Mathematical Modelling of Engineering Problems
Vol. 10, No. 3, Month, Year, pp. 1011-1016
Journal homepage: http://iieta.org/journals/mmep

Finite Element Analysis of Crankshaft Stress and Vibration in Internal Combustion Engines Using ANSYS

Mohammed Mousa Azzawi[1], Auday Shaker Hadi[2], Atheer Raheem Abdullah[1*]

[1] Department of Refrigeration and Air Conditioning, Al-Rafidain University College, Baghdad 10066, Iraq
[2] Mechanical Engineering Department, University of Technology – Iraq, Baghdad 10066, Iraq

Corresponding Author Email: atheer_raheem@ruc.edu.iq

https://doi.org/10.18280/mmep.100335

Received: 31 October 2022
Accepted: 13 March 2023

Keywords:
crankshaft, finite element method, modal analysis, ANSYS

ABSTRACT

Internal combustion engines are thermal engines in which a fuel-air mixture undergoes combustion within the combustion chamber. The high-pressure and high-temperature gases generated during this process expand, exerting force upon the pistons and converting chemical energy into mechanical energy. This retraction movement is transformed into a rotational motion that generates the necessary torque to rotate the wheels via the crankshaft. Common examples of these engines include four-stroke and two-stroke piston engines, gas turbines, jet engines, and most rocket engines. This study aims to develop a model of the piston-connecting rod-crankshaft system and investigate the thermal and mechanical stresses resulting from its operation. Utilizing the finite element method in conjunction with ANSYS software, the simulation of thermal and mechanical stresses is carried out, while random vibrations are applied to represent potential imperfections and errors during operation. The SOLIDWORKS program is employed for the design and illustration of the system. Simulation results indicate that maximum vertical stress and shear stress are primarily distributed along the piston heads and connecting rods.

1. INTRODUCTION

The crankshaft is a critical component in internal combustion engines, responsible for converting the reciprocating motion of the pistons into rotational motion through a four-link mechanism [1-3]. Due to its relatively large size and complex geometry [4], the crankshaft is subjected to cyclic loads, which can result in fatigue failure over time. Additionally, torsion and bending stresses arise from cyclic variations in centrifugal inertial forces, alternating inertia forces, and gas pressure. Consequently, optimizing the design and material selection for crankshafts is of paramount importance for researchers, aiming to achieve durability, lightweight construction, and cost-effectiveness while accommodating other operational requirements [5].

Modal analysis is an approach employed to investigate and analyze the dynamical properties of a structure under excitations such as vibrations. Modeling and simulating the dynamic behavior of the crankshaft under tension and stresses generated by the aforementioned factors is essential for its design and manufacturing process. This is typically accomplished using computer simulations and numerical methods, particularly the Finite Element Method (FEM) [5]. In this paper, a crankshaft with specific specifications is modeled and simulated using ANSYS 16.0 to analyze structural stress, thermal stress, and vibrations (modal analysis). These analyses provide valuable insights for the development of an optimal design and manufacturing process for the crankshaft.

Shahane and Pawar [6] focused on optimizing the geometry and shape of the crankshaft, comparing the existing design with an optimized variant that could replace it without modifying the engine block and cylinder head. Meng et al. [7] applied 3D FEM to a 380-diesel engine crankshaft using ANSYS software, primarily calculating the maximum strain and stress in the component. Harris and Birkitt [8] analyzed compressor crankshaft failure, identifying erosion weakness as the leading cause, which originated from localized surface damage. In contrast, Gu and Zhou [9] investigated static structural analysis for determining the maximum stress in a four-cylinder engine crankshaft, while Menacer et al. [10] studied heat transfer in the diesel engine combustion chamber using a two-zone combustion model.

Deshbhrater and Suple [11] conducted a modal analysis of a four-cylinder engine crankshaft using FEM, comparing aluminum alloy and alloy steel, with the aim of identifying the most suitable material based on maximum deformation and stress. Similarly, Murthy and Kollati [12] performed modeling and analysis of a four-cylinder engine crankshaft using FEM. Zheng et al. [13] offered effective theoretical and experimental evidence for optimizing crankshaft structural design through experimental design and the response surface method.

Balamurugan et al. [14] investigated computer-based modeling and optimization of crankshafts by comparing the fatigue behavior of two manufacturing technologies. They created a 3D model of the crankshaft using solid edge software, which was imported into the ANSYS environment for analysis. Optimization involved geometric modifications compatible with the engine under study, resulting in increased fatigue strength and reduced crankshaft costs without altering the engine block and connecting rod.

2. METHODOLOGY

The modal analysis of a 4-cylinder crankshaft is examined utilizing the finite element method during this paper. The analysis is completed on two dissimilar materials which their composition is supported. Three-dimension models of diesel motor crankshaft were created using computer software. The Finite Element Analysis (FEM) software ANSYS was accustomed analyze the vibration modal of the crankshaft. the most stress point and hazardous areas are identified by the analysis of deformation of the crankshaft. the link between the frequency and therefore the vibration modal is clarified by the modal calculation of the crankshaft. The results would offer an important theoretical basis for the optimization and development of motor layout. The crankshaft is one of the foremost significant moving parts of an internal combustion motor. It must be tough enough to soak up the downcast force of the stroke of the piston without excessive deformation that the reliability and lifespan of an IC engine rely on the strength-bearing capacities of its primary components one of which is the crankshaft. And because the engine runs, the facility impulses hit the crankshaft in a zone and afterwards in another. The torsional oscillation occurs when an influence impulse hits a crankpin toward the frontside of the engine and also when the power stroke ends [15]. If not regulated, it can shatter the crankshaft. Load calculation of the crankshaft becomes a fundamental factor to make sure the lifetime of the engine. Beam and accordingly the space shape models were accustomed calculate the strain of crankshaft usually within the past. But the quantity of nodes is proscribed in these models. With the event of computers, more and more formats of crankshafts have employed the finite element method (FEM) to calculate the strain of crankshafts. The applying of numerical simulation for the designing crankshaft helped designers to efficiently improve the event process avoiding the price and constraints of assembling a database of real-world components [16]. Finite Element Analysis allows a reasonable study of incidental mixtures of intake parameters encompassing design parameters and procedure conditions to be examined. A crankshaft could be a complicated continuous configuration. The performance of the vibration of the crankshaft has a vital role in effecting the engine.

3. FAILURE ANALYSIS OF CRANKSHAFT

The exhaustion crack advancement analysis of forged steel diesel motor crankshafts was researched by Guagliano et al. [17]. They exhibited experimentally that with a shaft-like geometry, the crack accumulates rapidly on the open surface while the main part of the crack front becomes straighter. Founded on this study, two techniques were described in relation; the first evaluates a three-dimensional model with a break formed on its cross-section from the inner depth to the outer surface. To deduce the stress intensity characteristics associated with methods I and II, a particularly fine mesh near to the crack ridge, consisting of a large quantity of nodes and components, and substantial computation time is required. The second strategy utilizes a two-dimensional prototype with a vertical break and with tangible crack depth, giving simpler prototypes and less analysis time. Asi conducted a failure analysis of diesel machine crankshafts used in automobiles, which are composed of ductile cast iron. The crankshaft severed into two pieces at the crank pin before the warranty period terminated [18]. The crankshaft has been induction strengthened. An examination of the defective crankshaft was conducted to evaluate its integrity, which included visual inspection, photographic documentation, chemical analysis, micro-hardness measurement, and tensile testing metal assessment. Regions that failed to be examined by surveying electron microscopy were prepared with an EDX setup. The findings demonstrate that fatigue is the primary cause of the crankshaft failure. An additional crack detection technique was inaugurated by Baxter [19]. He worked on crack detection using a modified edition of the gel electrode procedure. This method can specify both main fatigue cracks and the dispersion of secondary locations of less serious fatigue failures. The crankshaft of an internal combustion engine alters the rectilinear action of the piston into rotary motion. This process is used to navigate a car or any other appliance where the crankshaft is employed. Crankshafts amass a wide span of applications from elegant single-cylinder lawn mower engines to relatively huge multi-cylinder marine engines. The crankshaft is an element intended to lengthen the existence of the machine and/or automobile. As a high-speed rotating unit, its lifespan encompasses millions, if not billions, of redundant loading cycles. Therefore, crankshafts are usually constructed for endless life. Jensen revealed in her analysis on V-8 vehicle crankshafts that the engine's inertial and gaseous loads establish multi-axle stress circumstances in the shape of bending and torsion. The crankshaft has been assessed for defect and fatigue life of 106 cycles has been deduced. There is renewed demand for elements with less mass, more power, and lower creation expenses. The automotive business frequently pursues to enhance fuel economy by utilizing lighter pieces, including optimized geometries and materials, while lessening assembling expenses [20]. One way to decrease production costs is to employ various materials and/or procedures. There is a preference for crankshafts that are lighter weight and cheaper to provide, while maintaining the specified fatigue performance. The foremost common processes utilized in the bulk of the trade are casting and forging. Normally, ductile iron is employed within the casting whereas steel is employed with in the forging procedure. Micro alloyed steels will be accustomed eliminate the necessity for a heat treatment process in some forged steel applications. Testing was performed at a temperature which was monitored and conserved to ± 2°C. The humidity was also monitored utilizing a precision hydrometer. A test of monotonic tension was conducted on each of the two materials. One sample was assessed for every material. The testing was performed in line with ASTM Standard.

3.1 Finite Element Method (FEM)

Finite Element Analysis (FEA) or finite element method (FEM) could be used as a numerical method for solving difficulties of engineering and mathematical physics. Helpful for dilemmas with sophisticated geometries, loadings, and material properties where analytical solutions cannot be achieved or aren't adequately precise enough thus it's best to use numerical strategies [21]. Analytical solution: stress analysis for beams, trusses, and other simple structures is applied using dramatic simplification and idealization of the real problem: – mass focused at the center of gravitational attraction – beam facilitated as a line segment (same cross-section). Design is predicated on the calculations of the optimized configuration and an outsized ratio (1.5-3) given by knowledge. Computational solution: When the structure geometry is further complicated; the accurateness prerequisite is a lot bigger. We require comprehending the physical behaviors of a sophisticated object (strength, heat transfer ability, fluid flow, etc.), to anticipate the performance and nature of the design; to calculate the safety margin; and to specify the shortcoming of the structure accurately; and to recognize the optimal configuration with assurance.

The working principle of the FEM is a discretization issue that is solved as follows: 1) FEM slices a system into several elements (chunks of the system). 2) Then reconnects components at "nodes". 3) This procedure outcomes in a set of concurrent algebraic equations. The equations attained are effortlessly solved with a few steps nonetheless the precision usually relies on the kind of approximation utilized and the quantity of elements the structure was split up into. 4) FEM manipulates the notion of piecewise polynomial interpolation. 5) By attaching elements concurrently, the field amount becomes interpolated over the whole system in a piecewise manner. 6) A set of coexisting algebraic equations at nodes.

3.2 FEM thermal stress and temperature distribution analysis

Work all around the world has been conducted to formulate and ensure accuracy of the FEM strategy in solid and fluid body thermal analysis. Computer protocols exist to deal with two-dimensional transient or steady-state heat conduction with inner heat production and therefore the subsequent boundary conditions: convection or an indicated temperature along the threshold, and radiation or a continuing heat flux normal to that. ANSYS, defined during this paper, can handle the 2D case, further because the three-dimensional case under the similar boundary conditions, with radiation, convection, or normal flux over the surface (instead of along the boundary) of the 2D continuum.

As theoretical approach, for the 3-D case, the equation for 3-dimentional (x, y, z) transient heat flow (changing with time) in a uniformly shaped plate of thickness d is given as in Eq. (1):

$$\frac{\partial}{\partial x}\left(K_x\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(K_y\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(K_z\frac{\partial T}{\partial z}\right)+Q=\rho.c.\frac{\partial T}{\partial t} \quad (1)$$

where, the needed temperature has to fulfill the subsequent boundary limitations. 1) Certain temperature or T along boundary portion B. Restricted heat energy flow q_n into B or the surface S as in Eq. (2):

$$q_n=-k\frac{\partial T}{\partial n} \quad (2)$$

2) Heat convection between any region and its boundary regions along segment B or S as in Eq. (3):

$$h\left(T_{boundary}-T\right)=k\frac{\partial T}{\partial n} \quad (3)$$

And 3) radiation between the area and its boundaries along any segment B or over S, or radiation between two skin components i and j as in Eq. (4):

$$\sigma.F_{ij}\left(\epsilon_i T_i^4-a_i T_j^4\right)=-k\frac{\partial T}{\partial n} \quad (4)$$

Eq. (1) along with all of its boundary conditions can specify the matter with a very unique and easy way. Employing the science of calculus of variations, an alternate form of the equations may be obtained and thru it, it's possible that the complete problem could also be communicated in one equation—an integral equation in variational shape:

$$X=\iiint\begin{pmatrix}\frac{K_x}{2}\left(\frac{\partial T}{\partial x}\right)^2+\frac{K_y}{2}\left(\frac{\partial T}{\partial y}\right)^2+\\ \frac{K_z}{2}\left(\frac{\partial T}{\partial z}\right)^2-Q.T+\rho.c.\frac{\partial T}{\partial t}\end{pmatrix}dv+\iint q_n.Tds+ \quad (5)$$
$$\iint q_r.Tds+\iint h\left(T^2-T_{boundary}.T\right)ds$$

As computational approach by Euler's theorem, this function incorporates a minimum embodied by the previous equation with its boundary limitations. Diminishment of X needs an exact distribution of the temperature T to seek out this, the area is subdivided into a limited quantity of triangular components and Eq. (2) is also restated as: $=\sum X_e$. Where, the summation is confiscated over all ingredients e, and X_e represents the functional applied to every finite component. Assuming the temperature field is linearly allocated throughout each component which implies the approximation only get better when the element volume tends to zero.

$$T=a+bx+cy+dz=(1\quad x\quad y\quad z)\begin{pmatrix}a\\ b\\ c\\ d\end{pmatrix} \quad (6)$$

3.3 FEM structural loading stress analysis

The FEM could be a method for the numerical value solution of the equations that govern the issues found in climate. Usually, the attitude of nature is often theorized by equations asserted in differential or integral shape. For this justification, the FEM is known in mathematical circles as a numerical method for solving partial differential or integral equations. Generally, the FEM allows users to get the advancement in space and/or time of one or more variables exemplifying the demeanor of a physical system. When remarked the examination of systems the FEM could be a powerful technique for calculating the displacements, stresses and strains in a very large and complicated structure under a collection of loads. This is specifically what this paper is geared toward. The structural investigation should lead the way in the planning of and sizing of the engine organs and provide a high level of confidence. The investigation should be an a vital a part of the planning and testing procedure, thus minimizing design struggle and time by eradicating redesign influenced by failure during structural validation testing. A significant advantage of executing stress analysis is that the capacity to work out design sensitivities and to perform trade investigations. Thus, effective optimization of the structure is often attained, improving trustworthiness while decreasing cost and weight. It is crucial for the investigation to be cautious, i.e., the failure load foresaw should be but the particular load the structure can resist. This can be essential in picture of the uncertainties within the analysis hypotheses and also the differences within the applied loads and material properties within normal bounds. The concept of an all-around safety factor or SF is inaugurated to account for numerous uncertainties and also the limit loads are boosted in ratio to the SF (Max Load=Safety Factor x Limit Load) [22, 23]. A common SF value employed for the last word failure of flight configurations is 1.4. Further, a yield SF generally of 1.25 is chosen to stop structural devastation or destructive yielding during structural testing or flight. Additional safety factors could also be utilized for fittings, castings, etc. to account for similar suspicions. The SF regulations may shift depending on the project. In accessory to applying a safety factor, care should be taken to administer a cautious examination using lower limits for assessing the structure's load carrying ability. This may arise in a more credible design; nonetheless, there'll be a weight liability. It should even be reported that the analysis effort decreases with increasing conservativeness. Hence, at the beginning of the calculation, characteristics like weight criticality of the structure, suspicions in data, and usable time for examination should be evaluated.

4. THE MECHANISM PARTS' DESIGN BY SOLIDWORKS SOFTWARE

The crankshaft and its attachments including the connecting rode and the piston were designed according to the average design standards of ships' engines. The designed parts are illustrated as in Figures 1 and 2.

The giogetry and configuration dimensions are listed in Table 1.

Table 1. Engine configuration

Name of part	Dimensions (m)	
	Diameter	Long
Main bearing journal	0.64	0.38
Nose of pulley	0.32	1.10
Main journals	0.64	0.38
Flywheel mounting flange	1.44	0.15
Counterweight	1.6	0.15
Wrist Pin	0.34	0.38
Bearing Inserts	0.66	0.38
Rod		0.87*0.25*0.38
Small End	0.44	0.38
Piston Pin	0.34	

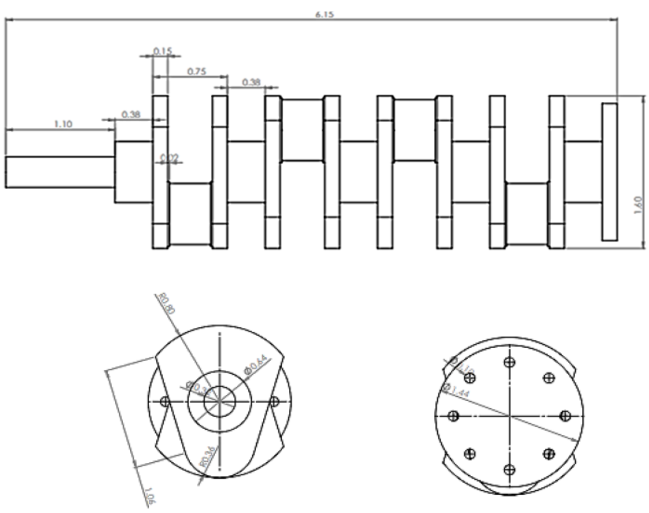


Figure 1. Top, front and right profiles of the crankshaft

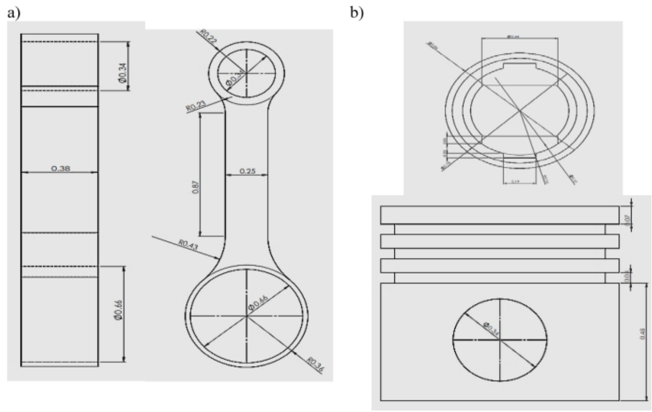


Figure 2. a) Top and right profiles of the connecting rode; and b) Top and front of the piston

5. RESULTS AND DISCUSSION

In this section the results and steps of the simulation of the mechanism (pistons-connecting rod-crankshaft) based on FEM in ANSYS environment. Several stresses are taken into consideration while simulating the mechanism.

In the first stage, the solid body is discretizationed into finite elements (mesh of finite elements) (see Figure 3).

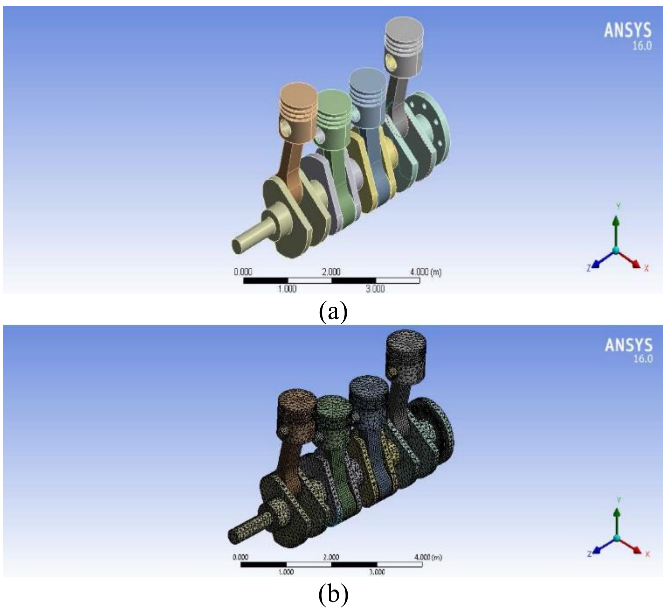
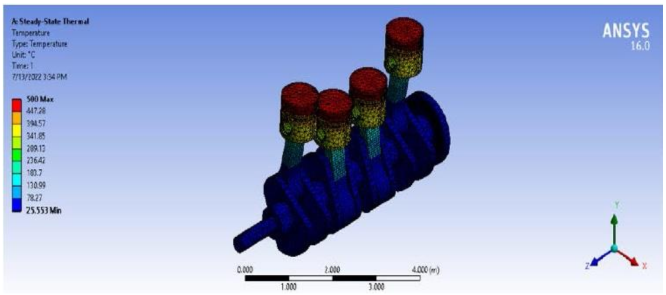


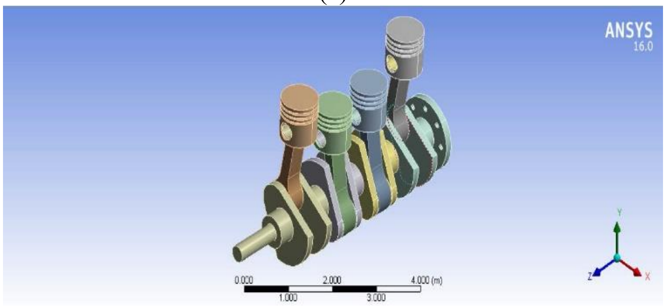
Figure 3. The body: a) before discretization; and b) after discretization

The software implements a number of algorithms (for determining the suitable types of the finite element). Those algorithms depend on the maximum volume of the element (in this study it's 1 cm). By experiments, the results did not change significantly (only about ± 0.02%).

The thermal distribution is simulated by determining the thermal source (the upper surface of the piston head), and its temperature is 500 celisuis (Figure 4). Moreover, the ambient temperature is 22 celisuis. The simulation ends when the body's temperature reaches the equilibrium state (when there is no heat exchange between the body parts or the surrounding atmosphere.



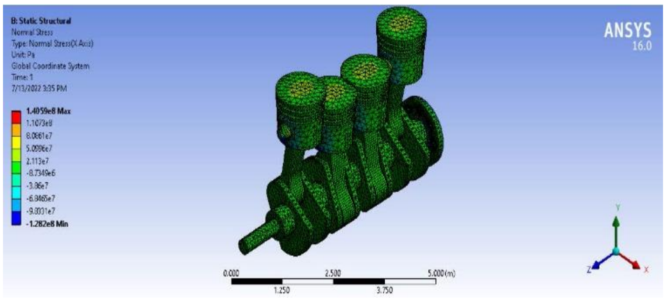
(a)



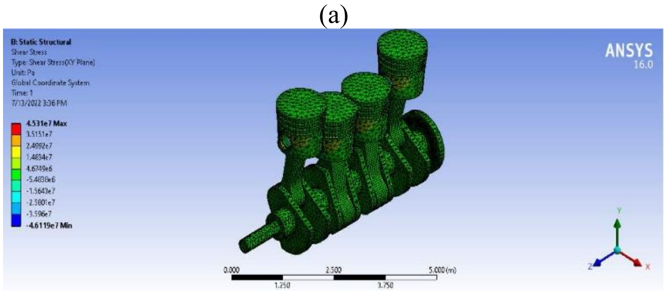
(b)

Figure 4. a) Thermal distribution through the body, measured in celisuis; and b) The body before the thermal distribution

For the simulation of the mechanical stress, The temperature variations through the body parts cause thermal stresses, and it was simulated with applying a momet of 500 N.m with repect to the crank shaft axis which is parallel to the axiz Z. Based on the results the maximum vertical stress equals to 1.4×10^{8} Pa and the shear stress equals to 4.61×10^{7} Pa in absolute value distributed mainly along the pistons' heads and connecting rods (see Figure 5).



(a)

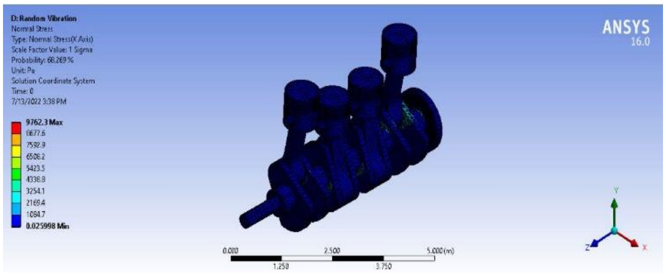


(b)

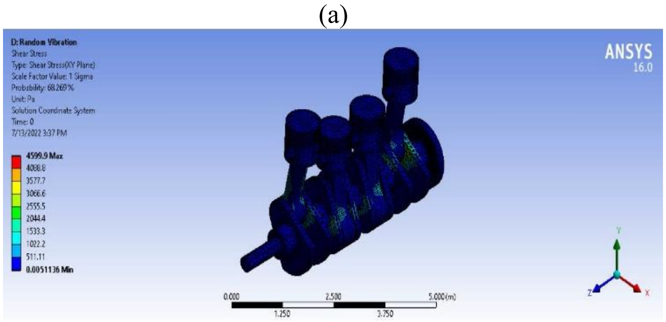
Figure 5. Stress measured in Pa: a) Vertical stress; and b) Shear stress

Grujic et al. [24] noticed from that the highest stresses occur at the transition between the crankpins and the counterweight the bending load of the crankshaft. However, it can be seen that the maximum stress is 26.944 MPa, while the maximum deformation is 0.31704 mm.

For the simulation of the mechanical stresses caused by random vibrations, because the engine works under real circumctnaces (not therotical) and unexpected incedints that will generate random vibrations. Thoes vibrations were simulated in the form of applied forces in a random manner, and at different frequencies (100-500 Hz). The results indicate a maximum vertical stress of 9762.2 Pa and a maximum shear stress of 4599.9 Pa in absolute values distributed mainly across the pistons' heads and connecting rods (see Figure 6).



(a)



(b)

Figure 6. Stress measured in Pa: a) Vertical stress; and b) Shear stress

6. CONCLUSIONS

In this study, a model of a mechanism piston mechanism - connecting rod-crankshaft of an internal combustion engine was designed, and the thermal and mechanical stresses to which the mechanism is exposed during the operation of the engine were studied and analyzed. This was done depending on the finite element method and using Ancis in order to implement the computer simulations of these stresses and also in order to apply random vibrations to the mechanism and thus determine the resulting response. The element ending with a scale of 1 has been selected, which ensures high accuracy of dynamic analysis. The results indicated that the maximum vertical stress equals to1.4×10^{8} Pa and the shear stress equals to 4.61×10^{7} Pa in absolute value distributed mainly along the pistons' heads and connecting rods when simulating the mechanical stresses. Whilst, when applying random vibrations ranging from 100 to 500 Hz , it was evicent that a maximum vertical stress of 9762.2 Pa and a maximum shear stress of 4599.9 Pa in absolute values distributed mainly across the pistons' heads and connecting rods.

REFERENCES

[1] Zhao, F.F. (2021). Modeling and thermal-mechanical coupling analysis of piston in car engines. Annales de Chimie - Science des Matériaux, 45(1): 83-92. https://doi.org/10.18280/acsm.450111

[2] Arapatsakos, C., Karkanis, A., Anastasiadou, C. (2015). The load and the gas emissions measurement of outboard engine. International Journal of Heat and Technology, 33(4): 221-228. https://doi.org/10.18280/ijht.330430
[3] Sheth, R., Kothari, T. (2020). Design and analysis of cast stainless steel crankshaft in solidworks software. International Research Journal of Engineering and Technology, 7(12): 1953-1956.
[4] Sandya, K., Keerthi, M., Srinivas, K. (2016). Modeling and stress analysis of crankshaft using fem package ANSYS. International Research Journal of Engineering and Technology, 3(1): 687-693.
[5] Uluocak, İ., Yavuz, H. (2016). Dynamic simulation of a crankshaft. Advances in Automobile Engineering, 5(2): 143.
[6] Shahane, V.C., Pawar, R.S. (2017). Optimization of the crankshaft using finite element analysis approach. Automotive and Engine Technology, 2(1): 1-23. https://doi.org/10.1007/s41104-016-0014-0
[7] Meng, J., Liu, Y., Liu, R. (2011). Finite element analysis of 4-cylinder diesel crankshaft. International Journal of Image, Graphics and Signal Processing, 3(5): 22-29.
[8] Harris, W., Birkitt, K. (2016). Analysis of the failure of an offshore compressor crankshaft. Case Studies in Engineering Failure Analysis, 7: 50-55. https://doi.org/10.1016/j.csefa.2016.07.001
[9] Gu, Y., Zhou, Z. (2011). Strength analysis of diesel engine crankshaft based on pro/e and ansys. 2011 Third International Conference on Measuring Technology and Mechatronics Automation, Shanghai, pp. 362-364. https://doi.org/10.1109/ICMTMA.2011.661
[10] Menacer, B., Khatir, N., Bouchetara, M., Larbi, A.A., Belhout, C. (2020). The heat transfer study in the diesel engine combustion chamber using a two-zone combustion model. Mathematical Modelling of Engineering Problems, 7(4): 614-620. https://doi.org/10.18280/mmep.070414
[11] Deshbhratar, R.J., Suple, Y.R. (2012). Analysis & optimization of crankshaft using FEM. International Journal of Modern Engineering Research, 2(5): 3086-3088.
[12] Murthy, B.D.N.S., Kollati, N.S.T. (2013). Modelling, analysisand optimization of crankshaft. International Journal of Engineering Research and Technology, 2(9): 749-753.
[13] Zheng, B., Zhang, J., Lei, J. (2022). Crankshaft optimization based on experimental design and response surface method. Mathematical Problems in Engineering, 2022: 9286969. https://doi.org/10.1155/2022/9286969
[14] Balamurugan, C.M., Krishnaraj, R., Sakthivel, M., Kanthavel, K., Deepan Marudachalam, M.G., Palani, R. (2011). Computer aided modeling and optimization of crankshaft. International Journal of Scientific and Engineering Research, 2(8): 1-6.
[15] Fu, Z., Liu, G. (2014). Simulative analysis for static and dynamic behaviour of qt8002 ductile iron crankshaft. Optoelectronics and Advanced Materials-Rapid Communications, 8(1-2): 115-120.
[16] Chaudhari, J.C., Barjibhe, R.B. (2016). Experimental and numerical analysis of crankshaft used in hero honda splendor motorcycle. International Journal of New Technology and Research, 2(7): 83-94.
[17] Guagliano, M., Terranova, A., Vergani, L. (1993). Theoretical and experimental study of the stress concentration factor in diesel engine crankshafts. Journal of Mechanical Design, 115(1): 47-52. https://doi.org/10.1115/1.2919323
[18] Asi, O. (2006). Failure analysis of a crankshaft made from ductile cast iron. Engineering Failure Analysis, 13(8): 1260-1267. https://doi.org/10.1016/j.engfailanal.2005.11.005
[19] Baxter, W.J. (1993). Detection of fatigue damage in crankshafts with the gel electrode. International Congress and Exposition, Detroit, Michigan, SAE Technical Paper 930409. https://doi.org/10.4271/930409
[20] Asiri, S. (2022). Modeling and analysis of automotive engine crankshaft made of composite and functionally graded materials. Advances in Materials Science and Engineering, 2022: 4005368. https://doi.org/10.1155/2022/4005368
[21] Jagota, V., Sethi, A., Kumar, D.K. (2013). Finite element method: An overview. Walailak Journal of Science & Technology, 10(1): 1-8.
[22] Ang, Y.Z., Ku, P.X. (2021). Study on failure analysis of crankshaft using finite element analysis. MATEC Web of Conferences, 335: 03001. https://doi.org/10.1051/matecconf/202133503001
[23] Patel, R.K., Ghosh, G.K., Pradhan, S.R. (2018). Fatigue and modal analysis of crankshaft using ansys software. 19th ISME Conference on Advances in Mechanical Engineering, Jalandhar.
[24] Grujic, I., Glisovic, J., Stojanovic, N., Davinić, A., Pesic, R., Narayan, S., Kaisan, M.U. (2018). Stress analysis of the crankshaft of ic engine. Machine Design, 10(2): 69-72.