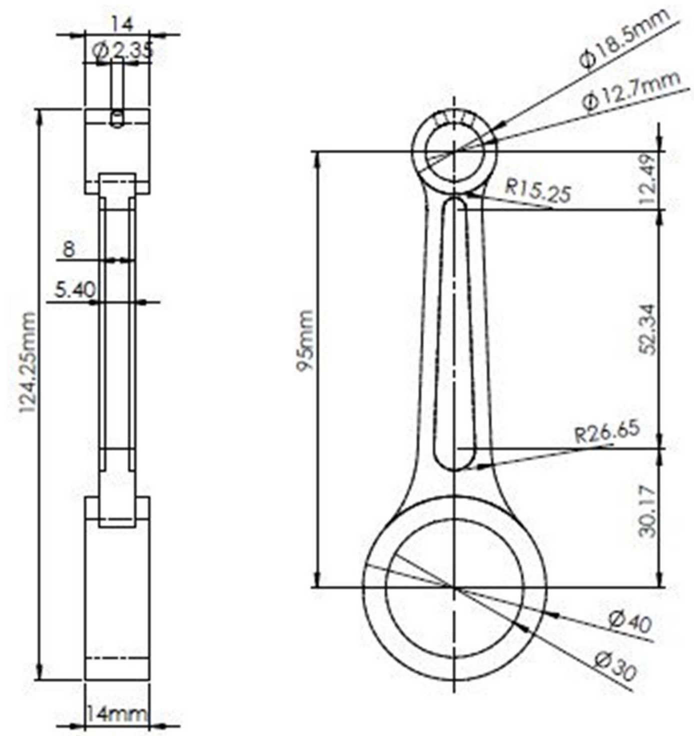


Fig. 1 Existing design of connecting rod [2]

TABLE I
MATERIAL PROPERTIES OF STRUCTURAL STEEL

Density (kg/m^3)	7850
Coefficient of thermal expansion (K^{-1})	1.2x10^{-5}
Young's Modulus (MPa)	2x10^5
Poisson's ratio	0.30
Bulk Modulus (GPa)	166.67
Shear Modulus (GPa)	76.92
Tensile yield strength (MPa)	250
Tensile ultimate strength (MPa)	460

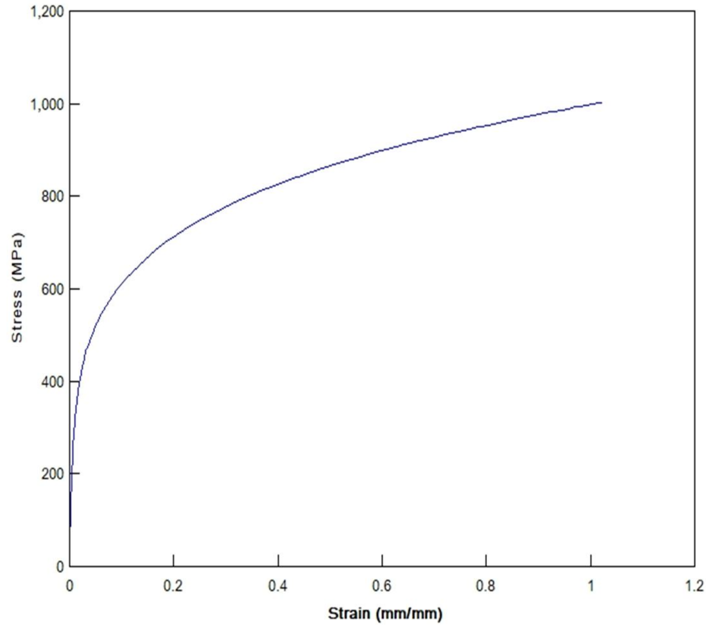
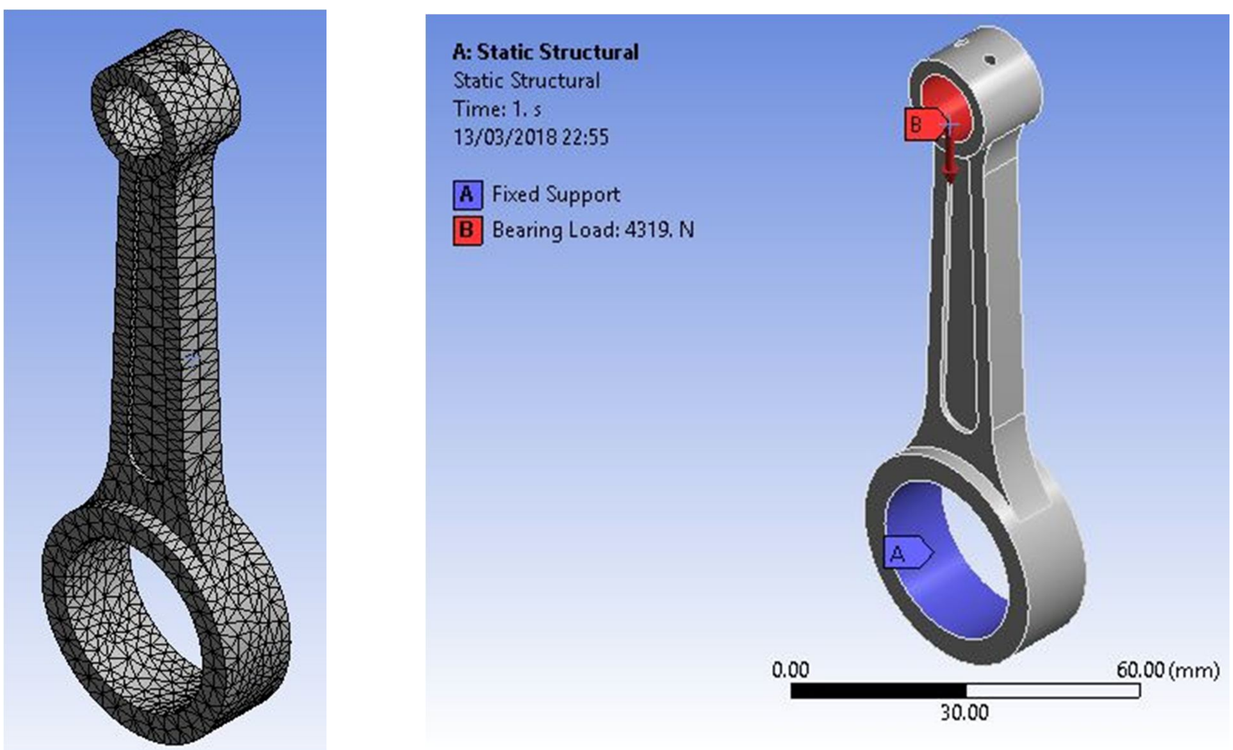


Fig. 2 Stress- strain curve for structural steel



(a) (b)

Fig. 3 Pre-processor details used for simulation (a) Meshed model of connecting rod (b) Loading and boundary conditions adopted

III. RESULT AND DISCUSSION

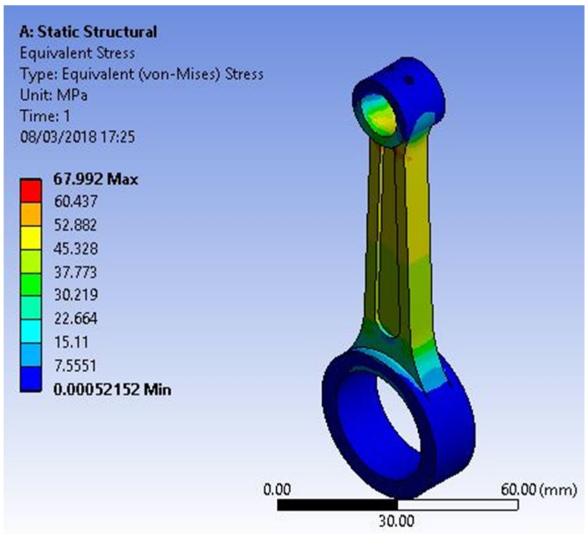
Initially static structural analysis is performed on a parametric model used by Roy [2]. The results obtained from the simulations were then compared with previously published results [2]. From Table. II it is evident that the results are in good agreement with each other. Variation of 1.3% was observed in equivalent von-mises stress magnitudes whereas deviation of 2.8% was observed in magnitude of equivalent elastic strain. 8.3 μm of variation was noted in magnitude of total deformation values.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue IV, April 2018- Available at www.ijraset.com

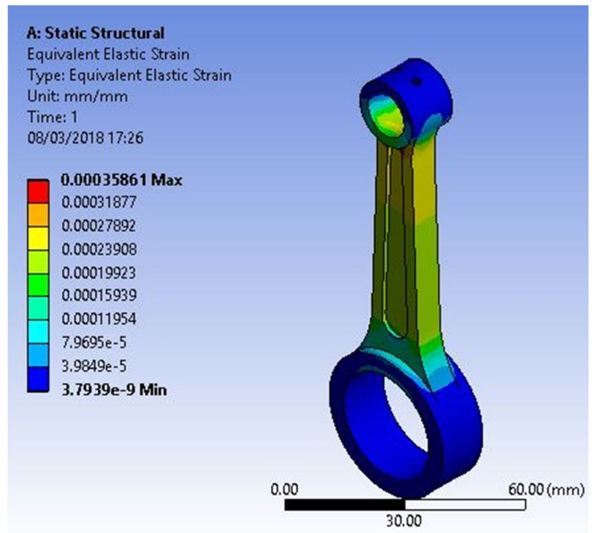
TABLE II
COMPARISON BETWEEN PUBLISHED AND OBTAINED RESULTS

Design Parameters	FEM Results	Published Results [2]
Equivalent von-mises Stress (MPa)	67.99	68.87
Equivalent elastic strain (mm/mm)	0.00035	0.00034
Total deformation (mm)	0.0178	0.0261
Factor of safety	3.67	---

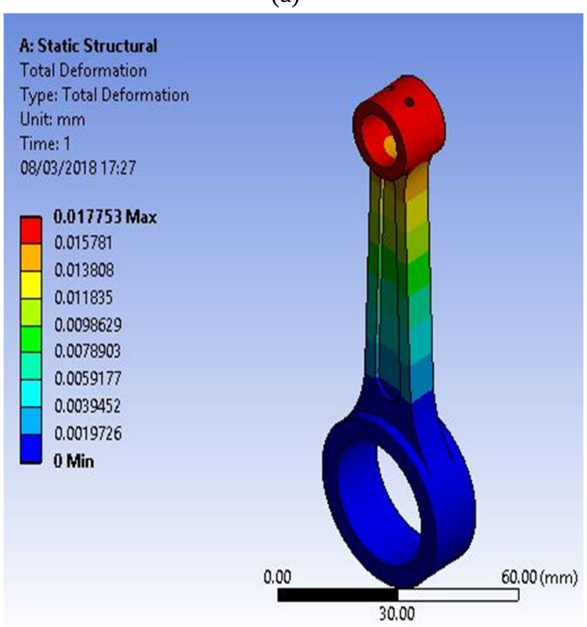
Fig. 4 shows the Equivalent von-mises stresses, equivalent elastic strain, total deformation and factor of safety (FOS) distribution in connecting rod.



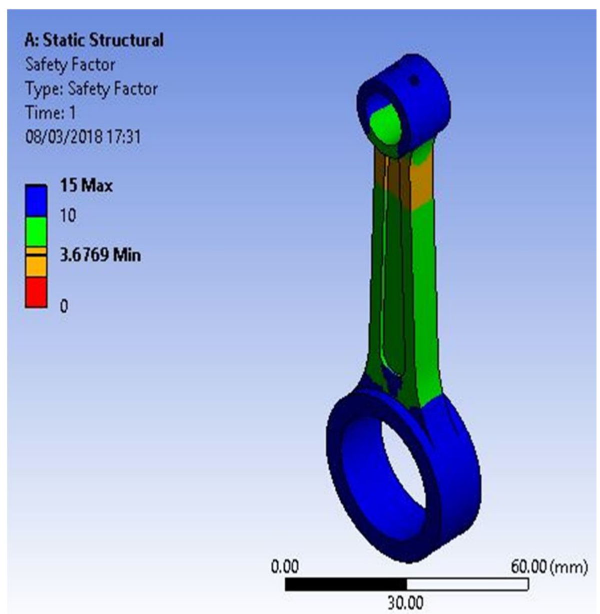
(a)



(b)



(c)



(d)

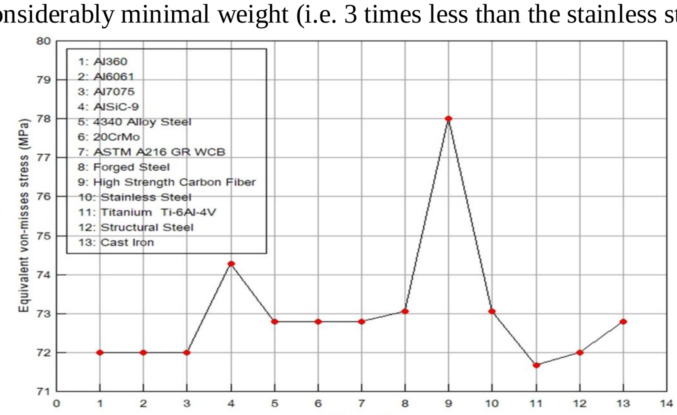
Fig. 4 FEA Results obtained for existing model (a) Equivalent von-mises stresses (b) Equivalent elastic strain (c) Total deformation (d) Factor of safety

Further, the study was extended to understand the effect of material selection in design process of connecting rod by considering different materials. Results obtained from analysis of different materials are listed in Table. III. Maximum stress was observed in high strength carbon fiber material (78.015 MPa) whereas Titanium Ti-6Al-4V experiences the lowest stress (71.67 MPa). Forged Steel experiences minimum strain (0.00036429 mm/mm) whereas Al360 alloy experiences highest strain (0.001319 mm/mm). Maximum deformation was observed in Al360 alloy (0.058255 mm) whereas Forged Steel experiences the lowest deformation of 0.015852 mm. FOS (at critical sections) of magnitude 10 was observed in 4340 Alloy Steel or EN24T material whereas minimum FOS of 2.36 was observed in Al360 material. It was also observed that minimum of 26.83 grams of material is required for high strength carbon fiber whereas maximum of 134.67 grams of material is required for Stainless Steel.

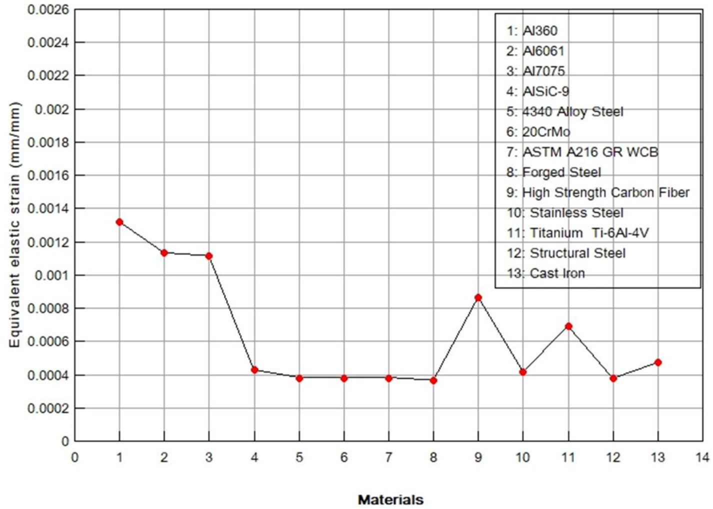
TABLE Ⅲ
COMPARATIVE FEA RESULTS FOR DIFFERENT MATERIALS

Materials	Equivalent von-mises stress (MPa)	Equivalent elastic strain (mm/mm)	Total deformation (mm)	Weight of connecting rod (grams)	Factor of Safety
Al360	71.99	0.001319	0.058255	46.95	2.36
Al6061	71.99	0.001131	0.049933	43.80	5.00
Al7075	71.99	0.001113	0.049160	45.28	4.50
AlSiC-9	74.28	0.000427	0.018287	50.47	7.40
4340 Alloy Steel or EN24T	72.79	0.000381	0.016673	131.65	10.00
20CrMo	72.79	0.000381	0.016673	131.82	9.41
ASTM A216 GR WCB	72.79	0.000381	0.016673	131.36	6.66
Glass-Epoxy	74.93	0.002437	0.103370	43.60	4.50
Forged Steel	73.05	0.000364	0.015852	129.13	8.56
High Strength Carbon Fiber	78.01	0.000866	0.035264	26.83	8.00
Stainless Steel	73.05	0.000417	0.018151	134.67	3.00
Titanium Ti-6Al-4V	71.67	0.000691	0.030691	74.29	9.00
Structural Steel	71.99	0.000376	0.017885	131.52	3.28
Cast Iron	72.79	0.000471	0.020596	120.68	2.75

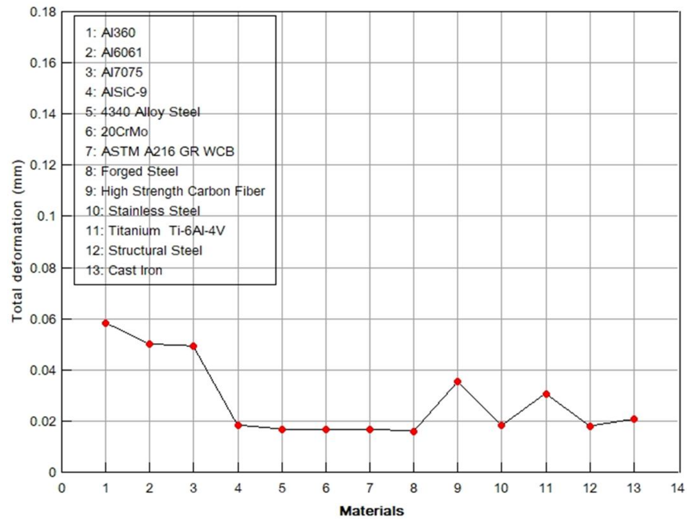
For better assessment the FEA results obtained for different materials (listed in Table. III) are plotted in Fig. 5. From Fig. 5 it is evident that maximum weight of 134.67 grams is required for stainless steel whereas high strength carbon fiber needs 26.83 grams of material to withstand the given load (i.e. 5 times less than the stainless steel material). It is also observed that alloys of aluminium can also withstand the load with considerably minimal weight (i.e. 3 times less than the stainless steel).



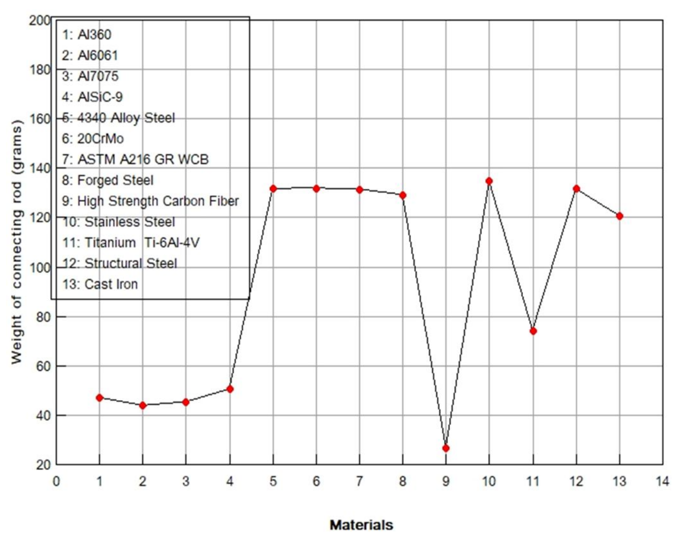
(a)


Equivalent elastic strain (mm/mm)
Materials
1: Al360
2: Al6061
3: Al7075
4: AlSiC-9
5: 4340 Alloy Steel
6: 20CrMo
7: ASTM A216 GR WCB
8: Forged Steel
9: High Strength Carbon Fiber
10: Stainless Steel
11: Titanium Ti-6Al-4V
12: Structural Steel
13: Cast Iron

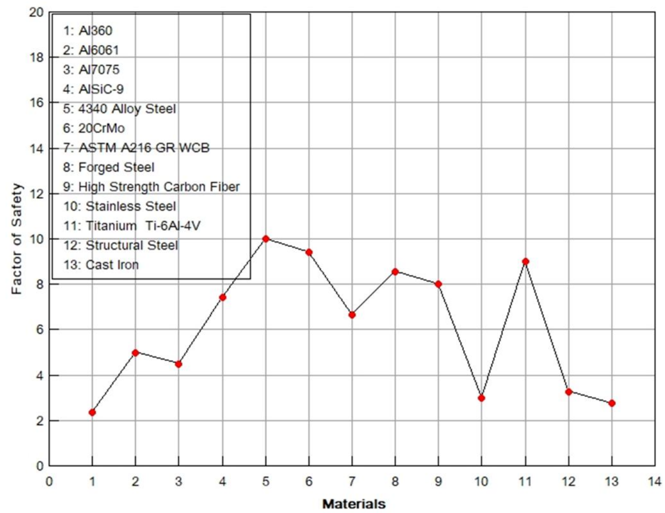
(b)


Total deformation (mm)
Materials
1: Al360
2: Al6061
3: Al7075
4: AlSiC-9
5: 4340 Alloy Steel
6: 20CrMo
7: ASTM A216 GR WCB
8: Forged Steel
9: High Strength Carbon Fiber
10: Stainless Steel
11: Titanium Ti-6Al-4V
12: Structural Steel
13: Cast Iron

(c)


Weight of connecting rod (grams)
Materials
1: Al360
2: Al6061
3: Al7075
4: AlSiC-9
5: 4340 Alloy Steel
6: 20CrMo
7: ASTM A216 GR WCB
8: Forged Steel
9: High Strength Carbon Fiber
10: Stainless Steel
11: Titanium Ti-6Al-4V
12: Structural Steel
13: Cast Iron

(d)



(e)

Fig. 5 FEA Results obtained for Different materials (a) Equivalent von-mises stresses (b) Equivalent elastic strain (c) Total deformation (d) weight of connecting rod (e) Factor of safety

From Fig. 5 it is evident that though the variations in results were observed, they are very small in case of stresses, strains & total deformation. Hence, the study was extended to change the design parameters involved in existing connecting rod model. Minor modifications were done on the existing parametric model of connecting rod design without altering main dimensions (not to disturb the mating part alignment) as shown in Fig. 6.

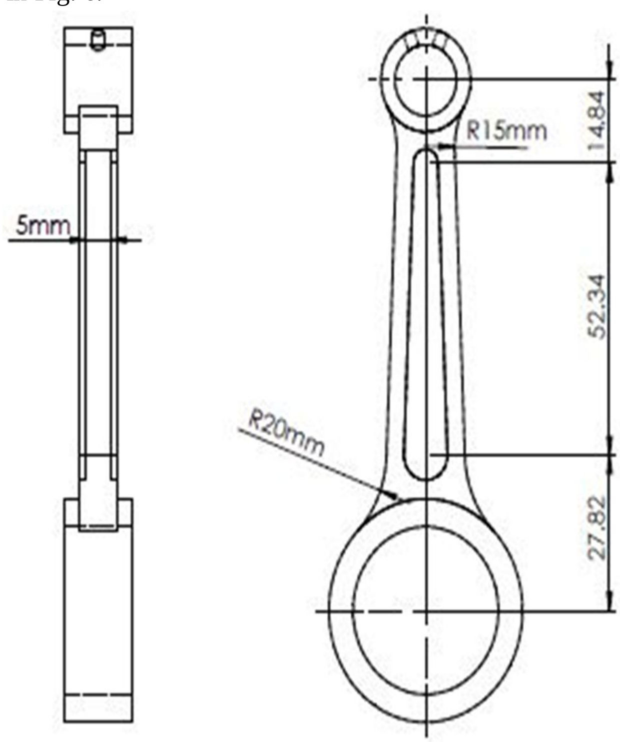


Fig. 6 Modified design of connecting rod

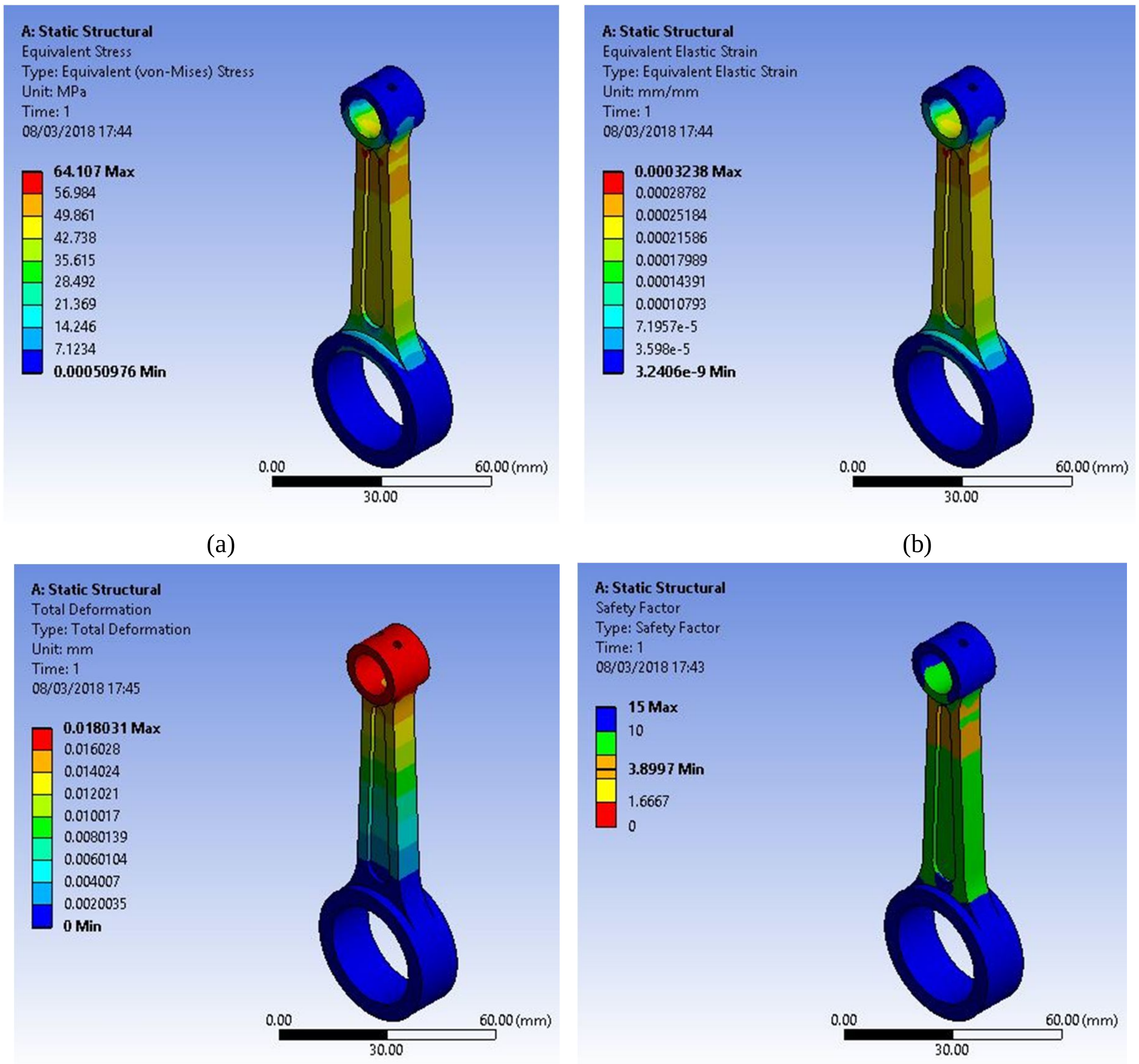
FEA simulations were conducted on the modified design considering the same material as that of existing parametric connecting rod. It was observed that the stress levels decreased drastically from 67.992 MPa to 64.107 MPa as shown in Fig. 7. Weight reduction of 1.5% was observed in new design compared to existing design of connecting rod. Comparative results of existing and new modified design are listed in Table IV.

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue IV, April 2018- Available at www.ijraset.com

TABLE IV

Comparative Fea Results Of Existing And Modified Design Of Connecting Rod

	Parameters	Existing Results	New Results	Variation
1	Equivalent von-mises stress (MPa)	67.99	64.11	5.7 %
2	Equivalent elastic strain (mm/mm)	0.00035861	0.00032381	9.7 %
3	Total Deformation (mm)	0.017753	0.018031	1.5 %
4	Factor of Safety	3.6769	3.8997	5.7 %
5	Weight of Connecting rod (grams)	129.90	127.96	1.5 %



(c) (d)

Fig. 7 FEA Results obtained for modified model (a) Equivalent von-mises stresses (b) Equivalent elastic strain (c) Total deformation (d) Factor of safety

IV. CONCLUSIONS

In this work, finite element analysis of connecting rod was carried out using ANSYS simulation tool. Series of simulations were carried out to understand the effect of material selection in connecting rod design by considering different materials. Wide range of variations were observed in the magnitude of Equivalent von-mises stresses, Equivalent elastic strain, Total deformation and Factor of safety for different materials. To optimize the design of connecting rod minor modifications were done without altering the main dimensions. Weight reduction in the component was observed in the modified design from 129.9 grams to 127.96 grams along with lower magnitude of stresses.

REFERENCES

[1]. Grover, Kamaldeep, and BalvinderBudania. "Optimization of connecting rod parameters using CAE tools." International Journal of Latest Trends in Engineering and Technology 3 (2012): 98-104.
[2]. Roy, D. R. B. K. "Design Analysis and Optimization of Various parameters of Connecting Rod using CAE software." International Journal of New Innovations in Engineering and Technology 1.1 (2012): 52-63.
[3]. Shenoy, Pravardhan S., and Ali Fatemi. "Connecting rod optimization for weight and cost reduction." No. 2005-01-0987. SAE Technical Paper, 2005.
[4]. Yogesh. B. Dupare, Raju.B.Tirpude and Akshay.Y.Bharadbhunje. "Fatigue analysis in connecting rod using ansys."International Journal of Modern Trends in Engineering and Research, Volume 02, Issue 02, February - 2015.
[5]. Vegi, Leela Krishna, and VenuGopalVegi. "Design and analysis of connecting rod using forged steel." International Journal of Scientific & Engineering Research 4.6 (2013): 2081.
[6]. Joshi, Prateek, and Mohammad UmairZaki. "FEM analysis of connecting rod of different materials using ANSYS." International Journal of Engineering and Techniques 3.1 (2015).
[7]. Kumar, Amit, P. P. Bhingole, and Dinesh Kumar. "Dynamic Analysis of Bajaj Pulsar 150cc Connecting Rod Using ANSYS 14.0." Asian Journal of Engineering and Applied Technology (2014): 19-24.
[8]. Bhargav, I. Sai, M. PavanKalyan, and N. Charishma. "Design and Comparative Analysis of Connecting Rod Using Composite Materials." International Journal of Engineering in Advanced Research Science and Technology 4 (2015).
[9]. Taware, RavirajYashwant, and AbhayArunUtpat. "Analysis of Connecting Rod Used in Two Wheeler under Static Loading by FEA."International Journal of Engineering Trends and Technology (IJETT) – Volume 20 Number 1 – Feb 2015.
[10]. Kuldeep B, Arun L.R, Mohammed Faheem. "Analysis and optimization of connecting rod using ALFASiC composites." International Journal of Innovative Research in Science, Engineering and Technology, Vol. 2, Issue 6, June 2013.
[11]. Ramakrishna, G., and P. H. J. Venkatesh. "Modelling and analysis of connecting rod using 4340 alloy steel and alsic-9."International Journal Of Engineering Sciences & Research Technology, 4.(12): December, 2015.
[12]. Charkha, Pranav G., and Santosh B. Jaju. "Analysis & optimization of connecting rod." Emerging Trends in Engineering and Technology (ICETET), 2009 2nd International Conference on. IEEE, 2009.
[13]. KallappaKhannukar, Vinayak Kallannavar, Dr. B. S.Manjunath ‘Dynamic Analysis Of Automotive ChassisUsing FEA’ International Research Journal OfEngineering And Technology (IRJET) E-ISSN: 2395-0056 Volume: 02 Issue: 09, Dec-2015.
[14]. Cho, J. R., et al. "Nonlinear finite element analysis of swaging process for automobile power steering hose." Journal of Materials Processing Technology 170.1-2 (2005): 50-57.
[15]. Helnwein, Peter, et al. "A new 3-D finite element model for cord-reinforced rubber composites—application to analysis of automobile tires." Finite elements in analysis and design 14.1 (1993): 1-16.
[16]. Sung, S. H., and Donald J. Nefske. "A coupled structural-acoustic finite element model for vehicle interior noise analysis." Journal of vibration, acoustics, stress, and reliability in design 106.2 (1984): 314-318.
[17]. Pifko, A. B., and R. Winter. "Theory and application of finite element analysis to structural crash simulation." Computational Methods in Nonlinear Structural and Solid Mechanics. 1981. 277-285.
[18]. Grujicic, M., et al. "Finite element analysis of the effect of up-armouring on the off-road braking and sharp-turn performance of a high-mobility multi-purpose wheeled vehicle." Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering 223.11 (2009): 1419-1434.