INTERNATIONAL JOURNAL OF ENGINEERING TECHNOLOGY AND SCIENCES (IJETS)
ISSN: 2289-697X (Print); ISSN: 2462-1269 (Online) Vol.6 (1) June 2019

DOI:http://dx.doi.org/10.15282/ijets.6.1.2019.1003

Static Analysis and Optimization of a Connecting Rod

Aisha Muhammad [1,3*], Ibrahim Haruna Shanono[2]

[1]*Mechatronics Engineering, Faculty of Engineering, University Malaya, Malaysia*
[2]*Department of Electrical Engineering Bayero University Kano Nigeria*
[3]*Department of Mechatronics Engineering Bayero University Kano Nigeria*
[*]*Email: ayshermuhd@gmail.com*

Abstract- Connecting rods are mechanical parts made use of for producing movement from a crankshaft's piston rotating activity. The integrity, as well as efficiency of vehicles, relies on the design of the connecting rod. There have actually been various reported situations of connecting rod failing based upon the architectural style, loading as well as the sort of materials made use of in its manufacturing. For the regular requirements as well as security of the customers, an enhanced as well as the optimal connecting rod is required resulting in the requirement for optimization to accomplish greater success in auto markets by enhancing its efficiency. Cheaper and qualitative production of the connecting rod can be achieved in a short period of time as a result of optimization. In this paper, Finite Element Method (FEM) using ANSYS workbench was used to carry out the weight optimization of the connection rod with target weight reduction of 20%, 30%, 40%, 50%, and 60% under a loading force of 30KN to determine the mass that needs to be remove to minimize both weight and cost. Furthermore, structural optimization is done to determine an optimized structure with new deformation, Von-misses stress, and equivalent elastic strain values followed by the comparison of these values before and after the structural optimization to verify the effect of the analysis. Based on the results obtained, it can be concluded that ANSYS software can be employed by production companies to minimize material wastages and maximize profits at the same time maintaining product quality and reliability.

Keywords - ANSYS, connecting rod, FEA, Deformation, stress analysis.

I. INTRODUCTION

Connecting Rod is a vital element in reciprocating I.C engines. Every automobile that makes use of an internal combustion engine needs at the very least one connecting rod relying on the variety of cylindrical tubes in the engine. It goes through high cyclic tons of the order of 108 to109cycles, which vary from high compressive loads because of inertia [1]. When it is subjected to alternative compressive and tensile stresses in a cycle of the engine, chances of fatigue failure in connecting rod are higher. [2]. Connecting rod is used in producing Rotating motion from a crankshaft's piston alternating movement. The engine combustion gases as well as parts activity of inertia apply stress that generates tensile and also compressive stress in the connecting rod. [3] [4]. Power transmission in a combustion engine is done using connecting rods. Its performance is influenced by the design and weight as such changing such factors will make a crucial improvement in its performance [5]. Longevity of the components is of crucial significance. Because of these aspects, the connecting rod has actually been the subject of research study for various facets such as manufacturing, materials, efficiency simulation, and so on.

It is essential to establish the fatigue life of the existing connecting rod. The connecting rod transfers motion and also pressure from piston to the crankshaft pin. When it works inside the engine, the connecting rod creates a component of 4 bar web link system. The connecting rod has 2 ends, a small end that is attached to the piston and also a big end that is linked to the crankshaft. The intermediate component in between these ends is "I Section", "H Section", "Circular area" or "Rectangular kind". The kind of section relies on the design demand as well as where it is likely to operate inside the engine. Usually this is enhanced based upon the forces acting upon it and the schedule of area. The connecting rod is subjected to rotating stresses of tensile as well as compressive forces when the crankshaft turns 2 revolutions in a 4-stroke diesel engine. [2].

The integrity of connecting rod has actually come to be a vital facet in the research study and also enhancement of its efficiency in automobile engine. With the constant growth of the automobile technology, specifically the fast advancement of computer system innovation, making use of the Finite element approach to evaluate the stress and also the optimization of the connecting rod is gradually taken seriously [6], [7], [8]. The light-weight design of the engine has actually ended up being a large issue that cannot be overlooked in the entire procedure of vehicle design. To accomplish this, the major elements of engine have to be optimized to decrease the quantity and also weight [9]. The credibility and performance of vehicles depends on the design of the connecting rod. Failure of connecting rod is attributed to the in availability of much strength needed to hold the load. This can be overcome with the life cycle extended by increasing the strength. [18], [19]. Failure analysis of a mechanical system becomes imperative as it subjected to fail with time. Based on the operating system, this failure can be static (point load) or dynamic (cyclic stress) [20].

In the last few years, lots of scholars have actually made great deals of evaluation as well as investigates on the engine connecting rod by utilizing finite element method. Presently, most research generally concentrates on the both ends as well as body of the connecting rod [10], [11], [12], [13]. Among the major demands in connecting rod design, is making sure optimum stiffness and also toughness at the most affordable weight. It's needed to make the form and also dimension of connecting rod moderately as well as utilize high toughness product. [9]. Failure of connecting rods is as a result of its overloading, bearing failing, uneven changes of the screws as well as faulty setting up or fatigue [14]. It is important for connecting rods to be able to withstand the complex high tensile loads that acts on them. As a result, numerous design technology, material selection, working and fatigue test of a connecting rod have been studied and presented [15]. Mechanical properties (such as hardness, tensile strength, rigidity and fatigue resistivity) of the materials used in the manufacture of a connecting rod need vehicles depends on the design of the connecting rod. Failure of connecting rod is attributed to the in availability of much strength needed to hold the load. This can be overcome with the life cycle extended by increasing the strength [16]. A number of regular as well as unusual failing modes in connecting rods of combustion engines were reported in [17]. The writer's focus is concentrated on summary of failing modes as well as the stress evaluation of examined components. The analysis of the cracks was sustained by typical estimations as well as progressed logical designs. Webster was the first to carryout optimization of a connecting rod in the year 1983. The frequent need of an ideal connecting rod by the consumer basis the importance of optimization to produce a durable, lighter and cheaper component. [5]

FEA is the frequently utilized computational device for screening as well as changing design frameworks within particular design limitation. It entails diving in to tiny devices referred to as 'elements' for static and also dynamic analysis of straightforward to intricate version under various layout constraints. Further investigation can also be done to improve the design for optimal performance and lifespan with regards to design failure [21]. Several type of literary works have actually worked on weight optimization. Gaikwad in his paper modifies a roller conveyor by performing weight optimization after carrying out static analysis on the roller conveyor [22]. In [23], [24], theoretical and FEA of an IC engine connecting rod was conducted. The result of the analysis obtained shows the causes of failure at the fillet of both ends due to the induced stress. In [25] static FEA for fatigue, deformation and weight optimization of a connecting rod utilizing ANSYS workbench is performed and presented. From the suggested design changes obtained from the weight optimization result, the failure result is further updated to achieve a better result. In a paper by Bansal, dynamic stress analysis was carried out on a single cylinder four stroke diesel engine connecting rod of Aluminum material using FEA. The optimization was also done under dynamic loading with the boundary conditions and inputs determined from the pressure-volume diagram and engine specification chart respectively are carried out with different meshing size for an accurate result. [26]. In a paper by [27], exhaustion life evaluation of engine connecting rod is made use of to forecast the life of connecting rod, which will certainly assist in the design of an engine connecting rod in the future. In [28], with the ANSYS, stress, fatigue life cycle and factor of safety of connecting rod were examined by utilizing finite element method in 3D. The result reveal that the subjected harmful setting is the change place of small end and also connecting rod shaft at optimum compression condition. The paper by [29] review the evaluation of stress, distribution and also stress and anxiety experienced by the connecting rod of a combustion engine to be used on an electrical motor extender engine

The objective of the present work is to perform static structural analyses and optimization of connecting rod made of Steel material the shape optimization is performed with target weight reduction of 20 to 60% with an interval of 10 under a fixed loading of 30KN acting upon its bigger end. Analysis for structural optimization is additionally done to identify a brand-new optimized structure with deformation and stress values respectively. The analysis is accomplished in ANSYS static structural mechanical solver.

II. DESIGN SPECIFICATION

Being among one of the most essential components in an IC engines layout, the connecting rod should be able to endure a remarkable amount of load and also send a lot of power. The failure in a connecting rod can be one of the most damaging and costly failures in an engine [30]

The various forces acting on an engine connecting rod include: Force on the piston due to gas pressure (F_G), Force due to inertia of the connecting rod and reciprocating mass(F_I),), Force due to friction of the piston rings and of the piston (F_F), and Forces due to friction of the piston pin bearing and crank pin bearing (F_B), [31], [30]. In this study we will highlight on the force due to gas pressure and that due to inertia. The engine which the connecting rod is to be connecting has the configurations shown in table 1 based on the study by [31].

- The force due to gas pressure (F_g), is given by:

$$F_g = Force\ X\ area = \frac{\pi}{4} * d^2 X\ P_{max}$$

Where P_{max} is the maximum combustion pressure

d is the Piston pin diameter

$$F_g = \frac{\pi}{4} X (100 * 10^{-3})^2 X\ 3.15 * 10^6$$

$$F_g = 24740N$$

- Inertia Force Due to Reciprocating Mass

$$F_I = MAss\ X\ acceleration = M\omega^2 r(\cos\theta + r\cos\frac{\theta}{l})$$

$$F_I = 1756$$

Where $M =$ mass of (piston and rings + Piston pin + $\frac{1}{3}$ *connecting rod*

$\omega = angular\ speed\ rad/s$

$r = crank\ radius\ mm$

$l = length\ of\ connecting\ rod$ mm

It might be kept in mind that the inertia of reciprocating components opposes the pressure on the piston downward strokes. On the various other hand, the inertia force assists the piston force while relocating from the bottom to top

The net force $F_P =$ The force due to gas pressure (F_g) ± force due to inertia F_I

$$F_P = F_G \pm F_I$$

Or $F_P = F_G \pm F_I \pm W_R$. When the reciprocating mass $(W_R = M_R * g)$ is taken into consideration

$$F_P = F_G \pm F_I$$

$$F_P = 24740 + 1756 + 22.07$$

$F_P = 26518.07$N $\quad F_P = 22961.93$N

The force F_P generates a force F_C in the connecting rod as well as a thrust FN on the sides of the cylindrical tube wall surfaces. Therefore, we see that the force in the connecting rod is given by:

$$F_C = \frac{F_P}{\cos\theta} = \frac{F_P}{\sqrt{1 - \frac{(\sin\theta)^2}{n^2}}}$$

$n =$ ratio of length of connecting rod to radius of crank

Maximum force is derived when the connecting rod is at right angle to the crank leading to a major decrease in gas pressure. Owing to this, the force acting on a connecting rod is assumed to be same as the maximum exerted force on the piston from the gas pressure F_i without considering the inertia [30]

The loading conditions are presumed to be static [32], [33]. Two loading situations were examined for every case, one with the load of 30KN used at the crank end as well as limited at the piston pin end, and the other used at the piston pin end as well as limited at the crank end

Table 1: Functional specification of connecting rod engine

Speed of IC Engine	1800 RPM
Piston diameter (mm)	100mm
Mass of reciprocating parts	2.25 kg
Factor of safety	6
Young's modulus	2.1 X 105 MPa
Poisson's ratio	0.3
Density of material	8000kg/m
Wall pressure for piston rings (oil rings)	0.137 Mpa
Number of rings	3
Coefficient of friction	0.05
Combustion pressure	3.15 MPa
Piston pin diameter	29 mm
Crank pin diameter	44 mm
Connecting rod length (mm)	141

2.1 Static analysis of a connecting rod.

Finite Element Analysis is a mathematical technique using partial differential formulas for exploring and also fixing problems to an approximate precise solution [21]. Fixing design issues including complicated frameworks is a great feature of the FEM. ANSYS software application is a FEA software that creates formulas which addresses as well as regulates the conduct of the components. [18]. Modeling of the Connecting rod is done in the SolidWorks software. For the analysis to be carried out, the model is imported from the SolidWorks software as it provides a user-friendly two-way option for modelling of very complex structures. The 3D model of the connection rod with its dimensions is shown in figure 1 below.

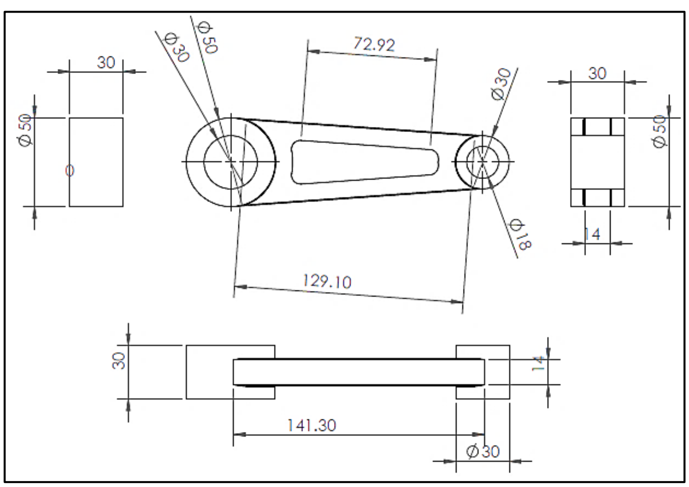


Figure 1: Connecting rod model

The geometry definition is initial performed depending upon the nature of evaluation that is to be done. The Analysis performed in this paper concentrates on stress and deformation. A 3D version can be imported in the ANSYS software application by either saving as an Initial Graphics Exchange Specification(IGES) design after which it is imported right into the ANSYS workbench, or by developing the whole framework in the ANSYS workbench [34] [35] [36]. In this paper, the analysis is done by importing the geometry from a CAD in the IGES format into the software.

For an efficient and qualitative analysis of material, the material properties which can be either linear or nonlinear, isotropic or orthotropic, constant or temperature dependent need to be accurately set. Based on the objective of the analysis, mechanical properties such as (density, strength and coefficient of thermal expansion definition is optional [37]. Recognizing as well as proclaiming the proper value of the property is extremely beneficial for the design evaluation function. The Young's modulus of a product conversely called modulus of elasticity is a mathematical constant that explains the flexibility as well as measures the ability of a solid to stand up to changes when exposed to stress or compression in a specific direction. The greater the Young’s modulus, the stiffer (i.e. exactly how it disperses under loads) is the structure which will certainly call for a much greater quantity of loads to deform. Poisson's ratio which is the proportion of compression to the expansion of a material along with Young's modulus (proportion of tension to strain) specifies the toughness as well as nature of just how a material framework deforms based upon a specific constrain. The deformation to due consistent volume as well as opposing forces are explained by the bulk as well as shear modulus respectively. Two various other vital aspects that figure out when some material losses their flexible behavior as well as the optimum stress are the yielding and tensile strength specifically. [38]. The structure material definition is done after the geometry is imported right into the software using the properties shown in Table 2.

Table 2: Table of Structural steel material properties

Young’s Modulus	200GPa
Density	7850 kg/m^3
Poisson’s ratio	0.30
Bulk modulus	1666.7GPa
Shear Modulus	76.923G Pa
Tensile strength	0.25GPa
Ultimate shear strength	0.46GPa

Breaking of the design or structure right into small aspects to evaluate each of the elements is called meshing [39]. It is a distinct realization of the structure, which assists in resolving the specific design solutions. The smaller the meshing size, the higher the computational time and accuracy of the analysis result [40]. The meshing size is set to default as determined by the ANSYS software. Taking into account the arrangement time as well as computational expense, with the rate and also simplicity of usage, a totally free mesh type is made use of. The default meshing control is used having a relevance value of +100 with a medium smoothing number of iteration. The meshed structure of the connecting rod as shown in figure 2. After the meshing process, another crucial and main step in the analysis is fixing support and loads in a way that is complying with the real-life circumstances [39], [34]. The boundary condition made use of in this evaluation are the fixed support and load. The support type used for the 3D structure is the fixed support that avoids movement or contortion of the framework geometry.

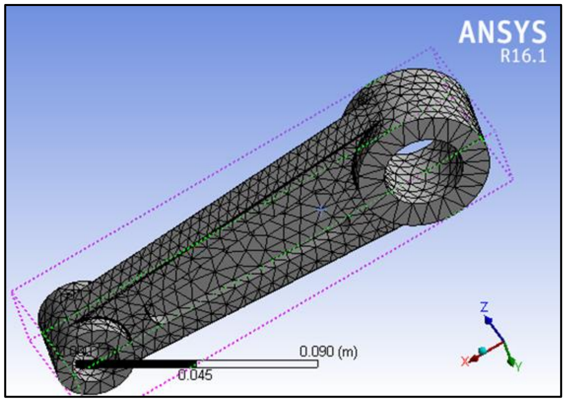


Figure 2: Meshed structure

2.2 Optimization of Connecting rod

Shape Optimization

To determine how and where failure occurs, and the way to prevent it, simulation in FEA is done. Failure of the whole or part of the system will result to the risk of life and financial loss. Just like in the human context, when the human body does much work, it becomes stressed, sick and finally, nervous breakdown may occur. Also, in an engineering structure, failure may occur when a structure is subjected to a high amount of stress. The amount of pressure in an engineering model that happens when it is exposed to external force or load is termed as stress, which indicates that the applied load is a function of the amount of stress [41].

In this paper, shape optimization via simulation in ANSYS workbench is used. The main value of optimization is figuring out a superb or ideal use of material within a collection of loadings as well as constraints by the predefined design. To accomplish a superb material distribution with efficient layout, the components that include the very least or absolutely nothing to the load bearing are determined and gotten rid of for weight decrease. The weight optimization for the connecting rod is carried out with target weight reduction of 20, 30, 40, 50, and 60% under the said constraints to determine the mass that needs to be removed to minimize cost. Furthermore, the deformation, stress, strain and factor of safety under the same loading condition was compared before and after 60% target weight reduction. The 3-D design of the rod is shown in Figure 3. A load force of 30KN is applied at the crank end of the connecting rod while the pin end is subjected to a fixed support.

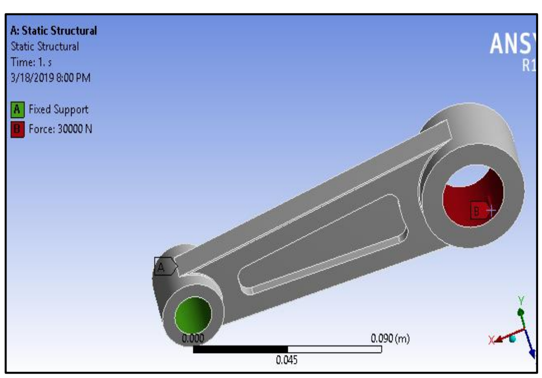


Figure 3: Constrains Application

Structural optimization

Upgrading of an engineering framework after failure is commonly done based upon an experimentation technique up until a criterion is reached. This procedure is lengthy, incorrect, as well as ineffective. With the current breakthrough evaluation approaches as well as software applications, mathematical optimization strategies are made use of to stabilize such compromises. To identify an optimum measurement or form of a design framework, parametric optimization is utilized to boost the layout by transforming the values of the variable criteria (such as density, sizes, and so on). Generally, for a specific design constraint, parametric optimization is the decrease of the layouts details [42]. In this analysis, the parameters for total deformation, equivalent stress and safety factor are set. A designer uses Von-Mises stress analysis to ascertain the failure of his design structure. Failure is inevitable when the strength of the material used is less than the maximum value of the stress. Von-Mises stress is ideally used for ductile materials. It becomes necessary to declare this parameter prior to the simulation in the material properties [43].

A Response Surface Optimization system is included in the task schematic with the lower as well as top bound of the rod bearing diameter, pin end diameter and also the connecting rod set to a new value as displayed in table 3. A response surface is acquired from the finite element simulation after the sampling of the design space through Centre Composite Design (CCD). Interpolation models created from the simulation (response datasets) gives a variation of these responses in terms of the design variables. Figure 4 is the solid mass response chart. Design of Experiment is made use of in the optimization to fit the information of the simulated design response right into models called reaction surface area equation [21]. In a design space, the layout variables are seen as one measurement having a great deal of distinct levels.

Table 3: Parameters range settings

Structure/Original Value	**Rod bearing Diameter 50**		**Rod small end diameter 30**		**Connecting rod length 142**	
Value	Before	After	Before	After	Before	After
lower Bound	45	35	27	17	127.8	117.8
Upper Bound	55	45	33	23	156.2	136.2

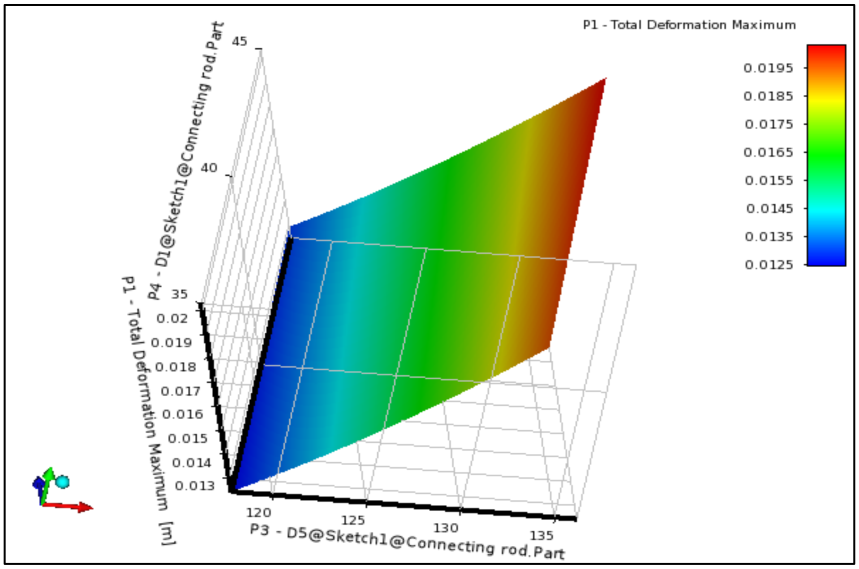
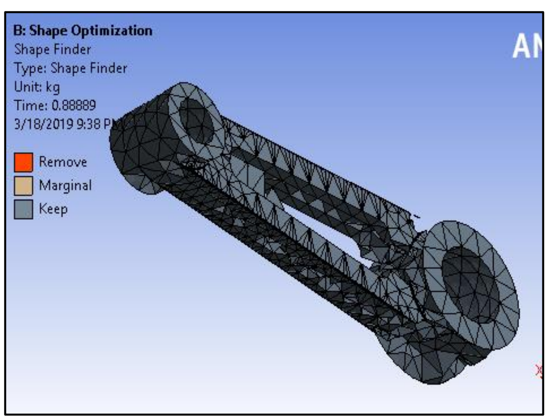


Figure 4: Total deformation response Chart

III. RESULT

3.1 Result for Shape optimization

Weight reduction is needed in Connecting rod in order to reduce the weight that is needed in construction which minimizes the cost by saving the amount of material. Figure 5 shows the weight optimization result of the connecting rod which is plainly recommended as the perfect design by the geography optimization study aiming at 80%. weight decrease. Materials are gotten rid of from the least stressed out areas in the simulation design, that is, areas adding the least to the total stiffness of the framework. This optimization research study has clarified why the connecting rod shape remains that way playing an essential component after many years of usage.



(a)

Figure 5: Optimized structure with 60% target weight reduction

The weight optimization result for the various percentages is tabulated in table 4. Whereas figure 6 is the graphical representation of the behavior. It can be seen that the optimized and marginal mass exhibit a non-linear and linear relationships respectively. As expected, the optimized mass has a negative linear plot as the percentage of reduction is increased, whereas the marginal mass obeys quadratic form

Table 4: Weight reduction

Target reduction (%)	Marginal mass (kg)	Optimized mass (kg)
20	2.64E-03	0.68493
30	4.26E-03	0.63167
40	3.69E-03	0.53215
50	3.69E-03	0.53215
60	2.51E-03	0.4822

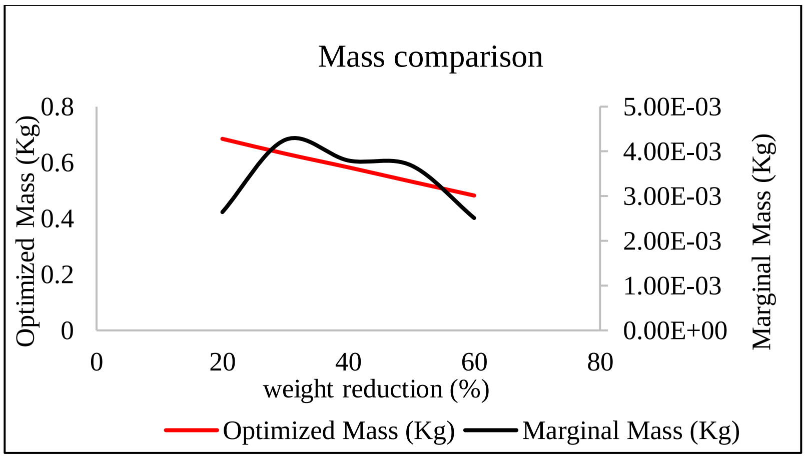


Figure 6: Mass comparison

3.2 Result for structural Optimization

The DOE samples were evaluated successfully showing that the designs sampled are meaningful in response to the change in geometry. Table 5 listed below the created DOE table for various layout circumstances that competes the 6 inputs.

Table 5: Design of Experiments

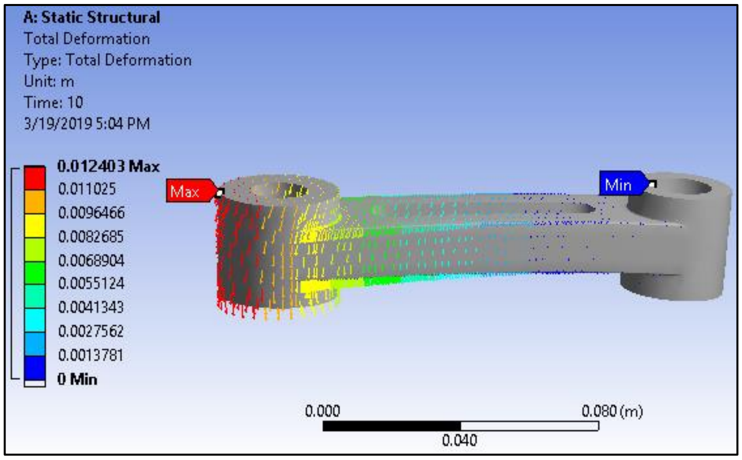
Design of Experiment (Central Composite Design: face-centered: standard							
1	Name	P4-D1 Connecting rod. part	P5-D2 connecting rod .part	P3-D5 Connecting Rod part	P1- Total deformation Maximum (m)	P2- Equivalent stress maximum (Pa)	P6- Equivalent Elastic Strain maximum (mm^-1)
2	1 [DP 8]	40	20	127	0.016025	4.71E+09	0.024164
3	2 [DP 3]	40	20	117.8	0.012667	4.86E+09	0.024332
4	3 [DP 13]	40	20	136.2	0.019943	6.16E+09	0.030895
5	4 [DP 6]	35	20	127	0.015741	6.33E+09	0.031752
6	5 [DP 10]	45	20	127	0.016322	5.73E+09	0.028742
7	6 [DP 7]	40	17	127	0.016015	5.21E+09	0.026192
8	7 [DP 9]	40	23	127	0.016043	5.27E+09	0.026456
9	8 [DP 1]	35	17	117.8	0.012403	4.61E+09	0.023539
10	9 [DP 11]	35	17	136.2	0.019577	6.91E+09	0.034653
11	10 [DP 4]	45	17	117.8	0.012883	4.22E+09	0.021211
12	11 [DP 14]	45	17	136.2	0.020284	6.32E+09	0.031696
13	12 [DP 2]	35	23	117.8	0.012449	5.43E+09	0.02725
14	13 [DP 12]	35	23	136.2	0.01962	6.42E+09	0.032215
15	14 [DP 5]	45	23	117.8	0.012897	4.56E+09	0.022956
16	15[DP 15]	45	23	136.2	0.020303	5.39E+09	0.02715

The objective and constraints are set up in the optimizer. The part that fulfils all the design constraints are identified using the fitted response models. Decision of ideal candidate was done by finding the entire design space region for the much better value that matches to the objective function. The ideal 3 prospects for the layout are detailed as revealed. Table 6 indicates the optimization and candidates point windows.

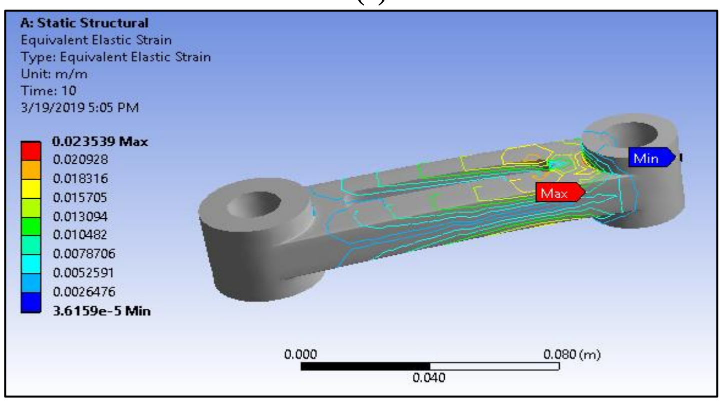
Table 6: Optimized parameters Candidate

	Candidate Point 1	candidate Point 2	Candidate Point 3
P1-D1-Connecting rod part	44.365	44.905	44.755
P2-D2- connecting rod part	17.513	19.903	22.716
P3-D5-Connecting rod part	199.56	119.21	119.89
P4-Total Deformation maximum (m)	0.00010079	0.0001034	9.68E-05
P5- Equivalent Stress Maximum (Pa)	6.02E+07	6.10E+07	6.42E+07
P6- Safety Factor Minimum	4.1126	4.0732	3.8964

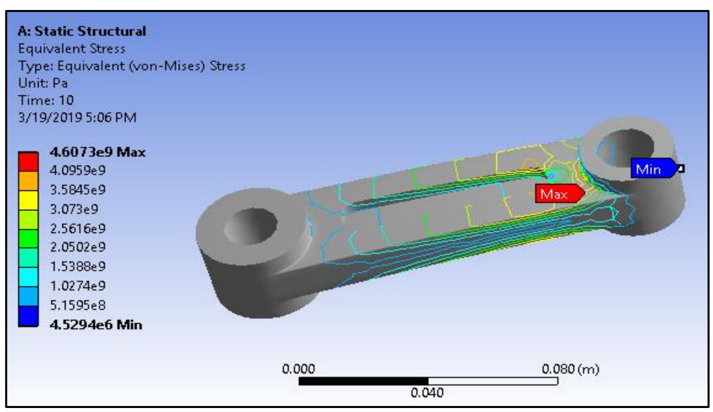
The chosen design point (DP) changed the current parameters, and the framework is upgraded with the brand-new values. Figure 7 are the brand-new optimized frameworks revealing the overall deformation and also equivalent stress and elastic strain. A tabular contrast of the parameters is summarized in table 7, whereas figure 8a, 8b and 8c are the graphical comparison of the before and after optimization. It appears that all the three parameters improve after optimization and exhibit a positive linear relationship with loading time.



(a)



(b)

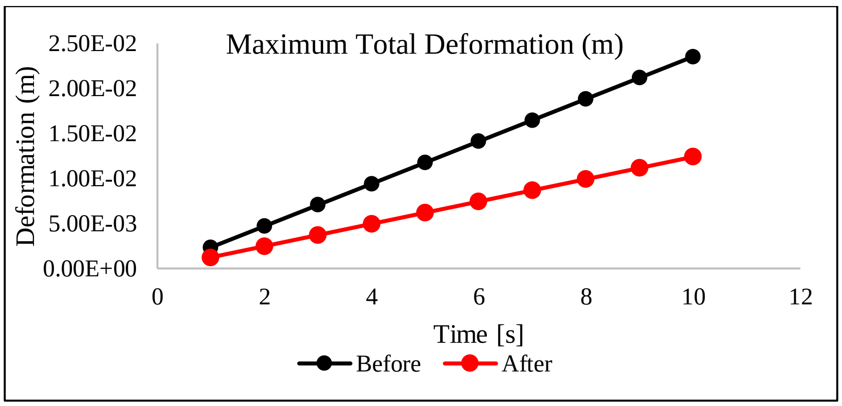


(c)

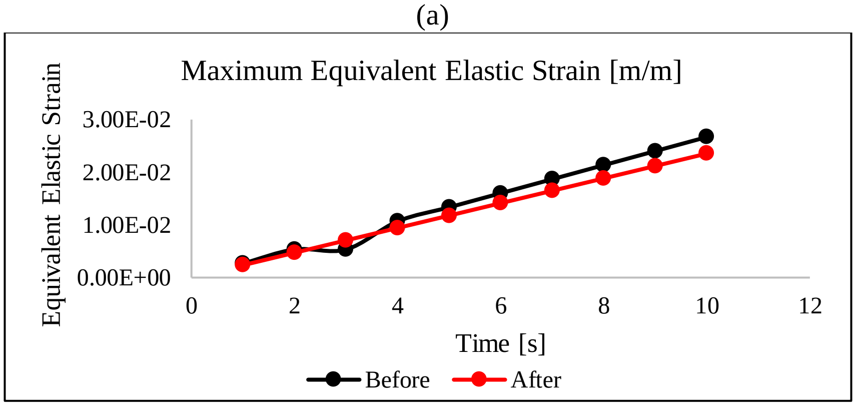
Figure 7: Updated design (a) Total Deformation. (b) Elastic Strain. (c) Equivalent stress

Table 7: Structural optimization analysis comparison

	Maximum Total Deformation (m)		Maximum Equivalent Elastic Strain [m/m]		Maximum Equivalent Stress [Pa]	
Time [s]	Before	After	Before	After	Before	After
1	2.35E-03	1.24E-03	2.67E-03	2.35E-03	5.30E+08	4.61E+08
2	4.71E-03	2.48E-03	5.34E-03	4.71E-03	1.06E+09	9.21E+08
3	7.06E-03	3.72E-03	5.34E-03	7.06E-03	1.59E+09	1.38E+09
4	9.42E-03	4.96E-03	1.07E-02	9.42E-03	2.12E+09	1.84E+09
5	1.18E-02	6.20E-03	1.33E-02	1.18E-02	2.65E+09	2.30E+09
6	1.41E-02	7.44E-03	1.60E-02	1.41E-02	3.18E+09	2.76E+09
7	1.65E-02	8.68E-03	1.87E-02	1.65E-02	3.71E+09	3.23E+09
8	1.88E-02	9.92E-03	2.13E-02	1.88E-02	4.24E+09	3.69E+09
9	2.12E-02	1.12E-02	2.40E-02	2.12E-02	4.77E+09	4.15E+09
10	2.35E-02	1.24E-02	2.67E-02	2.35E-02	5.30E+09	4.61E+09



(a)



(b)

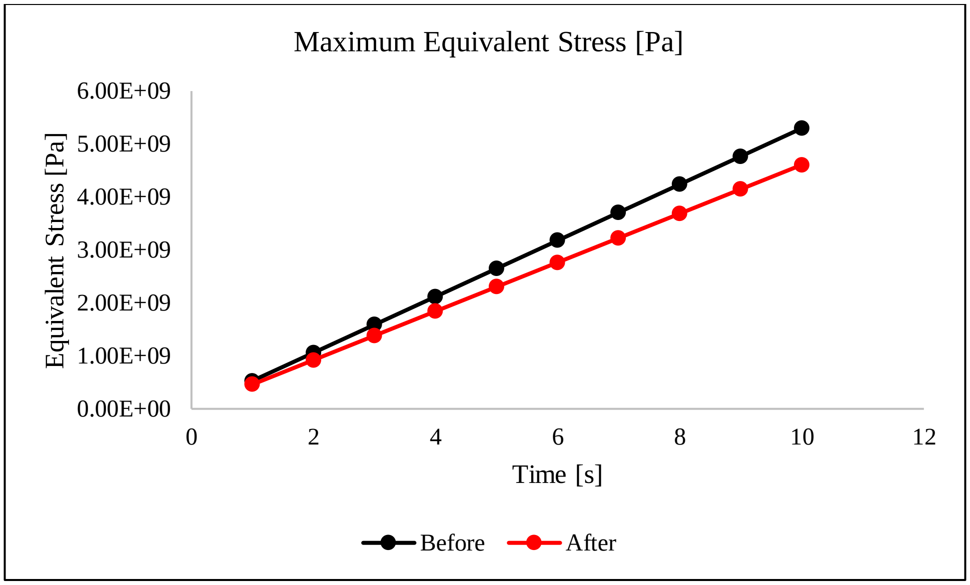


Figure 8: Comparison(a) Deformation (b) Elastic strain (c) Equivalent stress

IV. CONCLUSIONS

In any manufacturing process, product quality and cost are the two critical, essential factors that need to be carefully selected by the company to maximize profit without compromising the product quality. This paper demonstrates the structural and optimization analysis of a connecting rod suitable for diesel engine applications. The structural analysis assesses the endurance of the connecting rod when subjected to static load along with the deformation characteristics. Weight and structural optimization of the connecting rod together through Finite Element Method using ANSYS is carried out and presented. The processes were performed under a loading of 30KN static force on a connecting rod of structural steel material. Further comparison of the analysis result before and after the optimization is done to ascertain the available optimal design.

The result obtained shows that the initial structure before optimization experience a relatively high deformation, stress and strain values respectively. This improvement results in less deformation in the structure after the optimization. The weight optimized structure has fewer steel materials because unwanted portions that have no beneficial effect on the structure has been removed, thereby reducing the weight Hence, reduces material cost and wastages to the production industry. Finally, based on the results obtained, it can be concluded that ANSYS software package is an effective and efficient tool suitable for performing analysis and optimization of various engineering structures.

REFERENCES

[1] Gopinath D., Sushma Ch. V. "Design and Optimization of Four Wheeler Connecting Rod Using Finite Element Analysis." In 4th International Conference on Materials Processing and Characterization. India: Materials Today: Pages 2291-2299, 2015. India

[2] Uma S. V., Maheswara R., Rao T. V., Satyanarayana K., & Nagaraju B., "Fatigue Analysis of Sundry I.C Engine Connecting Rods." In 7th International Conference of Materials Processing and Characterization. Hyderabad: 2017, India.

[3] Heywood J., "Internal Combustion Engine Fundamentals". McGraw-Hill Education: 1 edition, 1998.

[4] Shenoy P., "Dynamic load analysis and optimization of connecting rod." US: (M.S. thesis). The University of Toledo, 2004

[5] Shaari M. S., Rahman M.M., Noor M.M., Kadirgama K., Amirruddin A. K. "Design of connecting rod of internal combustion engine: A topology Optimization approach." In National Conference in Mechanical Engineering Research and Postgraduate Studies (2nd NCMER 2010): Pages 155-166, 2010, Pahang, Malaysia: Universiti Malaysia Pahang.

[6] Wang Y., "FEA for Connecting Rod of Engine Based on AWE." Journal of Machinery Design and Manufacture: 7, 224-226, 2013

[7] Zheng B., Ji L., Liu Y., "Finite Element Analysis and Structural Improvement of Diesel Engine Connecting Rod." In International Conference on Computer Modeling and Simulation: Pages 175-178, 2010, Egypt: IEEE.

[8] Zhibo Y., Pengfei L., Kaifei L., "Optimal Design of Engine Connecting Rod Based on FEM." Journal of Henan Polytechnic University(Natural Science): 32(1), 65-68, 2013

[9] Jing Z., Zhiyuan Q., & Yan Z., "Finite Element Analysis of EA 113 Gasoline Engine Connecting Rod." International Journal of Mechanical Engineering and Applications: 5(4), 208-213, 2017

[10] Yirong X., Tenggang X., Jianjun Z., "Optimization Design of Engine Connecting Rod Based on ANSYS Workbench." Journal of Donghua University (Natural Science): 41(4), 527-531, 2015

[11] Hu X., "Mechanical Properties Analysis of Moto car Engine Connecting Rod Based on ANSYS Workbench." Journal of Manufacturing Automation: 4, 107-109, 2014

[12] Tengteng L., Shaohua Z., "Modal Analysis of Engine Connecting Rod Based on ANSYS." Journal of Automotive Engineering: 4, 25-27, 2010

[13] Slavko R., Ugljesa B., Igor R., Zeljko B., "Failure analysis of a special vehicle engine connecting rod." Engineering failure analysis: 79, 98-109, 2017

[14] Ilman M, Barizy R. "Failure analysis and fatigue performance evaluation of a failed connecting rod of reciprocating air compressor." Engineering Failure Analysis: 56,142-149, 2015.

[15] Mohammed M. H. A., Mohamed H., "Analysis of Fatigue Stresses on Connecting Rod Subjected to Concentrated Loads At The Big End" Materials Today: Proceedings:2 (4-5), 2094 – 2103, 2015

[16] Gopinath D., Sushma Ch. V. "Design and Optimization of Four Wheeler Connecting Rod Using Finite Element Analysis." Materials Today: Proceedings: 2(4-5), 2291 – 2299, 2015

[17] Strozzi A., Baldini A., Giacopini M., Bertocchi E., & Mantovani S, "A repertoire of failures in connecting rods for internal combustion engines and indications on traditional and advanced design methods", Engineering failure analysis: 20-39, 2016

[18] Gopal G., Suresh L. K, Vijaya Bahskar R., Maheshwara RaoU. M., & Srinivasulue G., "Analysis of Piston, Connecting rod and Crank shaft assembly." Materials Today: Proceedings:4(8) ,7810–7819, 2017

[19] Aisha M., Ibrahim H. S., "Strength analysis and structural optimization of an l-shaped bracket." Nigerian Journal of Technological Research (NJTR) 13(2), 14-19, 2018

[20] Orhorhoro E. K., Erhinyodavwe O., Ikpe A. E., "Failure Analysis of Automatic Access Pedestrian Gate Turnstile using SolidWorks Simulation Model." International Journal of Engineering Technology and Sciences 8(1) 2017

[21] Xiaolin C., Yijun L., “Finite Element Modeling and Simulation with ANSYS Workbench”, 2014 London NewYork: CRC Press
[22] Gaikwad S. S., Aitavade E. N., "Static analysis of roller of gravity roller conveyor for structure strength & weight optimization." IJAET: 4, 2013
[23] Vivek. C. P., Bhumeshwar P., Ajay N. I., "tress Analysis of I.C. Engine Connecting Rod by FEM." International Journal of Engineering and Innovative Technology: (IJEIT) 1(3), 12-15, 2012
[24] Lucjan W., Paweł Z., "Stress and failure analysis of the connecting rod of diesel engine." Engineering Failure Analysis: 97, 374-382, 2019
[25] Pushpendra Kumar S., Borse Rajendra R. "fatigue analysis and optimisation of connecting rod using finite element analysis." International Journal of advance research in Science and Engineering (IJARSE): 1(1), 3367-3371, 2012
[26] Bansal R, "Dynamic simulation of connecting rod made of aluminum alloy using finite element analysis approach." IOSR Journal of Mechanical and Civil Engineering (IOSR): 5(2), 1-5, 2013
[27] Dan Y., Zhen Y., Wentao C., & Leilei Z. "Fatigue Analysis of Engine Connecting Rod Based on Workbench." In 7th International Conference on Mechatronics, Computer and Education Informationization (MCEI 2017): Pages 192-196, 2017, China: Advances in Computer Science Research
[28] Zheng B., Liu Y., Ji L., "Finite Element Analysis and Structural Improvement of Diesel Engine Connecting rod." In 2010 Second International Conference on Computer Modeling and Simulation: 2010, Sanya, Hainan, China: Los Alamitos, Calif.: IEEE Computer Society Press.
[29] Suherman, Bambang W., Achmad P., Widodo B. S., & Arifin N., "Modeling and Static Analysis of a Connecting Rod in Range Extender Engine." In 2016 International Conference on Sustainable Energy Engineering and Application (ICSEEA): 17-21, 2016, Jakarta, Indonesia: IEEE
[30] Rao G., Naga M., "Design Optimization and Analysis of a Connecting Rod using ANSYS." International Journal of Science and Research (IJSR): 2(7) 225-229, 2013.
[31] Puran S., Debashis P., & Ran V. S., "Fatigue and Structural Analysis of Connecting Rod’s Material Due to (C.I) Using FEA." International Journal of Automotive Engineering and Technologies: 4(4) ,245– 253, 2015
[32] Sudershn K. K., Tirupathi R. K., & Syed A. H., "Modeling and Analysis of Two Wheeler Connecting Rod." International Journal of Modern Engineering Research (IJMER): 5(2), 2012.
[33] William D. W., Roy C., Dave A., "Three Dimensional Finite Element Analysis of a High Speed Diesel Engine Connecting Rod." SAE Technical, 1983
[34] Wang G., Zhou G., Yang J., Guan Y., Gong W., & Zhang H., "Modal analysis of high-speed spindle based on ANSYS." In 2012 7th International Conference on Computer Science & Education (ICCSE): Pages 475-478, 2012, Melbourne, VIC, Australia : IEEE.
[35] Kim B. S., Lee S. H., Lee M. G., Ni N., Song Y. J., & Lee C. W, "A comparative study on damage detection in speed-up and coast-down process of grinding spindle-typed rotor-bearing system." Journal of Materials Processing Technology:187-188, 30-36, 2007
[36] Janq G. H., Lee S. H., & Jung M. S., "Free vibration analysis of a spinning flexible disk- spindle system supported by ball bearing and flexible shaft using the finite element method and substructure synthesis." Journal of Sound and Vibration: 251(1), 59-78. 2002
[37] Barbero, Ever J. “Finite Element Analysis of Composite Materials Using ANSYS”. US: CRC press, Tylor and Francis group: 2014.
[38] Nipun, “Difference Between Yield Strength and Tensile Strength, Retrieve October 14. 2015 from. <http://pediaa.com/difference-between-yield-strength-and-tensile-strength/>.
[39] Basavaraj T., Kurbet S. N., Kuppast V. V., Arvind M. "Harmonic analysis of a two-cylinder crankshaft using ANSYS." In International Conference on Inventive Computation Technologies (ICICT). Pages 1-6, 2016: Coimbatore, India: IEEE

[40] Qiongying L., Yushi M., "Modal analysis of a magnetic climbing wall car frame based on the ANSYS." 2014 IEEE Workshop on Electronics, Computer and Applications (IWECA). Ottawa, ON, Canada, pages 938-940, 2014, IEEE

[41] Aisha M., Ibrahim H. S., "Structural Analysis of a Knuckle Joint using different materials." In 1st International Civil Engineering Conference (ICEC 2018), Pages 100-106, 2018, Nigeria: FUTMinna

[42] Smith C., "Automated Design". 2011 ANSYS, Inc., 2012.

[43] Aisha M., Ibrahim H. S., "Finite Element Analysis of a base stand using different materials." Science Technology and Arts Research Journal: 1-7. 2019

[44] Jing Z., Zhiyuan Q., & Yan Z., "Finite Element Analysis of EA 113 Gasoline Engine Connecting Rod." International Journal of Mechanical Engineering and Applications: 5 (4), 208-213, 2017