Available online at www.sciencedirect.com

ScienceDirect

Materials Today: Proceedings 22 (2020) 1600–1609

materialstoday: PROCEEDINGS

www.materialstoday.com/proceedings

ICMMM 2019

Design optimization of a diesel connecting rod

Aisha Muhammad[a,c]*, Mohammed A. H. Ali[a] Ibrahim Haruna Shanono[b,d]

[a]Faculty of Manufacturing Engineering, Universiti Malaysia Pahang, Malaysia
[b]Department of Electrical of Engineering, Universiti Malaysia Pahang, Malaysia
[c]Department of Mechatronics Engineering, Bayero University Kano Nigeria
[d]Department of Electrical Engineering, Bayero University Kano Nigeria

Abstract

Connection rods are mechanical components used for generating motion from a crankshaft's piston alternating motion. The credibility and performance of vehicles depends on the design of the connecting rod. There have been numerous reported cases of connecting rod failure based on the structural design, loading type and the type of materials used in its production. To insure safety and satisfies customers demand in automotive industries a robust and optimized connecting rod is required. The aim of this paper is to carryout topology and structural optimization of a connection rod suitable for diesel engine applications. Weight optimization of the connection rod is carried out with target weight reduction of 20%, 30%, 40%, 50%, and 60% under a static loading of 100N, which determines the mass that needs to be remove to minimize both weight and cost without compromising it reliability and durability. Furthermore, the deformation, stress, strain and factor of safety under the same loading condition was compared before and after a 60% target weight reduction. Under a loading condition of 500N, analysis for the structural optimization is done to determine an optimized structure with new deformation and Von-misses stress, equivalent elastic strain, and safety factor values. Comparison of the static structural deformation, Von-misses stress, elastic strain and safety factor before and after the optimization is carried out. 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.

Peer-review under responsibility of the scientific committee of the 2nd International Conference on Materials Manufacturing and Modelling, ICMMM – 2019.

Keywords: ANSYS, connecting rod, FEA, Deformation, stress analysis;

* Aisha Muhammad. Tel.: +2348032684794
E-mail address: ayshermuhd@gmail.com

Peer-review under responsibility of the scientific committee of the 2nd International Conference on Materials Manufacturing and Modelling, ICMMM – 2019.

Nomenclature

P1	Rod bearing Diameter
P2	Rod small end diameter
P3	Connecting rod length

1. INTRODUCTION

Rotary motion is generated from a crankshaft's piston alternating motion using a connecting rod. The engine combustion gases and components motion of inertia exert pressures that induces compressive and tensile stress in the connecting rod respectively [1], [2]. Connecting rods fail due to its overloading, bearing failure, irregular adjustments of the bolts and faulty assembly or fatigue [3]. 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 [4]. 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 [5].

Finite element analysis of connecting rod have been conducted and presented by a lot of researchers. In [6], 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 [7] static FEA for fatigue, deformation and weight optimization of a connecting rod using ANSYS workbench is carried out 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 [8].

FEA is the commonly employed computational tool for testing and modifying engineering structures within certain design limit. It involves diving in to small units known as 'elements' for static and dynamic analysis of simple to complex model under different design constraints. Further investigation can also be done to improve the design for optimal performance and lifespan with regards to design failure [9].

Many kinds of literature have 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 [10] In another paper by [10] the weight of a roller conveyor was reduced thereby saving the materials under specific load constraints using finite element method.

In this paper shape optimization with target weight reduction rate of 20 to 60% with an interval of 10 under a static loading of 100N. Further analysis for structural optimization is also done to determine a new optimized structure with new deformation and stress values respectively. The analysis is carried out in ANSYS static structural, mechanical solver with a tensile force of 100N acting on its larger end.

2. METHODOLOGY

2.1 *Modeling of the model*

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 structures. The 3D model of the connection rod with its dimensions is shown in figure 1 below.

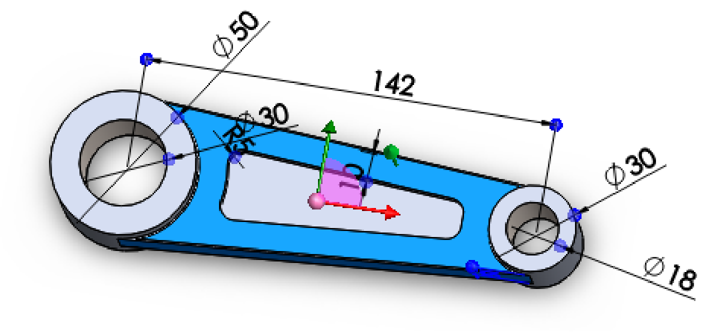


Fig. 1. Connecting rod model.

2.2 *Finite Element Analysis*

Finite element Analysis is a numerical approach described by partial differential equations for investigating and solving problems to its approximate exact solution [9]. Solving engineering problems involving complex structures is a good attribute of the FEM. ANSYS software is a FEA software package that generates equations which solves and controls the conducts of the elements. [11].

The geometry is definition is first carried out depending on the nature of analysis to be conducted. A 3D model can be introduced in the ANSYS software by either saving it as an Initial Graphics Exchange Specification(IGES) format then import it into the ANSYS workbench, or by building the entire structure in the ANSYS workbench [12] [13] [14]. In this paper, the analysis is done by importing the geometry from a CAD in the IGES format into the software.

For this analysis, the material used is structural with material properties as shown in table 1 below

Table 1: Table of Structural steel material properties

Young's Modulus	2E+11 Pa
Density	7850 kgm^-3
Poisson's ratio	0.3
Bulk modulus	1.6667E+11Pa
Shear Modulus	7.6923E+10 Pa
Tensile strength	2.5E+08 Pa
Ultimate shear strength	4.6E+08 Pa

Depending on the aim of the analysis, some mechanical properties such as density, strength and coefficient of thermal expansion definition is optional [15]. Knowing and declaring the correct value of the material property is very useful for design analysis purpose. Material deformation to due uniform volume and opposing forces are described by the bulk and shear modulus respectively Two other essential properties that determine when the material losses its elastic behaviour and the maximum stress a material can undergo are the yields and tensile strength respectively. [16].

Meshing involves breaking the model or structure into tiny elements to analyse each of the components is known as meshing [17]. It is a discrete realisation of the structure, which helps in solving the exact model solutions. The smaller the meshing size, the higher the computational time and accuracy of the analysis result [18]. Meshing tools in ANSYS can be classified under: [12], Unit size control, Level control of the intelligent division. Thinning grid control. Shape settings of meshing, Grid partition. The meshed structure of the connecting rod as shown in figure 2.

After meshing the model, constraints (fixed support and loads) set in a manner conforming to the real-life situations are applied this is very important and serves as a primary step required in the analysis [17], [12]. The boundary condition used in this analysis include support and loads. Figure 3 shows the applied load force of 100N and 500N at the small end of the connecting rod for the shape and structural optimization analysis respectively while the bigger end subjected to a fixed support.

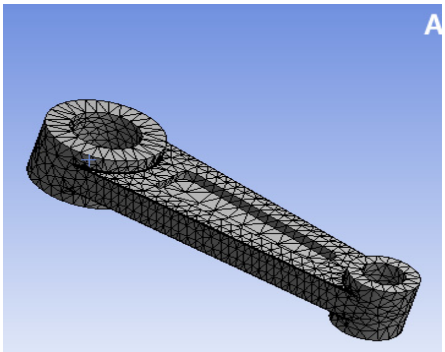
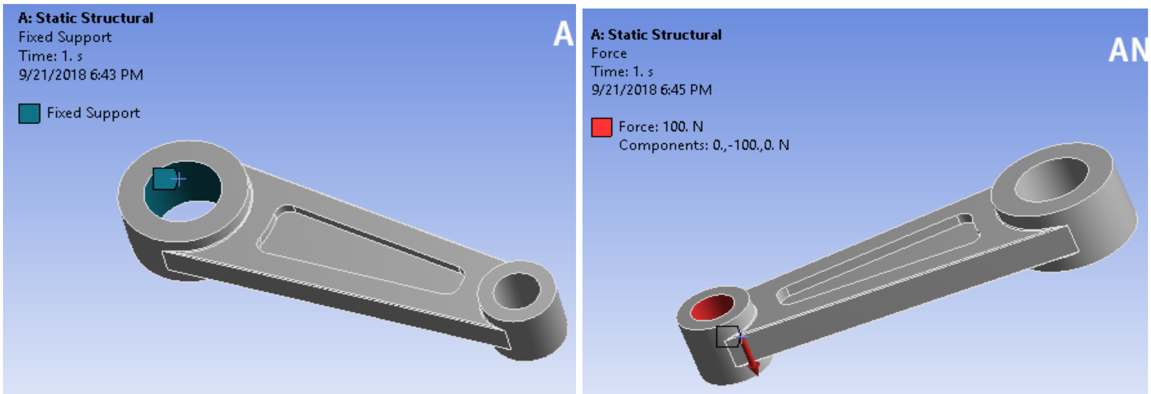


Fig. 2: Meshed structure



(a) (b)

Fig. 3 (a) Fixed support application (b) Shape optimization Load application

2.3 Shape optimization

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 [19].

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. It indicates that the stress of a point in a structure or model is higher than the material strength. The calculation of safety factor involves the yield strength; therefore, it becomes necessary to declare this parameter prior to the simulation in the material properties [20].

The main aim of optimization is minimizing the mass and cost with the load range of the connecting rod. 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.

2.4 Structural optimization

Redesigning of an engineering structure after failing is traditionally done based on a trial and error approach until a benchmark is reached. This process is time-consuming, inaccurate, and inefficient. With the recent advance analysis methods and software, numerical optimization techniques are used to balance such trade-offs. In general, for a particular design constraint, parametric optimization is the reduction of the designs specific objective function [21]. In this analysis, the parameters for total deformation, equivalent stress and safety factor are set.

In this paper, structural optimization via simulation in ANSYS workbench is used. A Response Surface Optimization system is added to the project schematic with the lower and upper bound of the rod bearing diameter, small end diameter and the connecting rod set to a new value as shown in table 2.

Table 2: 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

Design of Experiment is used in design optimization to fit the data of the simulated design response into models called response surface equations [9]. In a design space, the design variables are seen in a single dimension having a lot of discrete levels. A DOE table is then generated for the six inputs.

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 5 is the solid mass response chart.

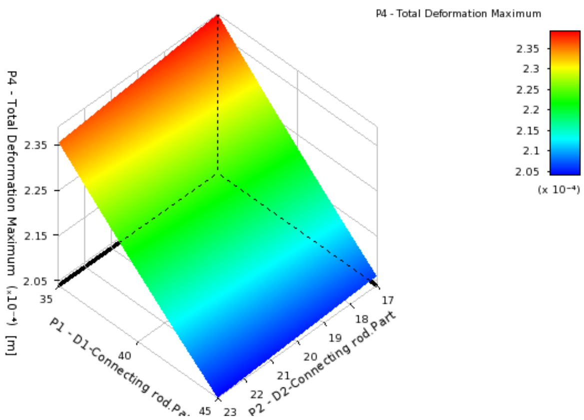


Fig. 5: Response Chart

3. RESULT AND DISCUSSION

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 6 shows the result of the weight optimization using a target weight reduction of 20 to 60% in an increment of 10 respectively under the constraints of a 100N static force.

Fig. 6: Weight Optimization (a) 20%. (b) 30%. (c) 40%. (d) 50%. (e) 60%

The weight optimization result for the various percentages is summarized in a graphical representation as shown in figure 7.

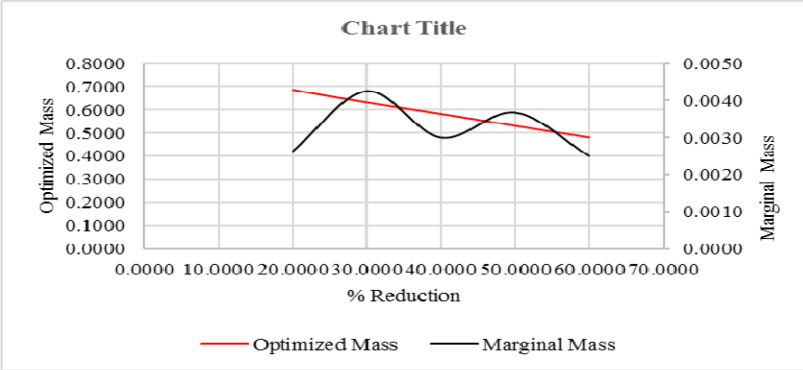
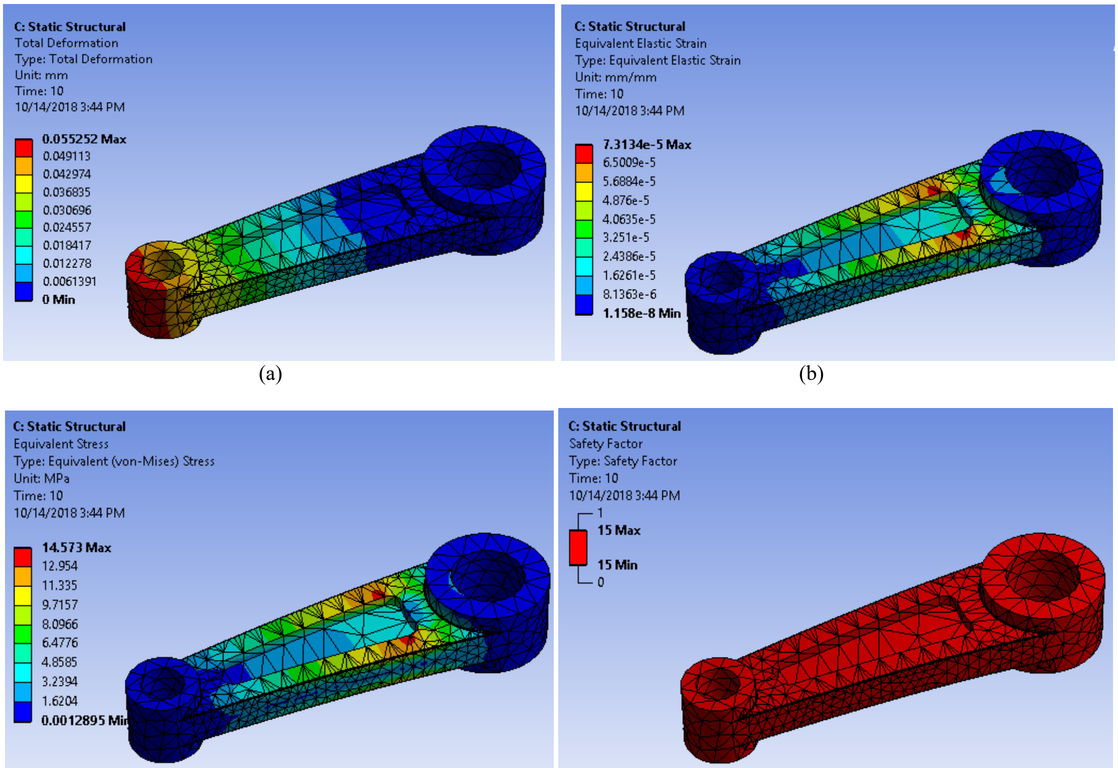


Fig. 7: Mass comparison

Figure 8 shows the deformation, stress, strain and safety factor after optimization with a target weight reduction of 60%.



(c) (d)

Fig. 8: 60% Weight Optimization (a) Total Deformation. (b) Equivalent stress. (c) Elastic Strain (d) Safety factor

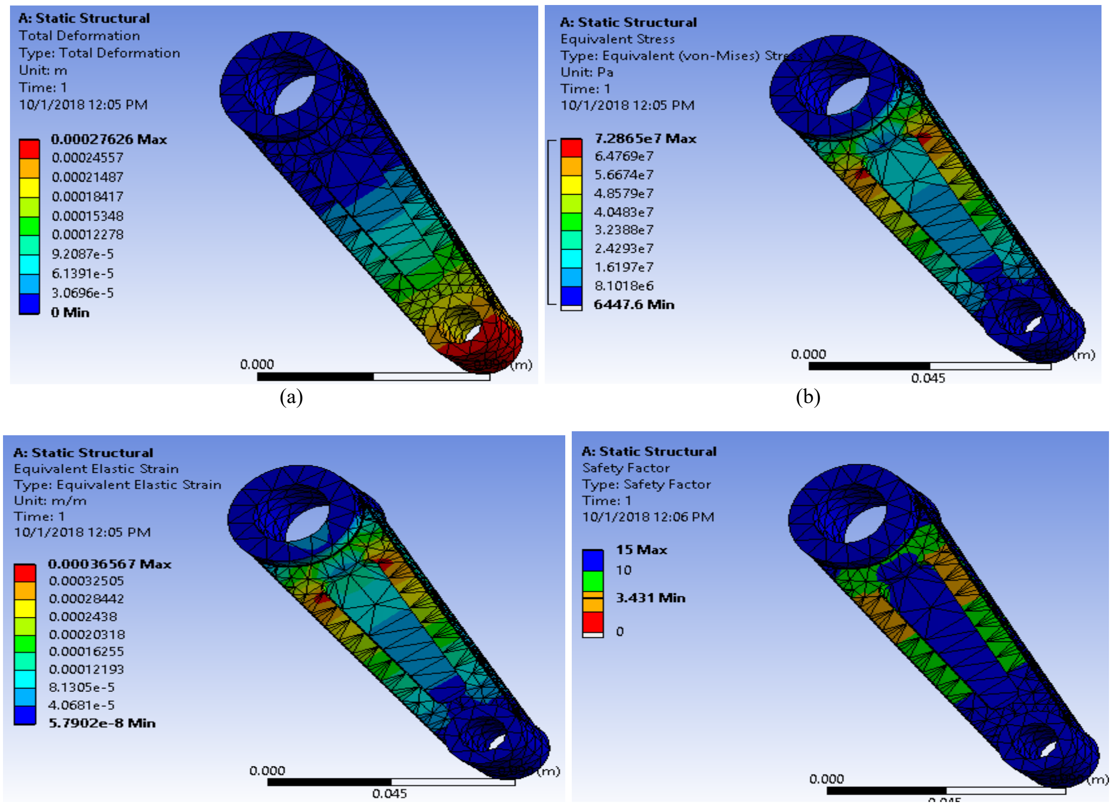
Table 3 below shows the value of the deformation, stress, elastic strain and safety factor before and after the optimization process with a target weight reduction of 60%. The analysis result is seeming to have improved after the optimization process.

Table 3: Weight optimization Analysis result comparison

Time[s]/ Analysis	Total deformation [mm]		Maximum Equivalent stress [MPa]		Elastic strain [mm/mm]	
	Before Opt.	After Opt.	Before Opt.	After Opt.	Before Opt.	After Opt.
0.1	0.005553	0.005525	1.6195	1.4573	0.0000081325	0.0000073134
0.2	0.011107	0.011050	3.2390	2.9146	0.0000162650	0.0000146270
0.3	0.016660	0.016576	4.8585	4.3719	0.0000243970	0.0000219400
0.4	0.022213	0.022101	6.4781	5.8292	0.0000325300	0.0000292540
0.5	0.027767	0.027626	8.0976	5.8292	0.0000406620	0.0000365670
0.6	0.033320	0.033151	9.7171	8.7438	0.0000487950	0.0000438800
0.7	0.038873	0.038677	11.3370	10.2010	0.0000569270	0.0000511940
0.8	0.044427	0.044202	12.9560	11.6580	0.0000650600	0.0000585070
0.9	0.049980	0.049727	14.5760	13.1160	0.0000731920	0.0000658200
1.0	0.055533	0.055252	16.1950	14.5730	0.0000813250	0.0000731340

3.2 Result for structural Optimization

Before performing the shape optimization analysis, the static structural analysis for deformation, Von-mises stress, elastic strain and safety factor under the loading of 500N force is carried out. Figure 9 shows the analysis result.



(c) (d)

Fig. 9: (a) Total Deformation; (b) Equivalent stress; (c) Elastic Strain; (d) Safety factor

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. Determination of best candidates was done by locating the whole design space region for the better value that corresponds to the objective function. The best three candidates for the design are listed as shown. Figure 10 indicates the optimisation and candidates point windows

Candidate Points	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	119.56	119.21	119.89
P4 - Total Deformation Maximum (m)	★★★ 0.00010079	★★★ 0.0001034	★★★ 9.6784E-05
P5 - Equivalent Stress Maximum (Pa)	★★★ 6.0225E+07	★★★ 6.1006E+07	★★ 6.4236E+07
P6 - Safety Factor Minimum	★★★ 4.1126	★★ 4.0732	★★ 3.8964

Fig. 10: Optimized parameters Candidate

Finally, the chosen design point (DP) replaced the current parameters, and then the structure is updated with the new values. A tabular comparison of the result is given in table 4.

Table 4: Structural optimization analysis comparison

Total deformation [mm]		Maximum Equivalent stress [Mpa]		Elastic strain [mm/mm]		Safety factor	
Before	After	Before	After	Before	After	Before	After
0.00024557	0.00017333	7.29E+07	6.15E+07	0.00036567	0.00031017	3.431	4.0646

4. CONCLUSION

Finite element method is as an effective method in the modelling as well as analysis of structures. In this paper, weight and structural optimization of a connecting rod together through Finite Element Method using ANSYS is carried out and presented. The processes were performed under a loading of 100N and 500N static force respectively 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.

From the analysis result, 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.

Acknowledgements

The authors would like to thank Universiti Malaysia Pahang (UMP) for providing the research grant (RDU1803138) and the facilities.

References

[1] J. Heywood, Internal Combustion Engine Fundamentals, 1 edition, McGraw-Hill Education, 1998.
[2] P. Shenoy, "Dynamic load analysis and optimization of connecting rod," (M.S. thesis). The University of Toledo, US, 2004.
[3] M. Ilman, R. Barizy, "Failure analysis and fatigue performance evaluation of a failed connecting rod of reciprocating air compressor," Engineering failure Analysis, 56(2015), 142-149.
[4] M. H. H M Ali, M. Haneef, Analysis of Fatigue Stresses on Connecting Rod Subjected to Concentrated Loads at The Big End," Materials Today: Proceedings, 2 (2015), 2094 – 2103.
[5] C. D. Gopinath, "Design and Optimization of Four Wheeler Connecting Rod Using Finite Element Analysis," Materials Today: Proceedings, 2, (2015), 2291 – 2299.
[6] B. P. Vivek. C. Pathade, "stress Analysis of I.C. Engine Connecting Rod by FEM," International Journal of Engineering and Innovative Technology (IJEIT), 1(3) (2012) 12-15.
[7] B. R. R. Pushpendra, K. Sharma, "fatigue analysis and optimization of connecting rod using finite element analysis," International Journal of advance research in Science and Engineering (IJARSE), 1(1) (2012), 3367-3371.
[8] R. Bansal, "Dynamic simulation of connecting rod made of aluminum alloy using finite element analysis approach," IOSR Journal of Mechanical and Civil Engineering (IOSR), 5(2) (2013), 1-5.
[9] Y. Liu. X. Chen, Finite Element Modeling and Simulation with ANSYS Workbench, CRC Press, London New York, 2014.
[10] S.S. Gaikwad, E.N. Aitavade, "Static analysis of roller of gravity roller conveyor for structure strength & weight optimization," IJAET, (4), 2013.
[11] G. Gopal, L S. Kumar, K V. B. Reddy, M U. M. Rao, G. Srinivasulue "Analysis of Piston, Connecting rod and Crank shaft assembly," Materials Today: Proceedings, 4(8) (2017) 7810–7819.
[12] W. Guanzhu, Z. Guoqing, Y. Jiancheng, G. Youping, G. Wei, Z. Hengcai, "Modal analysis of high-speed spindle based on ANSYS," in 2012 7th International Conference on Computer Science & Education (ICCSE), Melbourne, VIC, Australia, 2012.
[13] B.S. Kim, S.H. Lee, M.G. Lee, J. Ni, J.Y. Song, C.W. Lee, "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 (2017), 30-36.
[14] G.H. Janq, S.H. Lee, M.S. Jung, "Free vibration analysis of a spinning flexible disk- spindle system sup- ported by ball bearing and flexible shaft using the finite element method and substructure synthesis," Journal of Sound and Vibration, 251(1) (2002), 59-78.
[15] E. J. Barbero, Finite Element Analysis of Composite Materials Using ANSYS, US: CRC press, Tylor and Francis group, 2014.
[16] Nipun, "Difference Between Yield Strength and Tensile Strength," 14 10 2015. [Online]. Available: http://pediaa.com/difference-between-yield-strength-and-tensile-strength/.
[17] B. Talikoti, S. N. Kurbet, V. V. Kuppast, M. Arvind, "Harmonic analysis of a two-cylinder crankshaft using ANSYS," in 2016 International Conference on Inventive Computation Technologies (ICICT), Coimbatore, India, 2016.
[18] Q. Lv, Y. Mei, "Modal analysis of a magnetic climbing wall car frame based on the ANSYS," in 2014 IEEE Workshop on Electronics, Computer and Applications (IWECA), Ottawa, ON, Canada, 2014.
[19] A. Muhammad, I. H. Shanono. "Structural Analysis of a Knuckle Joint using different materials," in 1st International Civil Engineering Conference (ICEC 2018), 100-106. Nigeria, 2018.
[20] A. Muhammad, I. H. Shanono, "Finite Element Analysis of a base stand using different materials," Science Technology and Arts Research Journal, 2018.
[21] C. Smith, Automated Design, 2011 ANSYS, Inc., 2012.