Nigerian Research Journal of Engineering and Environmental Sciences 11(1) 2026 pp. 282-289
p ISSN: 2635-3342; e ISSN: 2635-3350

Nigerian Research Journal of Engineering and Environmental Sciences
Journal homepage: www.rjees.com

RJEES

Original Research Article

Fluid Fluent Analysis on the Effect of Combustion Gases on the Thermal Performance of the Cylinder Block of a 0.67 hp Petrol Generator

***Aliemeke, B.N.G., Obi, E.C., Idenobemhe, J.T., Muhammed, M.T. and Onokpasa, S.**

Department of Mechanical Engineering, Auchi Polytechnic, Auchi, Edo State, Nigeria.
*aliemekebng@auchipoly.edu.ng

http://doi.org/10.5281/zenodo.21048066

ARTICLE INFORMATION

Article history:
Received 04 Jun. 2026
Revised 20 Jun. 2026
Accepted 22 Jun. 2026
Available online 30 Jun. 2026

Keywords:
Cement
Combustion
Finite element
Fluid Fluent
Cylinder temperature
Pressure

ABSTRACT

Fluid Fluent analysis on the effect of combustion gases on the thermal performance of the cylinder block of a 0.67 hp petrol generator has been successfully carried out. It had become very stressful analysing internal combustion engines using trial-and-error experimental techniques. It was denying the automobile technology world the necessary understanding of the appropriate thermal conditions for engine components. The development of computational fluid dynamics and the finite element method is a breakthrough in the field of fluid mechanics. The Fluent fluid technique applied showed that the thermal performance was high, as obtained in the very high convergence given in the scaled residual plot. The governing equations were satisfied and duly applied to yield optimal thermal conditions. The pressure was higher at the inlet of the component, while low pressure was noticed at the fins. Similarly, a high temperature of 350 K was experienced at the inlet and a lower temperature of 300 K around the fin.

1. INTRODUCTION

The invention of the internal combustion engine is an evolving phenomenon that has brought about the advancement of modern engineering technology in the 20th century.

The design and construction of machines have always been conducted through empirical and trial by error technique (Asiri, 2022). The machines had so many shortcomings in terms of operation and maintenance. Early internal combustion engines operated at low speeds, had poor power outputs, air pollution inclined and operational noise (Andersen et al., 2024). It was noticed that as the engine became faster in operation, combustion gas production increased geometrically (Arapatsakos et al., 2015).

In the past, combustion analysis in internal combustion engines depended heavily on cumbersome experimental approaches, which made in depth study of the design ratio mixture of gases relatively challenging (Zhu, *et al*., 2024). Early engineers used various laboratory stoichiometric gauges for detection of mixing ratio of combustion gases (Oladeinde, *et al*. 2016). While these methods presented valuable insight into fluid behavior in the engine it was noticed to be time-consuming and labour-intensive. Manual or physical testing could only attempt to measure superficial details of the component with some difficulty in accessing the complex engine geometries.

Engine design has become very sophisticated and complex as such multicylinder configurations, integrated cooling system and generator engine intricacies are not sufficient enough to be investigated by mere physical testing (Deshbhratar and Suple, 2012). Thus, it is necessary to embark on computational fluid dynamics to be able to bring up solutions (Balamurugan, et al. 2011).

The development of computational fluid dynamics and the finite element method is a great revelation in the field of fluid mechanics where it allows complex structures to be determined (Fu and Liu, 2016).

The introduction of numerical methods, particularly the Finite Element Method (FEM), transformed engine vibration and combustion analysis (Grujic, *et al.* 2018). FEM allows complex structures of the combustion reactants and products to be discretized into finite nodes and elements so as to model the cylinder block geometries and the fluid flow within the component (Gu and Zhou, 2021).

The finite element method provides realistic material properties placed alongside boundary conditions so as to enable design engineers to predict favourable responses while targeting high accuracy (Harris and Birkitt, 2016).

Fluid Fluent tool analysis of the Finite element analysis gained prominence in the 1970s for its ability to analyze engine geometry in setting up the material properties and boundary conditions to determine pressure, temperature, wall temperature and heat flux by displaying the values in the parametric contour region and volume rendering outcome display (Meng and Liu, 2021). Early application of computational fluid dynamics occurred in the analysis of the fluid system of a radiator system (Menacer, et al., 2020). Temperatures of the car radiating system had been modeled in the past to determine the effect of water temperature in the vehicle radiator (Zhu, *et al*. 2024). Using computational fluid dynamics, optimization of fluid flow system in the vehicle radiating system had been carried out to determine the effect of organic coolants on the hot water generated in the radiator (Jagota, *et al,* 2013). It has been noticed that the tool analysis system had the tendency of identifying temperature sensitive domain within the radiator by selecting suitable materials and boundary conditions that may optimize the geometry and the thermal conditions of the fluid (Murthy and Kollati, 2013).

In modern engineering, finite element analysis is a standard technique for both design and analysis. In modern engineering practice, finite element fluid fluent has become a standard tool for both design and diagnostics (Shahane and Pwar, 2017). ANSYS Mechanical software allows for virtual testing of engine blocks under various boundary conditions and loading scenarios, minimizing reliance on costly physical prototypes (Sheth and Kothari, 2020). This project utilizes Fluent flow analysis to a generator engine block using modern FEA tools, linking historical and theoretical knowledge to practical engineering application (Aliemeke and Iyore, 2021). The study aims to provide insights into improved reliability, operational safety, and durability of generator systems, particularly in environments where uninterrupted power supply is critical (Sandya, et al. 2016).

From an academic perspective, this project integrates theoretical knowledge from strength of materials, dynamics of machines, vibration analysis, and finite element analysis, providing a practical learning experience that strengthens the student's analytical and engineering competence (Patel, et al., 2018). Moreover, the methodology and results of this study can serve as a reference framework for future research on fluid flow performance optimization in internal combustion engines (Ebhojiaye and Sadjere, 2017).

This study is highly significant for mechanical engineers and designers as it demonstrates the practical application of computer-aided simulation tools, such as ANSYS, in analyzing complex fluid systems without engaging in costly and time-intensive experimentation (Zhao, 2021).

B.N.G.Aliemeke et al. / Nigerian Research Journal of Engineering and Environmental Sciences
11(1) 2026 pp. 282-289

2. MATERIALS AND METHODS

2.1. Materials

The materials used in this study are AutoCAD software, ANSYS software, a vernier caliper, micrometer screw gauge.

2.2. Methods

The cylinder block was designed using mathematical formulae and graphically designed using AutoCAD software to produce an isometric and third-angle orthographic drawing. The isometric drawing produced was exported to the ANSYS Fluent environment (Uluocak and Yawuz, 2016).

3. RESULTS AND DISCUSSION

3.1. Graphical Modeling

The graphical modeling was done using AutoCAD. The isometric drawing of the cylinder block and third angle orthographic projection are shown in Figures 1 and 2, respectively. The values of the model were obtained from Table 1 as given in Aliemeke and Oladeinde (2020),

Table 1: Cylinder block designed data for the parameters

Parameters	Designed value	Recommended range
Bore diameter (mm)	44.30	43-50
Cylinder Length (mm)	56.58	50-75
Stroke length (mm)	49.20	45-76.5
Cylinder thickness (mm)	3.02	3.0-6.0
Brake power (kW)	3.591	1.5-5.0
Indicated power (kW)	4.788	1.5-5.0
Mean effective pressure (MPa)	42.47	-
Cylinder wall thickness(mm)	3.02	2.5-6.0
Engine torque (Nm)	11.49	-
Bolt thickness (mm)	5.51	4-7

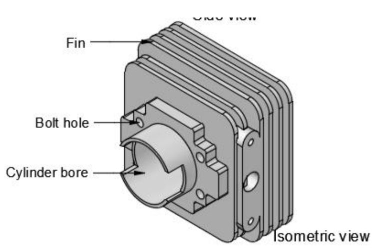

Figure 1: Isometric of the cylinder block

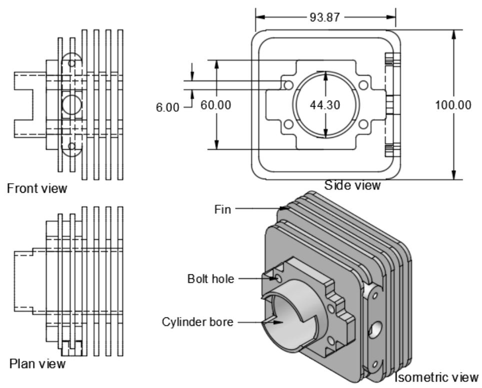

Figure 2: Third angle projection of the cylinder block

3.2. ANSYS Fluent Fluid Flow Finite Element Analysis

The ANSYS finite element software was used to carry out the effect of combustion gases on the engine. The imported modeled cylinder block into the ANSYS environment is shown in Figure 3. It was meshed to prepare it for proper simulation. It had 86701 and 421322 as nodes and elements. The meshed components is shown in Figure 4.

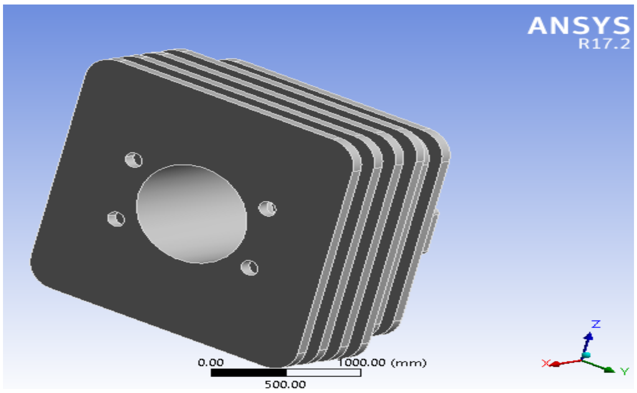

Figure 3: Exported cylinder block to the ANSYS environment

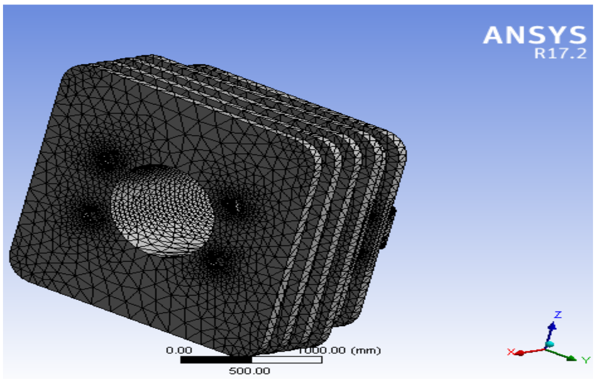

Figure 4: Meshed cylinder block to the ANSYS environment

3.2.1. Statistical analysis

The scaled residual shown in Figure 5 indicates that the simulation was effectively carried out. The continuity and the velocity lines connote adequate uniformity in the fluid interaction and utmost convergence attained in analyzing the cylinder block (Zheng, *et al.* 2022). It also, showed that the governing equations were duly satisfied and applied. The drag plot shown in Figure 6 shows that the iteration became uniform from the 10th iteration. Again the simulation set up was completely in order. The lift plot shown in Figure 7 indicates that the iteration got uniform from the 10th iteration. This was like a confirmation to that obtained from the scaled residuals and lift plots.

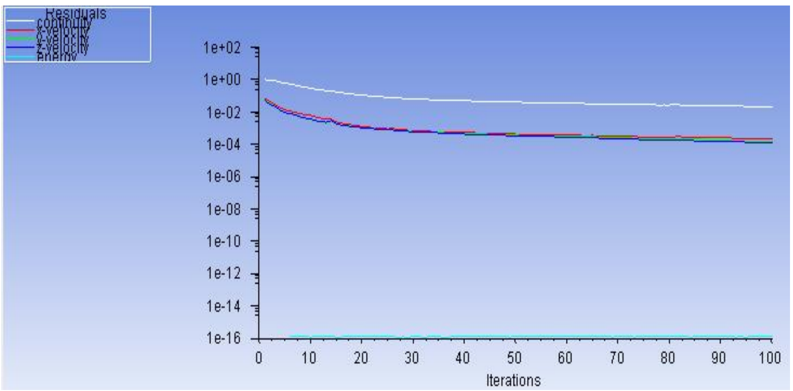

Figure 5: Scaled residual plot in the ANSYS environment

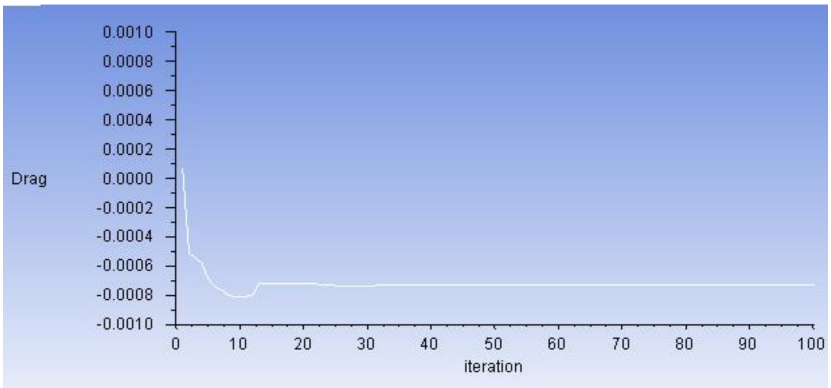

Figure 6: Drag plot

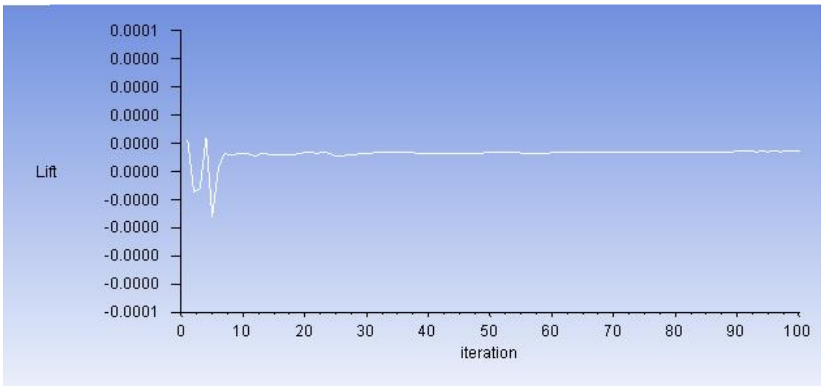

Figure 7: Lift plot

3.2.2. ANSYS analysis on the thermal conditions of the cylinder block

The component heat flux contour is shown in Figure 8. The contour shows that high thermal conductivity was noticed around the cylindrical bore while the fins maintained a lower thermal conductivity as shown in Figure 8.

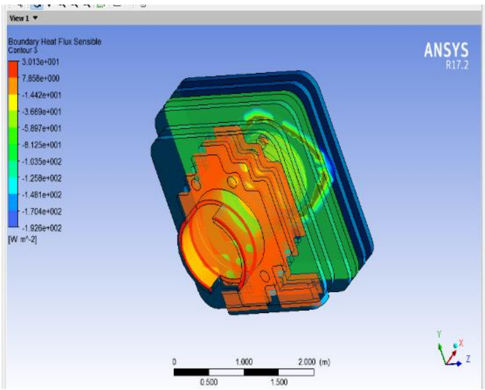

Figure 8: Heat flux plot the ANSYS environment

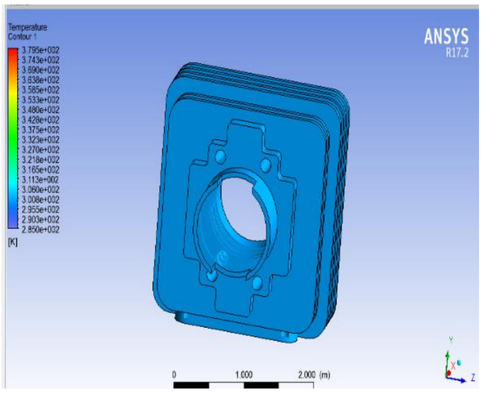

Figure 9: Temperature contour plot

The temperature and temperature volume rendering contour contours are shown in Figures 9 and 10 respectively. It shows that as high as 350 K was noticed around the cylindrical wall while the fins have 300 K. The pressure contour at outlet is shown in Figure 11. It showed that the pressure at outlet wall was 0.0005 Pa while the inlet cylindrical wall experienced higher pressure of about 0.0029 Pa as shown in Figure 12. It showed that more pressure was exerted on the inlet of the cylindrical wall as presented in Aliemeke and Oladeinde (2020). The pressure volume rendering is shown in Figure 13. It indicates that higher pressure is as a result of the combustion gases at inlet is exhibited at the entrance. Also, the inner wall temperature plot shown in Figure 14 shows that around 320 K inner wall temperature was obtained from the effect of the burnt gases in the combustion chambers.

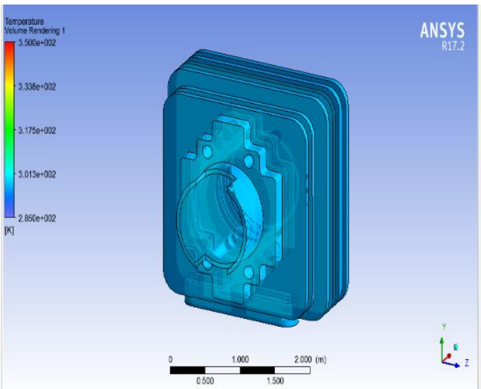

Figure 10: Temperature volume rendering contour plot

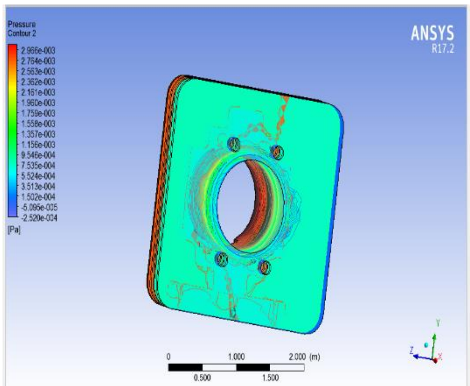

Figure 11: Pressure contour plot at outlet

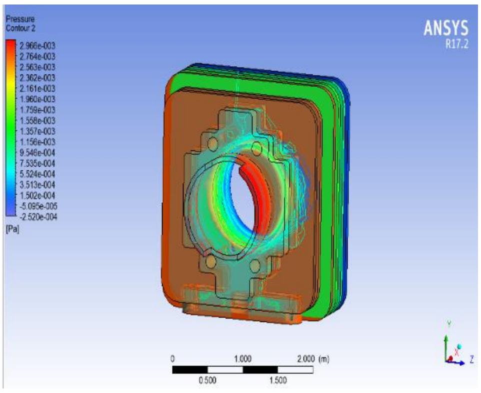

Figure 12: Pressure contour plot at inlet

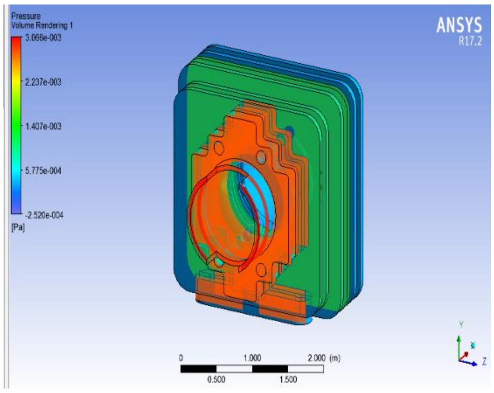

Figure 13: Pressure volume rendering plot the ANSYS environment

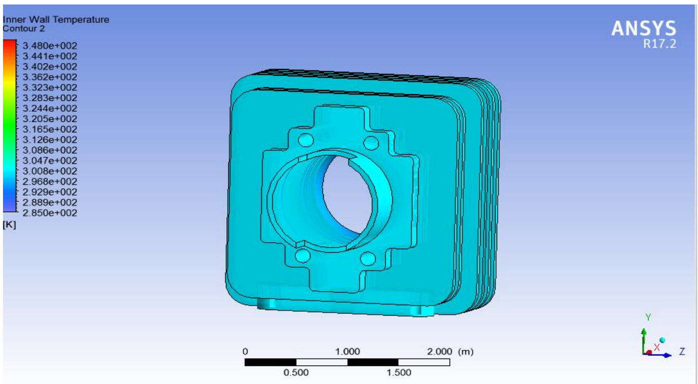

Figure 14: Inner wall temperature contour plot

4. CONCLUSION

Fluid Fluent analysis on the effect of combustion gases on the thermal performance of cylinder block of 0.67 hp petrol generator has been successfully carried out. In the past combustion analysis in internal combustion engines depended heavily on stressful experimental techniques which made profound study of designing ratio mixture of gases very difficult. In the recent past, engineers used various laboratory stoichiometric gauges for detecting mixing ratio of combustion gases. Although,-these method presented valuable insight into fluid behavior in the engine it was discovered to be time consuming and labour intensive. The development of computational fluid dynamics of the finite element method is a major feat in the field of fluid mechanics where it allows the complex structures to be determined by simulation before been applied in the practical field. The introduction of numerical methods, particularly the Finite Element Method, transformed engine vibration analysis. In this study it was noticed that much concentration of pressure was at the inlet of the component while less pressure was noticed at the fins. Similarly, the high temperature of 350 K was experienced at the inlet and lower temperature of 300 K around the fin.

5. CONFLICT OF INTEREST

There is no conflict of interest associated with this work.

REFERENCES

Aliemeke, B. N. G. and Oladeinde, M. H. (2020). Design of 0.67Hp Petrol generator pistons *Nigerian Journal of Technology*, 39(3,), pp. 839-843.

Aliemeke, B. N. G., and Iyore, T. A. (2021). Finite element analysis of an aluminium shaping tool and titanium workpiece. *International Journal of Engineering Science and Application*, 5(3), 45–54.

Andersen, P. R., Henriquez, V. C. and Aage, N. (2024) Validity of Numerical models for Visco Thermal Losses in Structural Optimization for Micro-acoutics, Journal Sound Vibration, 5(2), 117-122

Arapatsakos, C., Karkanis, A., and Anastasiadou, C. (2015). The load and the gas emissions measurement of outboard engine, *International Journal of Heat and Technology, 33(4), 221-228.*

Asiri, S. (2022). Modeling and Analysis of Atomotive Engine Components Made of Composite and Functionally Graded Materials, *Advances in Materials Science and Engineering, 20(2), 20-25.*

Balamurugan, C. M., Krishnaraj, R., Sakthivel, M., and Kanthavel, K., (2011). Computer Aided Modeling and Optimization of Crankshaft. *International Journal of Scientific and Engineering Research, 2(8), 1-6.*

Chaudhari, J. C., and Barjibhe, R. B.(2016). Experimental and Numerical Analysis of Crankshaft Used in Hero Honda Motorcycle, *International Journa of New Technology and Research, 2(7), 83-94.*

Deshbhratar, R. J., and Suple, Y. R. (2012). Analysis and Optimization of Engine Components using FEM, *International Journal of Modern Engineering Research, 2(5), 3086-3088.*

Ebhojiaye, R. S. and Sadjere, G. E. (2017). Design of a Spark ignition Aluminium Engine Cylinder Block, *Pacific Journal of Science and Technology,*18(1) pp. 22-30.

Fu, Z., and Liu, G.(2016). Simulative analysis for Static amd Dynamic Behavoiur of qt28002 Ductile Iron Crankshaft and Advanced Material-Rapid Communications, 8(1), 115-120.

Grujic, I., Glisovic, J. and Kaisan, M. U. (2018). Stress Analysis of Internal Combustion Engine, Journal for Machine Design, 10(2), 69-72.

Gu, Y. and Zhou, Z. (2021). Strength Analysis of Diesel Engine Components based on ANSYS,

Third International Conference on Measuring Technology and Mechatronics Automation,Shanghai pp. 362-364.

Harris, W. and Birkitt, K. (2016). Analysis of Failure of the Crankshaft of the Internal Combustion Engine, *International Journal of New Technology and Research, 2(7), 120-127.*

Jagota, V., Sethi, A., and Kumar, D. k. (2013). Finite Element Method, An Overview, *Walaikak Journal of Scieence and Technology, 10(1), 1-8*

Meng, J. and Liu, Y. (2021). Finite Element Analysis of 4-Cylinder Diesel Internal Combustion Engine, *International Journal of Image, Graphics and Signal Processing, 3(5), 22-29.*

Menacer, B. and Khatir, N. , Bouchetara, M. (2020). The haet transfer in the Diesel Engine combustion Chamber, *Mathematical Modeling of Engineering Problems,* 7(4), 614-420.

Murthy, B. D. N. and Kollati, N. S.T. (2013). Modeling, Analysis and Optmization of Engine cylinder, *Internaional Journal of Engineering Research and Technology*, 2(9), 749-753.

Oladeinde, M. H.,Ebojaiye, R.S. and Augoye, A. K (2016). "Manufacture of engine parts for a Single Cylinder 0.67 hp gasoline generator by the application of Reverse Engineering" *Journal of National Association of Mathematical Physics*, 33(3),pp. 387-394.

Patel, R. K. , Ghosh, G. K. and Pradhan, S. R. (2018). Fatigue and Modal Analysis of Pistons and Crankshaft using ANSYS Software, *19th ISME Conference on Advances in Mechanical Engineering*, Jalandhar.

Sandya, K. Keerthi, M. ,and Srinivas, K. (2016). Modeling and Stress Analysis of Crankshaft Using FEM Package ANSYS. *International Research Journal of Engineering and Tech*nology, 3(1), 687-693.

Shahane, V. C., and Pwar, R. S. (2017). Optimization of Internal Combustion Engine Components Using Finite Element Analysis Approach, *Automotive and Engine Technology, 2(1), 1-23.*

Sheth, R. and 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*.

Uluocak, I. and Yawuz, H. (2016). Dynamic Simulation of a Crankshaft. *Advances in Automobile Engineering, 5(2), 143*.

Zhao, F. F. (2021). Modeling and thermal-mechanical coupling analysis of piston in car engines. *Annales De Chime-Science Des Materiaux, 45(1), 83-92*.

Zheng, B. , Zhang, J. and Lei, J. (2022). Cylinder block and Crankshaft Optimization based on Experimental Design and Response Surfce Methodology Method. *Mathematical problems in Engineering*, 22(1), 56-61.

Zhu, L. Yi, W. and Wang. M. (2024). Sun Experimental and Numerical Study on the on the shear performance of Friction type high strength bolted connection used for CLT-steel hybrid walls, Engineering Structurec 6(2), 306-312.