IOP Conference Series: Materials Science and Engineering

Finite Element Analysis of a connecting rod in ANSYS: An overview

To cite this article: A Muhammad *et al* 2020 *IOP Conf. Ser.: Mater. Sci. Eng.* **736** 022119

IOP Conf. Series: Materials Science and Engineering **736** (2020) 022119 doi:10.1088/1757-899X/736/2/022119

Finite Element Analysis of a connecting rod in ANSYS: An overview

A Muhammad[1,3]**, M A H Ali**[1] **and I H Shanono**[2,4]

[1]Faculty of Mechanical and Manufacturing Engineering, Universiti Malaysia Pahang,
[2]Faculty of Electrical Engineering, Universiti Malaysia Pahang, Malaysia
[3]Department of Mechatronics Engineering, Bayero University Kano
[4]Department of Electrical Engineering, Bayero University Kano

E-mail : ayshermuhd@gmail.com

Abstract. Automotive engineering design has recently focused majorly on the stress analysis and optimization of a connecting rod with keen emphases to the vital parameters such as deformation, stress, fatigue and strain, factor safety, life values among others. The performance of a connecting rod in an automobile engine is influenced by its design and weight. Hence, for the production of a durable, cheaper and lightweight connecting rod, analysis and optimization become necessary. This article provides a review of some vital work done by various researchers in designing, analyzing and optimization of connecting rod of an engine using Finite element analysis in ANSYS workbench A comprehensive comparison table and graphs for the reviewed research articles have been provided. The article will serve as a stepping stone for both old and new researchers in the field of automotive design.

1. Introduction

One resource of energy in automobile sector is internal combustion engine. Internal combustion engine transforms chemical energy in the form of reciprocating movement of the piston right into mechanical energy. Internal Combustion engine has numerous components like cylinder, piston, connecting rod, crank and also crank shaft [1], [2]. Connecting rod is just one of the vital driving components of light engine, it creates an easy device that transforms direct movement right into rotating movement. Thus, a connecting rod is utilized to transform reciprocating movement of the piston right into rotating activity of the crankshaft [3]. Hence, the connecting rod has to be sufficiently strong to withstand forces without damages due to the pressure created from the burning [4]. A connecting rod is a framework that sustains the axial compressive tons that has a tendency to stop working due to inelasticity [5]. The connecting rod directly sees the pressure put on the drive as well as crank train as it equates the direct activity of the stroke of the piston to the rotational activity of the crank. Devastating damages would certainly take place to the engine needing costly repair services if the connecting rod was to stop working.

As the latest created internal combustion engine began to be standardized, the significance of the connecting rod along with the crank shaft was necessary to the procedure of a dependable engine. Throughout the manufacturing of these internal combustion engines, the power generated has actually progressively been increasing, causing enhanced pressures being established in modern engines.

Therefore, effort and time are essential to produce the very best connecting rod to permit it to take care of high stress with lesser weight [2]. There are two types of connecting rod: H-beam and I-beam or even a combination of the two. These connecting rods are utilized based on the area of application and usage. An I-beam is strong and light, however the kind of product utilized, limits its capability to manage high pressure. Whereas an H-beam are used in high power engines because they can take care of far more stress and anxiety without flexing [6]. The connecting rod ought to be such that it can maintain the optimum load with no failing throughout the cycle [1]. Figure 1 shows the parts of a connecting rod [2]: the crank end is connected to the crank pin by a shaft. The pin and crank-end pinholes located at the top and bottom ends are machined to allow precise installation of bearings. These openings need to be identical and parallel. The top end of the connecting rod is attached to the piston by the piston pin. As the bottom end of the connecting rod rotates with the crankshaft, the top end is compelled to reverse on and forth on the piston pin. The bushing is essential due to the fact that of the high stress and temperature levels. The bottom hole in the connecting rod is divided to allow it to be secured around the crankshaft. The material is used for the rod is used for the cap which is then screwed by two screws [7].

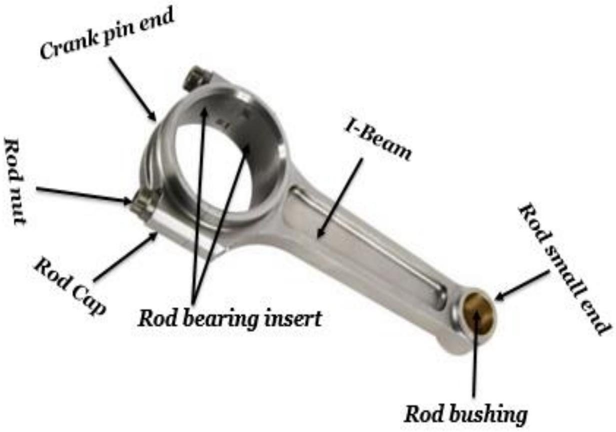


Figure 1. Connecting rod parts.

In modern engines, the connecting rods are most typically made from steel, however can be constructed from lightweight aluminium (for agility as well as the capability to soak up high influence at the expenditure of toughness) or titanium (for a mix of stamina as well as agility at the cost of price) for high efficiency engines, or of cast iron for applications such as electric motor mobility scooters. They are not fixed at both end, to ensure that the angle in between the piston and connecting rod alters as the rod goes up and also down as well as turns around the crankshaft. In competing engines, they might be called "billet" rods, if they are machined out of a strong billet of steel, instead of being cast [8]. It goes through complex loading of high cyclic loads of the order of 108 to 109 cycles, which vary from high compressive to high tensile loads due to inertia. Sturdiness of this element is of important value [9], [7]. As a result of these aspects, the connecting rod has actually been the subject of study for various facets such as manufacturing innovation, materials, efficiency simulation, stress, and so on [6]. It is essential for designers to appropriately figure out the stress existing within the connecting rod to make certain it will carry out even in worst situation without falling completely. By having the ability to design the connecting rod and also establishing the pressures existing on it under all problems, it is feasible to make use of programs such as ANSYS to establish the stresses created within it. The decrease of weight is very important to make certain that the engine can run securely in greater RPM's because of the lowered inertia held within the lighter weight connecting rods. The objective is to eliminate as much product while still keeping their toughness as well as stability [2].

1.1. Finite Element Analysis

The finite element analysis can be mapped back to the job by a Russian-Canadian Alexander Pavlovich Hrennikoff (1941) and a German American mathematician Richard Courant (1942). Hrennikoff presented the structure approach, in which an aircraft flexible medium was stood for as collections of light beams and bars. These leaders share one important feature: mesh discretization of a constant domain name right into a collection of distinct sub-domains, normally called Elements [10]. Finite element analysis (FEA) has actually ended up being a commonplace over the last few years, and also is currently the basis of a multibillion buck market annually. Mathematical solutions to extremely difficult stress problems are currently being solved consistently with FEA [11]. The Finite Element Method (FEM) is a mathematical strategy to discover approximate options of partial differential formulas given that patterns of mostly all physical system can be stood for by these formulas [12], [13], [10], [14],. It is a mathematical technique for fixing issues of design as well as mathematical physics. It was stemmed from the requirement of fixing complicated flexible as well as architectural analysis problems in Civil, Mechanical and also Aerospace design [10], [14]. If the precision requirements are not satisfied, the mathematical solution has actually to be duplicated with finer solution (meshes) till enough precision is gotten to [15]. Common FEA Applications includes: Mechanical/Aerospace/Civil/Automotive Engineering, Structural/Stress Analysis (Static/Dynamic, Linear/Nonlinear), Fluid Flow, Heat Transfer, Electromagnetic Fields, Soil Mechanics, Acoustics, Biomechanics etc. The growth of FEA for functional design issues started with the introduction of the electronic computer system. That is, the significance of FEA for design solutions is a collection of algebraic formulas developed to addressed these problem efficiently, by making use of the electronic computer system [15], [10].

FEA are commonly utilized in design evaluation, as such we can anticipate this usage to boost substantially in the coming years. The steps are used in evaluation of structures as well as solids liquids, heat transfers as well as every area of engineering evaluation [15]. Some of the advantages of FEA include: Irregular Boundaries, loading, materials types, constraints, element size, modification, etc. FEM permits whole layouts to be built, fine-tuned and also optimized prior to design. This effective tool has actually substantially enhanced both the requirement of design styles as well as the method of the style procedure in numerous commercial applications. Using FEM has actually considerably reduced the time needed from idea to the assembly line. One has to take the benefit of the development of faster generation computers for the evaluation and design with accurate degree of precision [10]. A variety of prominent finite element analysis brand are currently readily available. A few of the preferred bundles are STAAD-PRO, GT-STRUDEL, ABAQUS, NASTRAN, NISA as well as ANSYS [12] [4] Several of these software program are effective yet intricate indicated for expert designers with the training as well as education and learning essential to effectively translate the outcomes [10]. With these packages, one can examine a number of complicated structures.

The finite element analysis typically includes 3 basic primary steps: pre-processing, solution and post processing [11], [12]. In the pre-processing stage, the designer constructs a design of the component to be assessed in which the geometry is split right into a variety of distinct sub-regions, or "element", linked at distinct points called "nodes". The designer specifies the geometric domain name, the element type to be utilized, the material property of the components (product is specified by its material constants, every aspect needs to be appointed a specific material), the geometric of the components, the element connection (You can define a specific number of components in a details location, compel the mesh generator to preserve a certain component dimension), boundary conditions and loading. The dataset prepared by the pre-processor is utilized as input in the solution stage which constructs and addresses a system of nonlinear or direct algebraic equations. The solution is derived by collecting all defined information such as load application (that include pressure, moment, force, rotations) after specifying the entire problem. With the postprocessor software you can visualize the outcomes, for instance, the plot form of the geometry deformation or stress values or anxieties using advanced regimens made use of for arranging, printing, and also outlining results from a finite element solution. Also, if you favour tabular listings or data hard copies, listing the result is feasible. Figure 2 summarizes the process of finite element analysis [11], [12].

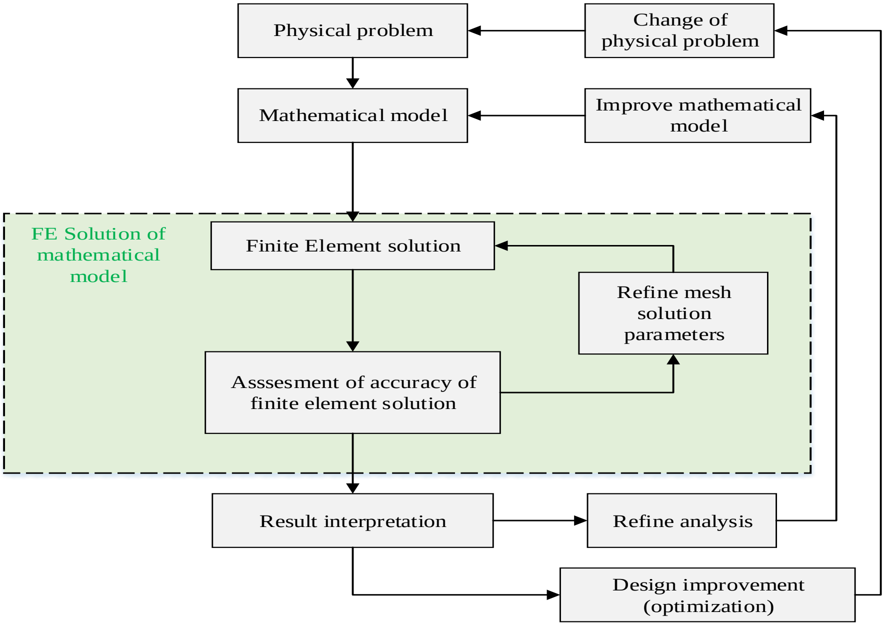


Figure 2: Process of Finite Element Analysis.

1.2. ANSYS

ANSYS is the initial (and also generally utilized) name for ANSYS Mechanical or ANSYS Multi-physics, general-purpose finite element analysis software program. The scholastic variations of these business products are referred to as ANSYS Academic Research, ANSYS Academic Teaching Advanced, Introductory etc. [16], [17]. ANSYS is a general-purpose finite-element modelling plan for numerically addressing a wide range of mechanical problems. [18]. The ANSYS program is an effective, multi-purpose evaluation device that can be made use of in a wide array of design techniques. Prior to making use of ANSYS to create an FEA design of a physical system, the following concerns need to be responded to base upon design judgment and monitoring [17].

- What are the goals of this evaluation?
- Should the whole physical system be designed, or simply a part?
- How much is information should be consisted of in the design?
- How fine-tuned should the mesh be?

In addressing such concerns, the computational expenditure ought to be stabilized versus the precision of the outcomes. The ANSYS program can be utilized in a reliable and also appropriate means after thinking about the following: type of problem, time reliance, nonlinearity and modelling idealizations. This type of analysis addresses numerous architectural problems such as: Static, modal, fatigue transient dynamic analysis, shape optimization, harmonic analysis, eigenvalue buckling, and heat transfer analysis [19], [17].

2. Materials and methodology

2.1. Modeling of the structure

The latest advanced computer –aided engineering (CAE) softwares are used in the analysis of repetitive computer aided designs. For an in-depth 3D design of a physical structure, the Computer-aided design CAD is primarily used, which can be use from design process through dynamic and strength analysis of structures. With these CAD software, using an integrated graphical user interface, engineering projects are designed modified, optimized, analyzed, and managed. The output from a CAD is normally in the form of an electronic file.

Some of the major CAD software include: AutoCAD, CATIA, SolidWorks, Pro Engineer (PTC), Unigraphics (UGS), I-DEAS (SDRC), BricsCAD and so on. ANSYS provides a two-way connecting to most of the CAD with the aim of boosting its efficiency. Before analysis is carried out, the connecting rod is either created in a CAD software, saved in an Initial Graphics Exchange Specification (IGES) format and then imported into the ANSYS workbench or by designing the complete structure in the ANSYS workbench [20], [21], [22]. Most of the revised articles in this paper performed analysis on a connecting rod by importing the geometry in an IGES format into the ANSYS workbench. In (Shrivastava, 2017), a Hero splendor connecting rod was analysed under static loading condition for stress, strain and deformation values. Before the analysis in ANSYS, design of the model was done using CRE-O software. The result obtained was compared with that using a different material to determine the best material that is needed in the design of a connecting rod.

In a paper by Saeed et al [23], using a connecting rod designed in Pro/E software, analysis with three different materials, starting from forge steel, followed by Aluminium and then finally replaced with carbon fibre was done in order to reduce the total weight of the connecting rod. A bike was used to test in an ideal condition the maximum capacity of the connecting rods with the gears varied at regular interval of speed and time. Also, a piston, crankshaft and connecting rod was further assembled and analysed. Figure 3 gives a pictorial representation of the CAD software used in the 3D design of the connecting rod in the revised literatures.

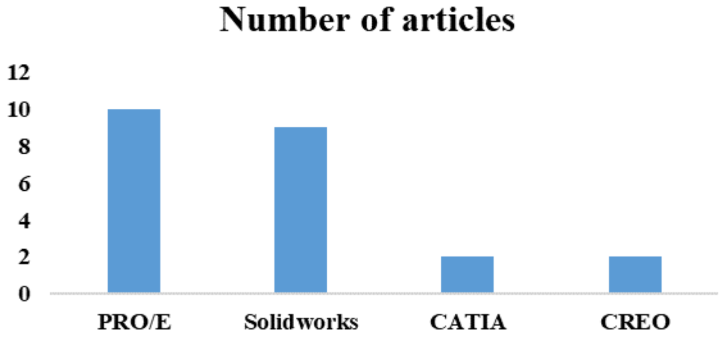


Figure 3. Connecting rod modelling software.

2.2. Material property definition

Material properties are defined based on the type of simulation analysis to be performed. For an efficient and qualitative analysis of material, the material properties need to be correctly and carefully entered. These materials which can be either linear or nonlinear, isotropic or orthotropic, constant or temperature dependent needs to be defined correctly. Depending on the aim of the analysis, some mechanical properties such as density, strength and coefficient of thermal expansion definition is optional [24]. Knowing and declaring the correct value of the material property is very useful for design analysis purpose. The different types of material indicated by different density will show different analysis result outcome as well.

The Young's modulus of a material alternatively called modulus of elasticity is a numerical constant that describes the elasticity and measures the capability of a solid to withstand changes when subjected to tension or compression in a certain direction. The higher Young's modulus, the stiffer (i.e. how it deflects under load) is the material which will require a much higher amount of load to deform. Poisson’s ration which is the ratio of compression to the expansion of material together with Young's modulus (ratio of stress to strain) defines the strength and nature of how a material structure deforms based on a particular constraint. 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 [25]

Analysis of a connecting rod in ANSYS 15.0 software was performed and presented in [26] to analyse its stress, strain and deformation using aluminium alloys 7068 T6, T6511and steel material. The obtained analysis result shows that aluminium alloys is stiffer with less stress value, weight and higher factor of safety when compared to the forged steel material connecting rod. The structural design of the connecting rod is performed in SolidWorks software based on the theoretical calculations done. Kuldeep B. et al [27] also performs similar analysis on a connecting rod using the same two materials (AL360 and ALFASIC composite). The displacement, stress and strain values using each of the materials were obtained and compared. Using the later composite, a lighter and stiffer optimized connecting rod was found with a reduction in weight and displacement of 43.48 and 75% respectively. Analysis in ANSYS software on five different materials (aluminium 360, forged steel, titanium alloy, ti-13v-11cr-3al, magnesium alloy, beryllium (alloy 25) was studied and presented in [1]. The deformation, stress, and strain values were determined after the replacement of the structure material with each of the material type. The minimum stress was found to occur at the piston and crank end respectively. Comparison of the different materials results reduces the inertia force as the weight is reduced. In [6], using four different materials (Aluminium 2024 T6, aluminium 7075 T6, Carbon steel CrMo4, and TI-6AI-7V) in ANSYS 15 software, Structural and fatigue analysis was done to estimate the deformation and strength of a connecting rod. The connecting rod design and modelling was done in SolidWorks 15 software. Analysis was carried out. The analysis result obtained shows titanium alloy with the highest factor of safety and minimum deformation value, making it the best material for the production of connecting rod.

In [28], strength analysis on a connecting rod using different materials (structural steel, aluminium alloy, titanium, and Magnesium alloy) was performed and presented. The deformation, stress, strain and factor of safety where analysed and compared. Structural steel material proved to be more durable when compared to the other materials. Savanoor et al [29] present an analysis on an IC connecting rod by comparing the result obtained using two different materials (aluminium alloy and forged steel). Aluminium alloy was found to be the best material having a lesser weight with a reduction of about 63.18% in weight. After changing an 180cc engine connecting rod of material Al360 to a reinforced composite of silicon carbide and fly ash (Alfasic composite) material, analysis using ANSYS workbench was performed by Nikhil et al [30] The simulation result shows weight reduction from the changing of the material of about 39.48% with a stiffer structure.

Analysis of a connecting rod using different materials have been carried out and presented by these researchers. From the analysis result presented in each case when compared with the different materials, a coherent argument on the loading and type of materials used have been made. We analyse a connecting rod for its strength and deformation in ANSYS software under the constraints and loading with different materials. Figure 4 shows a graphical comparison of the Von-Mises stress and deformation result of the various materials obtained. The best material choice for the connecting rod as stated in proves to be structural steel having a lower deformation value followed by titanium, aluminium and then magnesium alloy. A graphical chart of the material type used in the revised literatures is shown in figure 5.

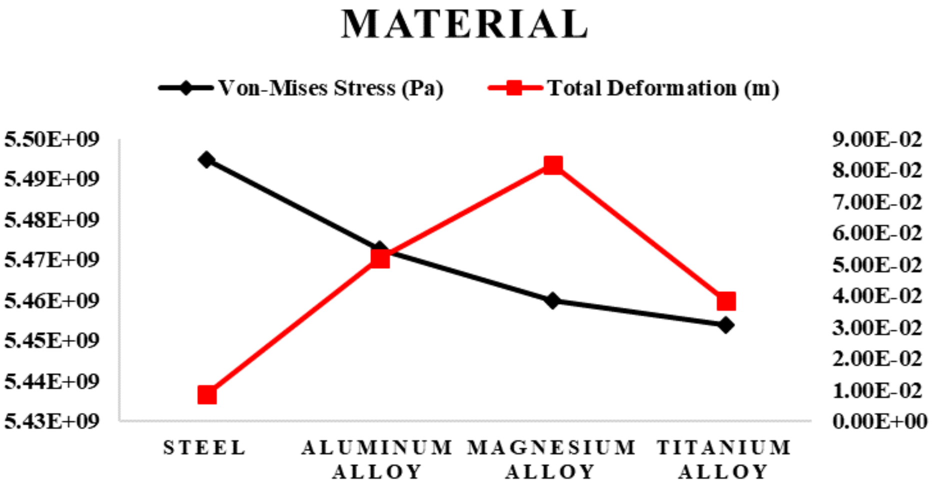


Figure 4. Materials stress-deformation comparison.

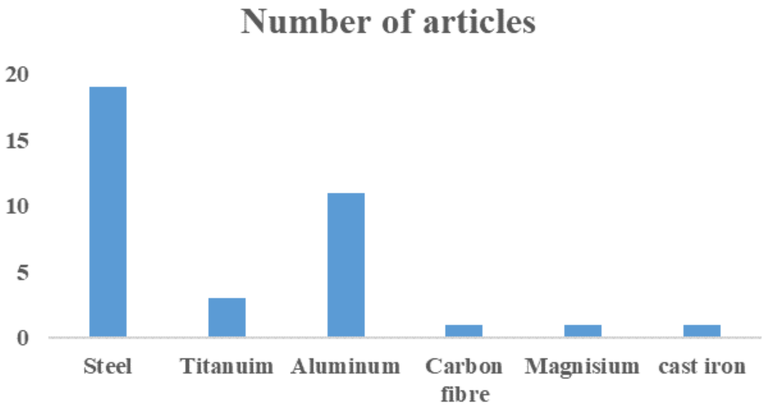
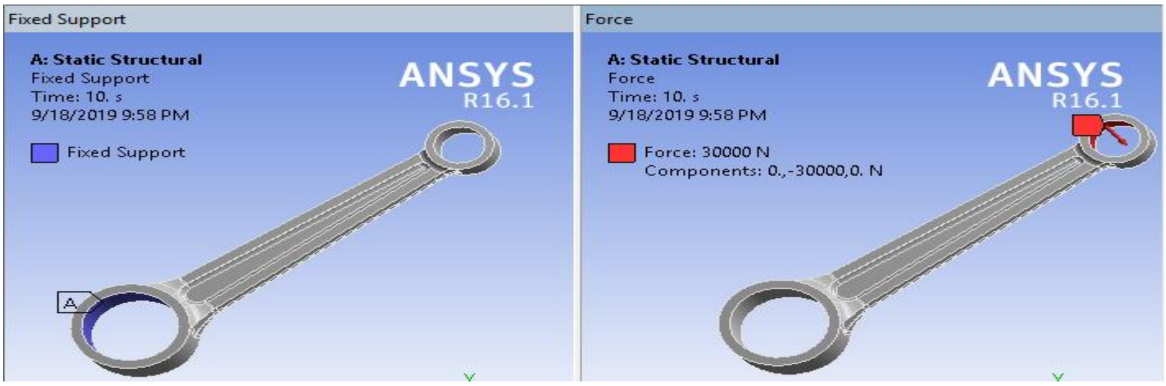


Figure 5. Type of materials.

2.3. Load application

Establishing boundary condition is an important factor to consider in FEA. The first key action required prior to the start of an analysis is the setting of boundary conditions [31]. Constraints that include fixed support, forces, and pressure applied to the model after meshing process are embedded in a way that adapt to the practical real life situation, which is an important and main action needed in analysis [20]. The deformation, fatigue, stress and strain values of the connecting rod are based on an acceptable design constraint using either linear or non-linear static analysis.

The static and dynamic analysis using ANSYS software of connecting rod using different materials (aluminium and titanium alloys) was done and presented in [32]. The deformation, stress, and strain values under a loading condition of 16Mpa was determined for the static structural analysis which shows Titanium allow with the less deformation value. Dynamic analysis for each material with different modal frequencies was also done and tabulated. From the revised literatures, the boundary conditions on the connecting rod include supports and loads (external force and pressures) as shown in figure 6, the connecting rod small end is subjected to the loading while the big end is fixed.



Fixed Support
A: Static Structural
Fixed Support
Time: 10. s
9/18/2019 9:58 PM
Fixed Support

Force
A: Static Structural
Force
Time: 10. s
9/18/2019 9:58 PM
Force: 30000 N
Components: 0.,-30000,0. N

Figure 6. Boundary condition.

3. Strength analysis

As a result of the unforeseen failing of design structures, analysis comes to be really considerable to identify the safety of these structures. Such failures over a details period of time leads to the unexpected collapse of the structure. A designer makes uses of Von-misses stress evaluation to establish the failure pattern of the structure. When the maximum stress value is more than the material strength, then failure will occur [25]. Von Mises-Hencky distortion energy theory is one of the criteria used by designers to predict the failure of structures, Von-Mises stress is ideally used for ductile materials [33].

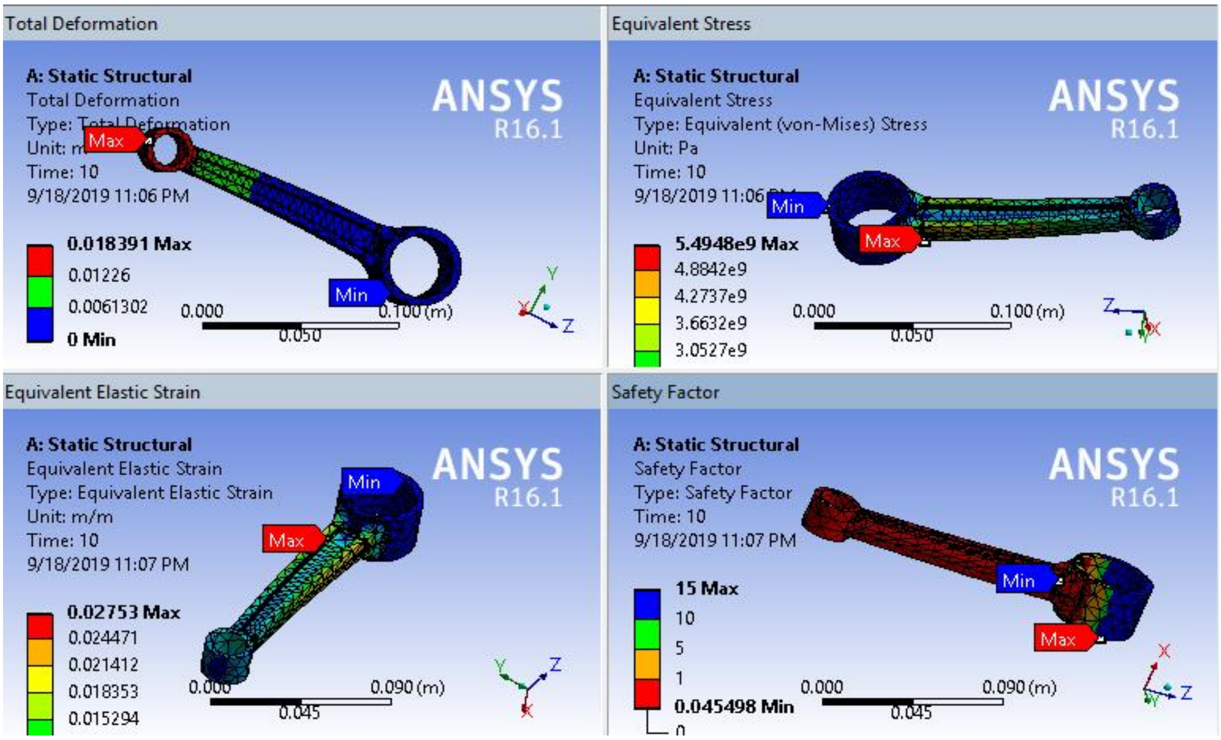
Failure of connecting rod is attributed to the in availability of much strength needed to hold the load. This can be overcome with the life cycle extended by increasing the strength. Static analysis of the stress that emerged in a connecting rod was carried out and presented in [2]. The CAD design of the connecting rod was done in a Pro-E 2.0 software. The analysis under static loading of 4319N (a function of crank angle) at the piston end with the crank end restrained was performed using ANSYS software. Using a tetrahedral meshing with 18921 and 31237 umber of element and nodes, simulation was performed. Initially, the maximum loading force was assumed to be at the power stroke top dead center. The maximum and minimum values of the Elastic strain, Von-misses and shear stress was also noted. In [34], Finite element analysis simulation was conducted to determine the cause of failure of a motor cycle connecting rod. the simulation in ANSYS software was used to validate the inspected surface failure and strength. Results obtained shows that the fatigue loading cracks was as a result of the existence of scale build-up inclusions Further design improvement and future failure avoidance were also predicted from the simulation results.

Omid et al [35] present a continuum mechanic method for fatigue analysis of a connecting rod of a universal tractor (U650). Using a connecting rod of steel alloy material subjected to a fully reversed maximum loading of 9500N with a 10-node tetrahedral mesh size was analysed for stress value. Result obtained shows the shank end and piston pin area as the critical stressed points. The authors further conclude that reducing the stress coefficient concentration, the fatigue life of the connecting rod can be further improved. In another research study by Puran et al [36], the calculation of the toughness as well as distortion attributes of a connecting rod was attend to. Furthermore, the finite element analysis was done using ANSYS to calculate the fatigue and deformation arising in the connecting rod under the loading condition. Architectural systems of connecting rod can be evaluated utilizing Finite Element Methods. In a paper by Ranjbarkohan et al [37], FEM method was also used in ANSYS workbench to perform analysis of a samand engine connecting rod for fatigue failure. The connecting rod was subjected to a maximum tensile and compressive load condition to evaluate the critical failing points.in order to improve the strength and life of the connecting rod, further optimization was also performed. Experimental data form Iram Khodo's lab was used with MSc. ADAMS software on a

C70S6 steel connecting rod material was used under four different speed (1000, 3000, 4500, and 6000PRM). Analysis result obtained shows a maximum stress (297.361Mpa) occurring at the pin end.

The design and analysis of an internal combustion engine connecting rod was done and presented in [8]. the analysis was perform using FEM to determine the deformation, strain and stresses that result in the connecting rod when subjected to the loading condition (pressure during combustion). The connecting rod is designed and modelled using SolidWorks software and analysed using ANSYS workbench with AISI 4340 steel material under an axial force of 32517N. In [38], FEM was used to analyse a connecting rod for high Cycle Fatigue Strength using C70S6 material. Six different loading cases were considered and then considered one more combination of two loading cases. The stress values and factor of safety are the factors considered during the analysis which gives a different pattern as the different load case were used.

Anusha et al [39] and [40] also presents an analysis on a single cylinder four stroke engine in ANYS for the stress, shear and strain values under certain loading conditions. In [41], the design examination of a Hero Honda Motor Cycle connecting rod using Finite Element Analysis for fatigue life was carried out and presented. To start with, a correct Finite Element Model is established making use of Cad software program CATIA and then analysis is done to identify the stress and deformation of the connecting rod under the given loading conditions. The structural strength of the connecting rod was further confirmed with a Universal Testing Machine (UTM) for tensile loading. Dilip in his paper [42] provides the solution of real-time problem in the engine of DI tractor of Mahindra & Mahindra by utilizing the CAD and analysis methods. The design of the elements of engine is done with measurements set up those parts and afterwards simulate the entire system up for every turning of the crank. This will certainly provide the specific area and cause of fouling. The design modelling was done using PRO-E wildfire 4.0 for modelling the engine elements while analysis software ANSYS is utilized to assess as well as imitate the entire setting up. Results obtained by the presented study was further analysed. Under the same loading and design constraint, analysis for stress, deformation, strain and safety factor was done and the result shown in figure 7.


Total Deformation
A: Static Structural
Total Deformation
Type: Total Deformation
Unit: m
Time: 10
9/18/2019 11:06 PM

Equivalent Stress
A: Static Structural
Equivalent Stress
Type: Equivalent (von-Mises) Stress
Unit: Pa
Time: 10
9/18/2019 11:06 PM

Equivalent Elastic Strain
A: Static Structural
Equivalent Elastic Strain
Type: Equivalent Elastic Strain
Unit: m/m
Time: 10
9/18/2019 11:07 PM

Safety Factor
A: Static Structural
Safety Factor
Type: Safety Factor
Time: 10
9/18/2019 11:07 PM

Figure 7. Strength analysis result.

4. Design optimization

Traditionally, when failure occurs in an engineering structure, to redesign the structure, a trial and error are done until a benchmark is reached. This trial and error approach is time-consuming and inefficient. Recently, numerical optimisation techniques are used to balance such trade-offs. From the reviewed literatures, structural optimisation via simulation in ANSYS workbench is used. To achieve an excellent material distribution with effective design, the parts that add least or nothing to the load bearing are identified and removed for weight reduction [25].

Yuanchun [43] performs dynamic analysis and optimization of an engine connecting rod to determine the stress distribution. Transient analysis was used to find the maximum stress value which occurs during the working stage of the engine. A static load generated from dynamic loading using the method of equivalent static load to optimized the analysis. Figure 8 shows the equivalent static load method used for the optimization process. Pro/MECHANICA coupled with ANSYS helps not only just only to gets over the negative aspect that it is tough to construct complex design in ANSYS but also takes complete benefit of excellent graphics drive innovation in Pro/E which sustains the manageability as well as the operability of user interface. Xianjun et al [44] present a local and global sensitivity analysis of a LJ276M connecting rod with the application of Pro/Mechanical software. Optimization in ANSYS of the structures design parameters (D, L, h) was also done and comparison in terms of the deformation, safety coefficient, mass and stress values before and after the optimization process was carried out. The optimization results show a reduction of 4.9% in the stress value with an increase of 1.7 and 5.4% in deformation and safety value respectively. Su et al [45] present another analysis on a connecting rod to ascertain its failure at the big end by taking the big end radius as the design variable. The stress value being higher at the big end than the small end under the compression load condition Further optimization of the corner radius was performed after the evaluation of the stress distribution at the area.

The Optimization accomplished in evaluation offers deep understanding by thinking about optimal criterion for recommendation of alteration in the existing connecting rod. Pranav et al [46] performs stress analysis on a connecting rod and further optimize the structure taking the fatigue stress as the design factor using ANSYS in order to reduce the weight. Result achieve shows about 20% weight reduction under static loading. Prakash et al [47] performs buckling analysis on a connecting rod by changing its cross sectional area under different loading condition. Using this approach, the authors were able to move the stress region to the middle from the smaller area. Hence, the connecting rod have been optimized by improving its shape.

Ramani et al [7] analysed the stress arising during the compression of a connecting rod, he presents an optimized connecting rod by performing the weight reduction analysis using ANSYS workbench on a steel material connecting rod. Simulation result of the stress analysis at some certain points was shown and the structure was further optimized to a lighter weight (approx. 15%) to minimizes space and material. FEA was also used to carry out fatigue analysis and optimization of a connecting rod using a forged steel material by Pushpendra et al [48]. Under static tensile loading, the life, deformation, Factor of safety, Stress, and Fatigue sensitivity were observed from the analysis result obtained using ANSYS workbench. furthermore, based on these factors, optimization was done. A final conclusion was drawn by the authors that using a C-70 material in place of the forged steel could further reduced the weight of the connecting rod by 20%.

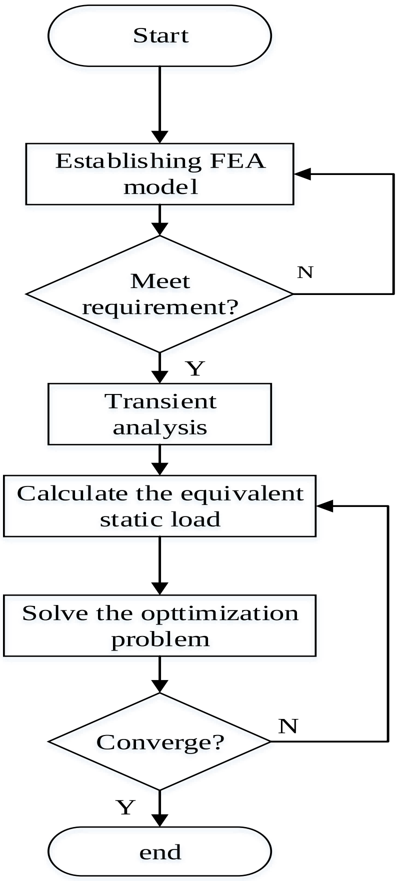
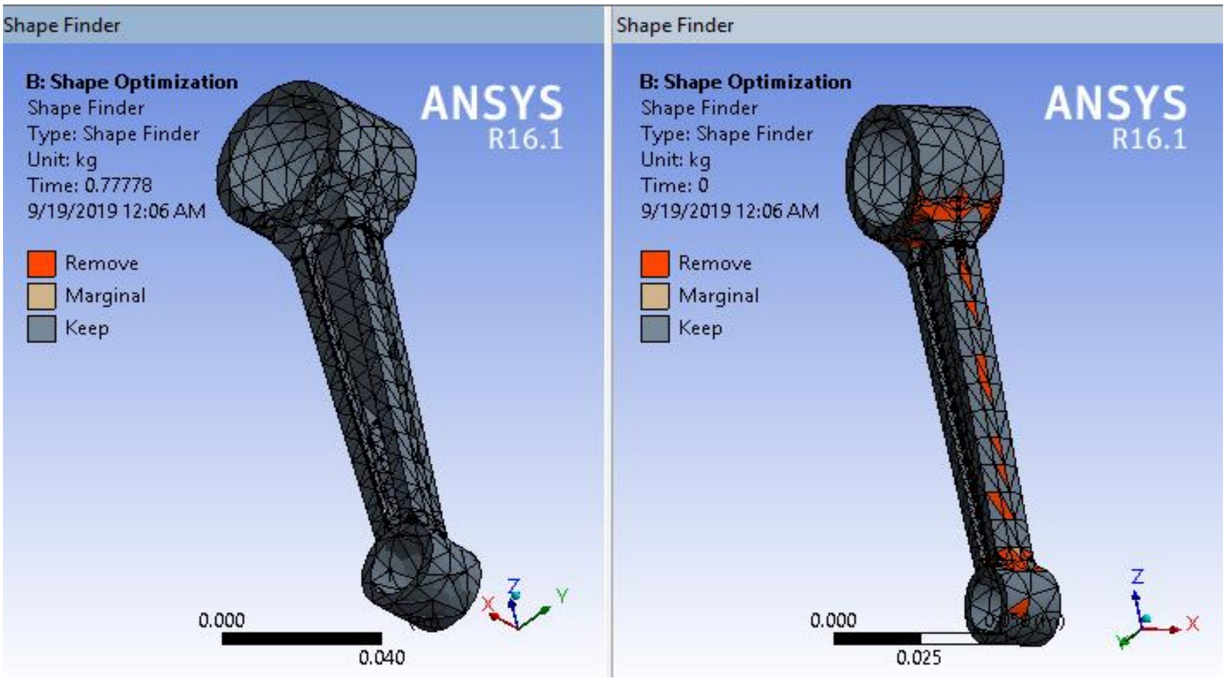


Figure 8. Optimization process [43].

Muhammad et al [49], [50] present a topology and structural optimization of a diesel engine connection rod. In order to determine the mass needed to be eliminated, Optimization is carried out with target weight reduction of 20%, 30%, 40%, 50%, and 60% under a static loading of 100N. Furthermore, comparison in the deformation, stress, strain and factor of safety under the same loading condition was compared before and after a 60% target weight reduction and structural optimization. The authors finally conclude 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. Santhosh et al [51] uses a Bajaj Pulsar 150c.c engine to perform weight reduction analysis on a connecting rod. The analysis is conducted in two different situations. First, multi body dynamic analysis and fatigue failure and stress were determined under static loading condition. Then, optimization was done to determine the weight reduction using various load by elliptical cut out I-section in the connecting rod. Yogesh et al [52] presents a fatigue analysis of a connecting rod using ANSYS. Stresses at various point on the connecting rod was determined to ascertain which part is more susceptible to failure that needs extra hardening treatments. Furthermore, optimization was conducted for a weight reduction of 20%.

Design and analysis of a connecting rod in CREO 3.0 and ANSYS software was presented in [53]. The deformation, fatigue life, factor o safety, stress and stress values were evaluated to the determine the weak areas in the structure while considering the variations in design and weight. Shaik et al [54] performs analysis on a connecting rod of Ashok Leyland Bharat Stage –II engine was optimized for weight and cost reduction under both static and cyclic dynamic compressive tensile load subjected at its two extreme ends. Mathematical modelling calculations was also provide before performing the analysis. Ran et al [55] also present static analysis and weight optimization (13%) of a connecting rod considering alloy steel material.

Optimization of connecting rod have been done using various weight target. From the revised articles, using ANSYS under similar loading condition, optimization for a target reduction in weight of 15, 20, 30, 40 50 And 60% was done to ascertain the available optimal design as shown in figure 9 and 10, respectively. Summary of finding reported by literature is shown in table 1.


Shape Finder

B: Shape Optimization
Shape Finder
Type: Shape Finder
Unit: kg
Time: 0.77778
9/19/2019 12:06 AM

Shape Finder

B: Shape Optimization
Shape Finder
Type: Shape Finder
Unit: kg
Time: 0
9/19/2019 12:06 AM

Figure 9. Shape optimization.

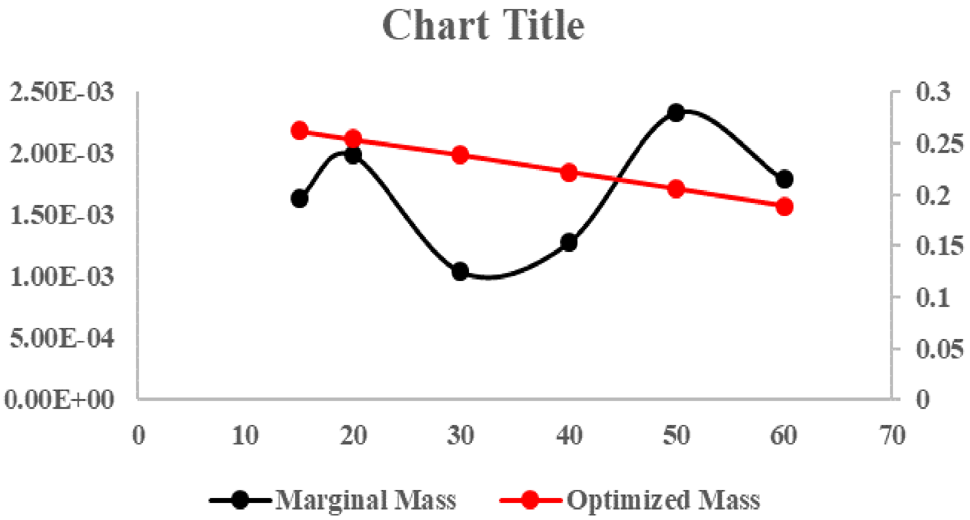


Figure 10. Mass comparison.

Table 1. Comparison of the reviewed articles.

Author	Modelling software	Material	Loading point	No.of Element/nodes	Result
[30]	Pro/E	Carbon steel	Piston end	18921/31237	Shear Stress, Von Mises Stress, Elastic Strain, Elastic Deformation
[6]	Solidworks	2024-T6, 7075-T6, 43CrMo4, Ti-6Al-7v	Piston end	11628/20929	Deformation, stress, mass, safety factor
[34]	3D scan laser camera	NA	Small end	NA	Von-mises stress distribution
[1]	SolidWorks	Aluminum, carbon fiber, Magnesium, Titanium and Beryllium alloy	Small end	NA	Strain, stress, total deformation and factor of safety
[8]	SolidWorks	AISI 3430 steel	Small end	214835/343779	Von Mises stress, elastic strain, shear stress, Total deformation
[43]	NA	Steel	Small end	NA	Maximum stress
[7]	Pro-E	Steel	Big end	179355/304977	Stress and strain
[30]	NA	Al360, ALFASiC	Small end	NA	von misses stress and displacement
[27]	NA	Al360, ALFASiC	Small end	187/117439	von misses stress and displacement
[52]	Pro/E	NA	Small end	NA	Fatigue life, damage, factor of safety, stress biaxiality, fatigue sensitivity
[44]	Pro/MECHANICA	40Cr	Small end	NA/22536	Displacement and stress
[26]	SolidWorks	Aluminum Alloy and Forged Steel	NA	NA	Von misses Stress and strain, Deformation, Factor of safety and weight reduction
[45]	CATIA	AISI 4140 Steel	NA	NA	Stress and strain
[36]	SolidWorks	Structural steel	Small and big end	413465/ 511298	Maximum stress
[53]	PTC CREO 3.0	Al 7075	NA	NA	Deformation and Stress and safety factor
[32]	SolidWorks	aluminum alloy, 42CrMo4, Titanium Alloy, Al Metal Matrix	Small end	16190/8821	stress, deformation, strain, and maximum shear stress
[26]	Pro/E	aluminum 6061, 7075, 2014,carbon fiber 280 gsm bidirectional	NA	NA	Displacement, Stress, Strain and Ultimate tensile strength
[46]	Pro/E	Forged Steel	NA	NA	fatigue life, damage, factor of safety, stress biaxiality, fatigue sensitivity
[42]	Pro/E	NA	Big end	NA	Von-mises stress distribution
[40]	NA	45 steel	Small end	118143/NA	Stress, safety factor and fatigue life
[54]	Pro/E	Forged steel	Small end	NA	Stress and deformation
[35]	NA	Steel alloy	Small end	NA	Deformation, stress, factor of safety.
[37]	MSc. ADAMS	C70S6 steel	Piston end	92/NA	Tensile stress and factor of safety
[48]	Pro/E	Forged steel, C-70	Small end	NA	Total deformation, Fatigue life, damage, Factor of safety, Stress biaxiality, strain, and Fatigue sensitivity
[47]	CATIA	Forged steel	Both end	8541/ 4298	Equivalent stress
[41]	CRE-O	Steel	Small end	NA	stress, strain and deformation values
[50]	Solidworks	Steel	Small end	NA	deformation, Von-misses stress, elastic strain and safety factor
[29]	NA	Forged steel, aluminium5083 alloy	Piston end	NA	Deformation and stress
[51]	CATIA V R20	Carbon steel	Small end	389061/262888	Von misses stresses, frequencies with respect to modes; total deformation, buckling analysis, MBD analysis
[39]	Pro/E	Cast iron	Small end	8373/ 16076	von-misses stresses shear stress and strains
[55]	SolidWorks	Alloy steel	Both	43367/ 67038	maximum Von-Misses stresses
[28]	SolidWorks	structural steel, aluminium alloy, titanium, and Magnesium alloy	Small end	18097/10404	deformation, Von-misses stress, elastic strain and safety factor

5. Conclusion

An important piece that makes an engine to function properly is the connecting rod. It is the "backbone" of an engine. Analysis of a connecting rod have been carried out and presented by

different researchers. This article provides a comprehensive study by reviewing some of the vital contributions made to an engine connecting rod from strength analysis to design optimization.
The strength analysis result presented evaluates the tolerance of the connecting rod under certain loading conditions and design constraints with different materials. Optimization of the structure for a target ratio to remove the unwanted portions, which have no use on the structure, have helped in reducing the weight of the connecting rod. Hence, it can be concluded that ANSYS software can be employed by production companies to minimize cost while maintaining quality. Finally, from the revised literatures, we can suggest that in designing an engine connecting rod:

- The material used in the design of the connecting rod should have high strength value with reduced weight due to the demands for a stronger connecting rod
- The characteristics of a H- and I-beam rod can be utilized together in order to achieve the prior suggestion
- The material used in its manufacture should be light without compromising its strength. This can be achieved by using a hybrid of two materials in order to intensify their strength and eliminate their weakness

Acknowledgments

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

References

[1] Singh V, Sumit V, Harish R C, Vishal B and Abhinesh B 2017 *Inter. J. Res. Appl. Sci. Eng. Tech* (IJRASET) **5** 1152-1178
[2] Gurunath S, Vahid J, Ganesh S, Jadhav V and Vinayak D Y 2015 *Stress Analysis of Connecting Rod using FEA* ICRAME2015(GH Raisony COE, Pune)
[3] Swapnil B I and Saifullah K 2014 Modelling and Analysis of Connecting Rod of Two Wheeler (Hero Honda Splendor) *International Journal of Analytical, Experimental and Finite Element Analysis (IJAEFEA)* **3** 38-41
[4] Mohammed M A and Mohamed H 2015 *Analysis of Fatigue Stresses on Connecting Rod Subjected to Concentrated Loads at The Big End* 4th International Conference on Materials Processing and Characterization (India)
[5] Moon K Lee, Hyungyi L, Tae S and Hoon J 2010 *J. Materials & Design* **31** 2796-2803
[6] Adnan A Haider, Akash K, Ajinkya C, Moin K and Suresh P 2018 *Int. Res. J. Eng. Technol.* (*IRJET*) **5** 282-285
[7] Ramani J, Sunil S and Pushpendra S 2014 *Int. J. of Engineering Research and Applic.* **3** 216-220
[8] Suherman, Bambang W, Achmad P, Widodo B Santoso and Arifin N 2016 Modeling and Static *Analysis of a Connecting Rod in Range Extender Engine* International Conference on Sustainable Energy Engineering and Application (ICSEEA) (Jakarta, Indonesia)
[9] Savanoor R, Abhishek P, Rakesh P and Amit R 2014 *International Journal of Mechanical Engineering & Robotic Research* **3** 511-516
[10] NPTEL Course Web IIT 2009 *Module 1: Introduction to Finite Element Analysis*
[11] Roylance D *2011 Finite Element Analysis* Massachusetts Institute of Technology
[12] Shinde and Gurunath V 2015 *Fundamental Concept of Finite Element Method* (India) DACOE, Karad
[13] Nikishkov G 2004 *Introduction to the Finite element method* (Japan) University of Aizu, Aizu-Wakamatsu
[14] Dixit and Uday S *Finite element method: an introduction* (Guwahati, India) Department of Mechanical Engineering, Indian Institute of Technology

[15] Bathe and Klaus J 2014 *Finite Element Procedures* (USA) Prentice Hall, Pearson Education, Inc
[16] ANSYS 2017 *ANSYS Fluent Tutorial Guide* (USA) ANSYS, Inc
[17] Erdogan and Ibrahim G 2015 *The Finite Element Method and Applications in Engineering Using ANSYS* (USA) Springer International Publishing
[18] Nakasone Y, Yoshimoto S and Stolarski T 2006 E*ngineering Analysis with ANSYS Software* (UK) : MPG Books Ltd., Bodmin, Cornwall
[19] ANSYS 2019 Introduction to a finite element analysis program: ANSYS 26 08 2019 http://innomet.ttu.ee/martin/mer0070/wb/ws1/ansys_classic_tutorial_lab1_.pdf
[20] Wang G, Guoqing Z, Yang J, Guan Y, Gong W and Zhang H 2012 *Modal analysis of high-speed spindle based on ANSYS* 7th International Conference on Computer Science & Education (ICCSE) (Melbourne, VIC, Australia)
[21 Kim B S, Lee S H, Lee M G, Ni J, Song J and Lee W 2017 *J. Materials Processing Technology* **187-188** 30-36
[22] Jang G, Lee S H and Jung M S 2002 *J. Sound and Vibration* **251** 59-78
[23] Sayeed A G, Sirajuddin E K and Syed H S 2014 *International Journal of Innovative Research in Science* **3** 16674-16687
[24] Barbero E J 2014 *Finite Element Analysis of Composite Materials Using ANSYS* (US) CRC press, Tylor and Francis group
[25] Muhammad A and Shanono I H 2018 *Nigerian Journal of Technological Research* **13** 14-19
[26] Mohamed A H, Prabhat K S and Arvind S D 2014 *International Journal of Mechanical Engineering and Technology (IJMET)* **5** 57-69
[27] Kuldeep B, Arun L and Mohammed F 2013 *Journal of Innovative Research in Science, Engineering and Technology* **2** 2480-2487
[28] Muhammad A, Ali M A and Shanono H I 2019 *Analysis of a diesel engine connecting rod using different materials* International Conference on Mechatronics, Automation and Cyber-Physical Computer Systems (MAC-2019) (Nigeria).
[29] Savanoor R A, Abhishek P, Rakesh P and Amit R 2014 *International Journal of Mechanical Engineering & Robotics Research* **3** 510-516
[30] Nikhil U T, Nitin D B, Rahul P S and Mahesh M P 2015 *International Journal of Advances in Science Engineering and Technology* **3** 82-86
[31] Talikoti B, Kurbet S N, Kuppast V and Arvind M 2016 *Harmonic analysis of a two-cylinder crankshaft using ANSYS* International Conference on Inventive Computation Technologies (ICICT) (Coimbatore, India)
[32] Ganta K. Reddy and Badde N. Vadithe 2017 *Journal of Research in Advance Engineering Technology* **6** 240-251
[33] Mathew S *What is Von Mises stress* 12 06 2019 http://www.learnengineering.org/2012/12/what-is-von-mises-stress.html
[34] Bari K, Rolfe A, Christofi A, Mazzuca C and Sudhakar K V 2017 *Case Studies in Engineering Failure Analysis* **9** 9-16
[35] Omid M, Mohtasebi S S, Mireei S A and Mahmoodi E 2008 *J. Applied Sciences* **8** 4338-4345
[36] Puran S, Debashis P and Ran V S 2015 *International Journal of Automotive Engineering and Technologies* **4** 245-253
[37] Mohammad R, Mohammad R Asadi, Masoud M and Ahangari H 2011 *Australian Journal of Basic and Applied Sciences* **5** 841-845
[38] Gaba P and Sethi A 2014 *International Journal of Research in Advent Technology* **2** 459-464
[39] Anusha B, Vijaya C and Bhaskar R 2013 *J. Mechanical and Civil Engineering (IOSR-JMCE*) **6** 83-87
[40] Zheng B, Liu Y and Ji L 2010 *Finite Element Analysis and Structural Improvement of Diesel Engine Connecting Rod* Second International Conference on Computer Modeling and Simulation (Sanya, Hainan, China)

[41] Bedse, Dipalee S 2017 *International Journal of Recent Engineering Research and Development (IJRERD)* **2** 203-213
[42] Dilip G G and Metkar R M 2009 *Computer Aided Modelling and Finite Element Analysis of Connecting Rod Fouling on Camshaft of DI Engine of Mahindra and Mahindra Tractor* Second International Conference on Emerging Trends in Engineering & Technology (Nagpur, India)
[43] Yuanchun K and Zhi D 2015 *Dynamic Optimization of Engine Connecting Rod* International Conference on Intelligent Transportation, Big Data & Smart City (China)
[44] Xianjun H, Cuicui T, Dan F, Fuming P and Fuwu Y 2009 *Sensitivity Analysis and Optimization for Connecting Rod of LJ276M Electronic Gasoline Engine* International Conference on Computational Intelligence and Software Engineering (Wuhan, China)
[45] Su Young L, Seong B Lee, Heung S Kim, Tae-gu K, Moon G Kam and Jung W Y 2016 *Engineering Materials:Fracture and Strength of Solids VI* **306-308** 345-350
[46] Pranav G Charkha and Santosh B J *Analysis & Optimization of Connecting Rod* International Conference on Emerging Trends in Engineering & Technology (Nagpur, India)
[47] Kumar, Prakash S Kumar and Kaushik 2009 *J. on Emerging trends in Modelling and Manufacturing* **2** 44-50
[48] Pushpendra K Sharma and Borse R 2012 *International Journal of advance research in Science and Engineering* **1** 1-6
[49] Muhammad A, Ali A M and Shanono H I *2019 Design optimization of a diesel connecting rod* 2nd International Conference on Materials Manufacturing and Modelling (India)
[50] Muhammad A and Shanono I H 2019 *International Journal of Engineering Technology and Science (IJETS)* **6** 24-40
[51] Santhosh R, Shivraj J and Maruthi B H 2015 *International Journal for Scientific Research & Development* **3** 228-232
[52] Yogesh. B Dupare, Raju B Tirpude and Akshay Y B 2015 *International Journal of Modern Trends in Engineering and Research* **2** 215-221
[53] Mohankumar D and Lenin R 2017 *International Journal of Pure and Applied Mathematics* **116** 105-109
[54] Shaik H Saheb, Sampath K and Ramesh A *2014 Design and Analysis of a Connecting Rod for the 117kw Six Cylinders Turbocharged Diesel Engine* Advance Research and Innovations in Mechanical, Material Science, Industrial Engineering and Management – ICARMMIEM (India)
[55] Ganesha R, Nain P K and Pramod K 2014 *International Journal of Engineering Research & Technology (IJERT)* **3**,1875-1878