Nigerian Journal of Engineering Science Research (NIJESR).
Vol. 5, Issue 2, pp.40-50, June, 2022

ISSN: 2636-7114
Journal Homepage: https://www.iuokada.edu.ng/journals/nijesr/

Simulation of Flow Parameters of Exhaust Gas of a Prototype Engine for Generation of Power from Exhaust Gas of Gasoline Generator

[1*] Zubair, A. M., [2] Musa, N. A., [3]Okegbile, O.J and [4]Muhammadu, M.M

[1]Department of Mechanical Engineering, Federal University of Technology, Minna, P.M.B. 65, Minna, Niger State, Nigeria,
mustaphazubair5@gmail.com
[2]Department of Mechanical Engineering, Federal University of Technology Minna, P.M.B. 65, Minna, Niger State, Nigeria,
madonick1@yahoo.com
[3] Department of Mechanical Engineering, Federal University of Technology Minna, P.M.B 65, Minna, Niger State, Nigeria,
ojokegbile@gmail.com
[4] Department of Mechanical Engineering, Federal University of Technology Minna, P.M.B 65, Minna, Niger State, Nigeria,
masin.muhammadu@futminna.edu.ng

***Corresponding Author: Zubair Adeboye Mustapha; mustaphazubair5@gmail.com (08035377919)**

Manuscript History
Received: 02/04/2022
Revised: 10/06/2022
Accepted: 20/06/2022
Published: 30/06/2022

Abstract: *According to literature, low-income earners, who make up the majority of the Nigerian economy, primarily employ gasoline generators with low power ratings, which do not match their lighting and appliance power requirements. The pressure energy of the generator's exhaust gas was investigated for usage in order to improve the output power of low power rating generators. However, before designing or developing a prototype engine that uses exhaust gas to generate power, a thorough understanding of the exhaust gas's flow properties is required, since this will determine the viability of using exhaust gas for power generation. As a result, a prototype engine was created in this study, and the flow through it was simulated using SolidWorks 2020. The average flow characteristics, such as pressure, velocity, density, and temperature, were 130830.97N/m^2, 10.56m/s, 0.90Kg/m^3, and 174^{0}C, according to the simulation results. With these magnitudes of average exhaust gas flow characteristics, on which the power generated by the exhaust gas is based, it can be determined that a reasonable power may be generated from the exhaust gas of a gasoline generator, which will serve as a boost to the generator's power. This research makes a significant contribution to knowledge by revealing the potentials of the pressure energy of gasoline generator exhaust gas for generating electricity.*

Keywords: *Prototype Engine, Simulation, Pressure, Velocity, Temperature, Exhaust Gas*

INTRODUCTION

Internal combustion engine performance as a result of changes in gas exchange components cannot be predicted in advance because the interactions between the acoustic responses of the various elements involved (intake or exhaust systems, turbomachinery, etc.) are quite strong (Peat *et al.* 2006., Munjal, 1987) stated by Fajardo, (2012). He went on to say that CFD simulations allow researchers to understand flow behavior and quantify critical flow parameters like mass flowrates and pressure drops, as long as the CFD tools have been appropriately validated against experimental results. The following researchers' work emphasizes the necessity of exhaust gas flow simulation in internal combustion engines. Modeling and study of four stroke gasoline engine exhaust gas systems were reported by Mohamad *et al.* (2017). The temperature of the manifold, the pressure of the manifold, the exhaust piping system, and the velocity of the exhaust gas were all investigated in CFD. The pressure and velocity contours, as well as the streamline, were used to display the results of the CFD simulation. They came to the conclusion that the maximum velocity of flue gases is obtained near the exhaust manifold's outlet because the pressure is low there, causing the gases to rush out of the duct's exit. Furthermore, the study of CFD analysis on the consequences of exhaust gas behaviour generated in the exhaust system of a 4-stroke gasoline engine supplied knowledge to the internal combustion engine designer and researcher. Vamsi and Chuen-Sen (2019) used SolidWorks flow simulation computer program to study exhaust gas flow through a corrugated concentric heat exchanger with twisted tape and its changes in order to find the best design. By comparing simulation findings with well-known correlations and field data, the program's reliability was confirmed. They discovered that an annularly corrugated tube heat exchanger with a twisted tape and rod inserted had the greatest increase in heat transfer rate. The improvement was limited by the need to keep the pressure drop of the exhaust gas going through the heat exchanger under control. When compared to plain tube, annularly corrugated tube heat exchangers without twisted tapes, and annularly corrugated tube heat exchangers with plain twisted tape, this exchanger transports around 279.3 percent, 126.17 percent, and 35.1 percent more heat. Based on best results for a 120 kWe diesel generator, they found that constructing an annularly corrugated tube heat exchanger with a plain twisted tape insert could save 2,266 gallons of fuel per year, while using a rod twisted tape insert could save up to 7622 gallons per year. They calculated that this is a savings of almost 4252 gallons more than the current heat recovery system can provide. CO2 emissions were also reduced by up to 43 metric tons per year.

To determine the gas temperature profile in the exhaust pipe, Brito *et al.* (2015) published a heat transfer model for the exhaust gas of a diesel power generator. The numerical methodology for solving the mathematical model was established utilizing a finite difference method approach for energy equation resolution and temperature profile determination when turbulent fluid flow and changing fluid characteristics were included. The simulation was performed for engine operating at loads ranging from 0 to 40 kW. The findings of the model were compared to those obtained using the multidimensional Ansys CFX program, which was used to solve the turbulent fluid flow governing equations. They claimed that the results for temperature profiles in the exhaust pipe demonstrated a close match between the mathematical model produced and the actual temperature profiles. The findings of simulations of exhaust gas flow through a ceramic carrier inserted in the exhaust system of an engine with direct fuel injection were presented by Fuc *et al.* (2019). For the simulation of non-engine CI and SI exhaust gas systems, they employed AVL Fire After treatment software. They investigated the passage of exhaust gases through a filter whose volume was determined by the displacement volume of a turbocharged combustion engine, and found that the temperature of the exhaust gases and the carrier itself fluctuated. Bodin (2013) looked at the flow in the exhaust port, adjacent to the valve, and in the exhaust manifold.

He stated that because the flow at the port could be transonic, he first investigated numerical modelling of such a flow in a simpler geometry, such as a bump in a wind tunnel. His findings demonstrated that transonic flow is quite sensitive to tiny changes in boundary conditions in general. He claimed that in the transonic flow regime, flow in the wind tunnel is invariably highly unsteady, with self-excited shock oscillations and unsteady boundary-layer separation. The researcher used one- and two-point correlations as well as dynamic mode decomposition (DMD) to investigate the relationship between separation zone and shock dynamics, and discovered that there was a definite link between separation bubble dynamics and shock oscillation. In the wind tunnel, the outlet pressure was changed on a regular basis with a small amplitude to test susceptibility to periodic disturbances. He discovered that low-amplitude oscillations induced hysteretic behaviour in the mean shock position as well as the development of shocks with a wide range of patterns. He stated that a study of a model exhaust port revealed that the flow at the exhaust port is transonic at realistic pressure ratios. Furthermore, the wake behind the valve stem, as well as inertial forces and the pressure differential in the port, form two pairs of vortex structures downstream of the valve plate. Mantilla *et al*. (2009) opined that unsteady gas flow theory can be used for simulating a spark ignition internal combustion engine's exhaust system, using pressure waves. Moreso there is a great influence exerted by pressure wave movement on flow through the engine and therefore on its final performance. Using Kiva4, Lungu *et al*. (2021) looked into the performance parameters of a spark ignition engine, specifically the relationship between performance, exhaust gas temperature, and speed. The test data to validate the kiva4 simulation results was conducted on a three-cylinder, four-stroke Volkswagen (VW) Polo 6 TSI 1.2 gasoline engine, according to the researchers. As a result, three distinct experiments were carried out. Variations in exhaust gas temperature were studied in one set by adjusting engine load while maintaining constant engine speed. In another experiment, the temperature changes of exhaust gas were investigated by idling the engine and adjusting the speeds. Variations in exhaust gas temperature under a constant load with varying engine speeds were the subject of a third test. A basic grid/mesh generator, K3PREP, was used to produce an itape17 file with a 45° asymmetrical mesh to analyze fluctuations in exhaust gas temperatures under test settings. This was predicated on the symmetry of the engine's combustion chamber, which was used in the experiments. As a result, simulations were run using the input parameters determined during the tests. Simulations using the kiva4 code revealed that the engine's performance characteristics are very predictable. This was demonstrated by the significant agreement in simulation findings when compared to test data under the specified test circumstances. According to the authors, a percentage error of only 2% to 3% was noted between experimental data and simulation results using the kiva4 code.

For the assessment of flow characteristics, thermal features, and minimum back pressure, Bajpai *et al*. (2017) investigated the performance of a four-stroke four-cylinder gasoline engine exhaust manifold using three distinct fuels - gasoline, alcohol, and LPG. Creo2.0 was used to model the manifolds, which was followed by meshing and analysis in ANSYS. They found that the LPG fuel has the lowest back pressure, although the temperature and velocity are roughly the same for all three fuels (gasoline, alcohol, and LPG). As a result, LPG might be considered a viable alternative to gasoline in terms of manifold backflow. When the exhaust temperature of a gasoline engine surpasses 200 degrees Celsius, according to Durat *et al.* (2013), the catalysts become active. For the entire exhaust pipe, they completed a three-dimensional transient CFD analysis. They claimed that the results of the CFD analysis were quite close to those of the experimental data. CFD analysis in the transient regime was also used to find an appropriate catalyst position. They also used different Nusselt number correlations from the literature to study the heat transfer phenomena analytically. More specifically, the analytic results were compared to the experimental data. They claimed that over a wide range of Reynolds numbers, each connection produced respectable findings.

The exhaust pipe, according to Xu *et al.* (2016), is a crucial component of a gasoline engine. It is one of the major technologies in multi valve engine development since its structure and performance have a direct impact on engine power, economy, and emissions. SolidWorks flow simulation was utilized to study a 1.5L gasoline engine exhaust pipe in order for the researchers to test the theoretical design. Pressure and velocity were studied selectively at the three-way catalytic converter and the oxygen sensor. The internal flow is in a laminar flow state, and the sensor position is suitable, according to the CFD simulation findings. They came to the conclusion that the design is reasonable and capable of accomplishing the design aim. Jang *et al.* (2018) conducted a fundamental investigation into the intake and exhaust flow of a range-extended electric vehicle equipped with small engine generators for battery charging in order to determine the design point for high power at a medium engine speed. For a 0.6-liter naturally aspirated (NA) gasoline engine, test factors included intake valve open (IVO) time, exhaust manifold geometry, and intake runner length. In their investigation, they discovered that the influence of IVO timing on engine performance was minor. The extended intake runners and the geometry (shape) of the exhaust manifold influenced torque and BSFC (brake specific fuel consumption) and contributed to increase engine performance at medium engine speeds. Chatterjee (2017) investigated the flow characteristics of exhaust gases via several types of mufflers in depth (e.g., absorptive, reactive and resonating). The researcher used PRO-E to create geometric models and ANSYS FLUENT 14 to analyze them. The pressure, velocity, and turbulence contours were plotted in order to optimize the muffler design based on known thermodynamic characteristics. The researcher presented numerous hybrid designs for a muffler and even a combination of mufflers based on the modification in these characteristics. The need to boost the electrical power output of a gasoline generator by utilizing the generator's exhaust gas has been established by Zubair (2022) and many other researchers. However, a full study of the flow dynamics of exhaust gas is essential before building or creating a prototype engine that uses exhaust gas to generate power, as this will decide the viability of employing exhaust gas for power generation. SolidWorks 2020 was utilized in this study to model and simulate flow through the prototype engine and determine these flow characteristics.

Mathematical Model for the Simulation of Flow of Exhaust Gas of Internal Combustion Engine

When the equations governing fluid flow and determining velocity, temperature profiles, and other parameters are studied in the pipe, it leads to complicated computations and a difficult analytical solution. Numerical methods such as the finite difference method (FDM) and the finite volume method (FVM) can help to solve this problem. FDM is a numerical approach for analyzing heat transport and fluid flow dynamics (Mohamad *et al*, 2017).

Governing Equations

The law of conservation of mass for a compressible fluid flow is expressed as the continuity equation (Bodin, 2013)

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho_{uj}\right) = 0 \qquad (1)$$

Where, t is the time, and states that the change of mass per unit volume in a point is zero. Newton's second law states that the rate of change of momentum on a particle is equal to the sum of the forces acting on it. Formulating this law for a fluid gives the equation for conservation of momentum per unit volume (ρui) at a point.

However according to Jianmin and Shuiting (2014) as stated by Mohamad *et al*. (2017) under the condition of multidimensional compressible steady flow, the mass and momentum conservation equation are as follows

$$\frac{\partial}{\partial x_j}\left(\rho_{uj}\right) = 0 \quad (2)$$

$$\frac{\partial(\rho u_j u_i)}{\partial x_j} = -\frac{\partial P}{\partial x_{ij}} + \frac{\partial \tau_{ij}}{\partial x_j} + F_b \quad (3)$$

The terms from the left-hand side of equation 3 represent the rate of change of momentum by convection (the second). On the right-hand side are the pressure forces (first term), the effect of viscous stresses (second term), and the body forces (when considered) and Mohamad *et al*. (2017) referred to it as a source term which represent the catalytic converter resistance. The momentum equations are usually called the Navier-Stokes equations (Bodin, 2013). According to Mohamad *et al* (2017), τ_{ij} is the stress tensor, for Newtonian flow described as follows:

$$\tau_{ij} = 2\mu\left(S_i - \frac{1}{3}\frac{\partial u_k}{\partial x_k} S_{ij}\right) - \overline{\rho u_\iota\, u_J} \quad (4)$$

Where μ is the molecular dynamic viscosity coefficient; δ_{ij} is Kroneker number; $-\overline{\rho u_\iota\, u_J}$ is Reynold's stress tensor. The fluid deformation rate tensor is given by the following formula (Mohamad *et al*. (2017):

$$S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial n_j}{\partial x_i}\right) \quad (5)$$

Turbulence Model (Ɛ)

Using a standard k- Ɛ model to calculate the Reynold's stress to solve the flow control equations above (Jianmin and Shuiting 2014) as stated by Mohamad *et al*. (2017)

$$\overline{\rho u_\iota\, u_J} = -2\mu S_i + \frac{2}{3}\left(\mu_i \frac{\partial u_k}{\partial x_k}\, \rho k\right)\delta_{ij} \quad (6)$$

where μ_τ is the turbulent viscosity, given by the following formula:

$$\mu_\tau = \frac{C\mu\rho k^2}{\varepsilon} \quad (7)$$

k- Ɛ are turbulent kinetic energy and turbulent energy dissipation respectively. Their transport equations are

$$\frac{\partial}{\partial x}\left(\rho\mu_j\, k - \frac{\mu_{eff}}{\delta_k}\frac{\partial k}{\partial x_j}\right) = \mu_\tau S_{ij}\frac{\partial u_i}{\partial x_j} - \rho\varepsilon - \frac{2}{3}\left(\mu_\tau \frac{\partial u_i}{\partial x_i} + \rho k\right)\frac{\partial u_i}{\partial x_j} \quad (8)$$

$$\frac{\partial}{\partial x}\left(\rho\mu_j\, \mathcal{E} - \frac{\mu_{eff}}{\delta_{\mathcal{E}}}\frac{\partial \mathcal{E}}{\partial x_j}\right) = C_{\mathcal{E}1}\frac{\mathcal{E}}{k}\left\{\mu_\tau S_{ij}\frac{\partial u_i}{\partial x_j} - \frac{2}{3}\left(\mu_\tau \frac{\partial u_i}{\partial x_i} + \rho k\right)\frac{\partial u_i}{\partial x_j}\right\} - C_{\mathcal{E}2}\rho\frac{\mathcal{E}^2}{k} + C_{\mathcal{E}4}\rho\mathcal{E}\frac{\partial u_i}{\partial x_i} \quad (9)$$

In the formula above,

$$\mu_{eff} = \mu + \mu_\tau \quad (10)$$

The empirical coefficients $C_\mu, \delta_k, \delta_\varepsilon, C_{\varepsilon 1}, C_{\varepsilon 2}, C_{\varepsilon 4}$ have been determined by Jianmin and Shuiting (2014) to be 0.09, 1.0, 1.22, 1.44, 1.92 and -0.33 respectively as reported by Mohamad *et al* (2017).

According to Bodin (2013), the energy equation for compressible flows can be formulated in terms of the total enthalpy of a particle:

$$\frac{\partial(\rho ho)}{\partial t} + \frac{\partial(\rho u_i ho)}{\partial x_i} = \frac{\partial p}{\partial t} + \frac{\partial q_i}{\partial x_i} + \frac{\partial(u_j \tau_{ij})}{\partial x_i} + Q + W_{ext} \tag{11}$$

Where $h_0 = h + \frac{1}{2}(u_1{}^2 + u_2{}^2 + u_3{}^2)$ (12)

q_i is the heat flux. W_{ext} is the work of external volume forces, h_0 is the total enthalpy and Q is an external heat source. The heat flux q_i can be related with the local temperature gradient through Fourier's law of heat conduction.

$$q_i = -k\frac{\partial T}{\partial x_i} \tag{13}$$

where k is the thermal conductivity. The left-hand side of equation 11 contains the change in ρho of the fluid particle due to time and convective transport. On the right-hand side, the first term accounts for the work done on the particle generated by the time rate of change of the pressure, the second term represent the rate of heat addition to the particle while the third term accounts for work done by (viscous) surface forces.

MATERIALS AND METHODS

2.1 Simulation Procedure

The prototype engine, which featured the exhaust system of the gasoline generator, was designed using SolidWorks 2020 software. On the window interface, the SolidWorks 2020 software icon was selected from the desktop menu, followed by the selection of a new part. The sketch tab was then selected, which activated all of the design tools, which were then utilized to create the prototype engine. Fig. 1 and Fig. 2 show the assembly drawing and exploded view of the planned prototype engine, respectively.

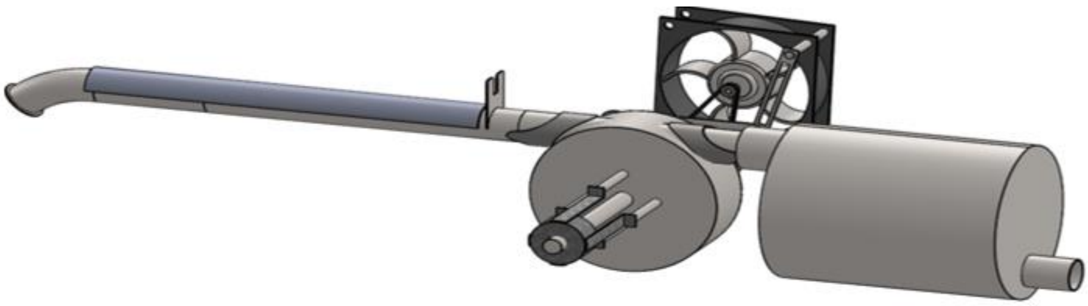

Fig. 1 The assembly drawing of the prototype engine

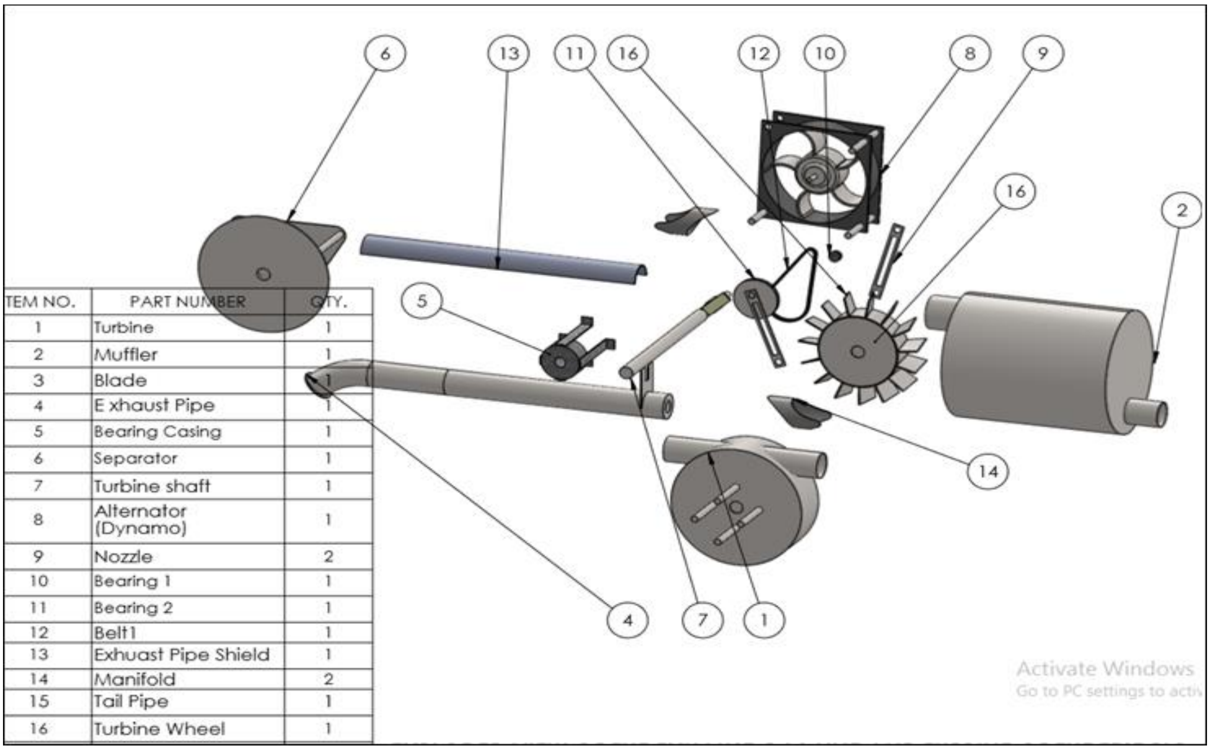

Fig. 2 The exploded view of the prototype engine.

Design tools were deactivated after the prototype engine's design was completed by clicking on the exit drawing on the window interface. The flow simulation tab was examined, and all of the simulation tools were engaged and presented. A wizard option was selected, which brought up a pop-up window. The simulation parameter selections were displayed after clicking on a new project option. The initial or input flow parameters of the exhaust gas used for the simulation were;

Pressure = 1.3bar

Velocity = 22.9m/s

Temperature = 327^{0}C and

Density = 0.592Kg/m^{3}

So, after selecting all of these settings, clicking the completion icon, the computational domain was automatically constructed around the planned or modeled prototype engine. The computational domain was altered to a two-dimensional plane before performing the simulation. The input and outlet boundary conditions were established, global goals were chosen, and the run icon on the window interface was clicked. Flow trajectories including exhaust gas pressure, velocity, temperature, and density contours or distributions were generated as part of the simulation findings. Figs. 3, 4, 5, and 6 show the contour plots of the pressure, velocity, density, and temperature of the exhaust gas flow through the prototype engine, respectively.

RESULTS AND DISCUSSION

Fig. 3 shows the pressure of exhaust gas contour of the simulated flow through the prototype engine.

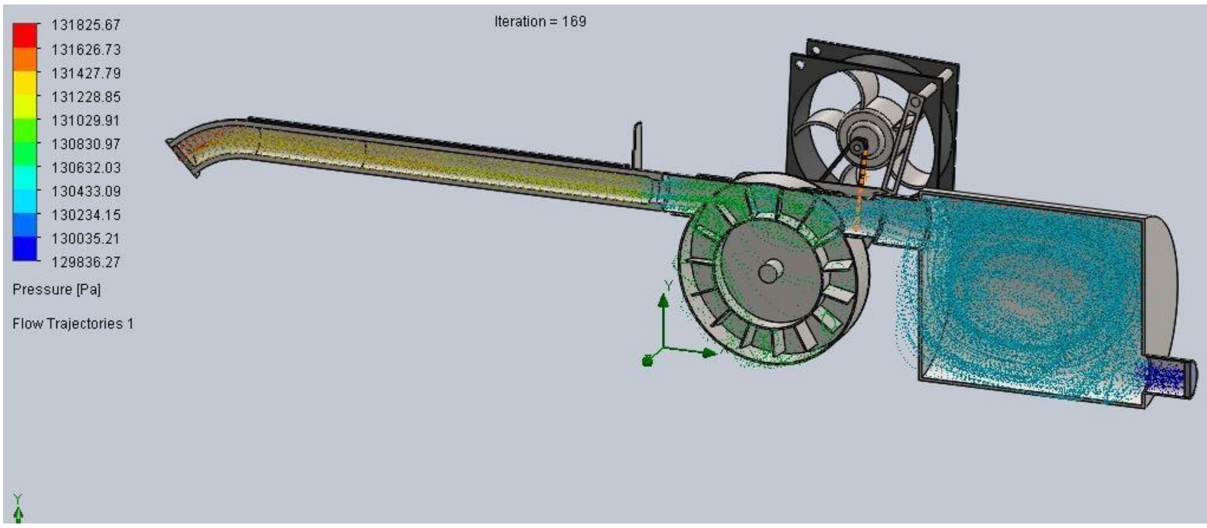

Fig. 3 Pressure of exhaust gas contour of the simulated flow through the prototype engine

The different colours in Fig. 3 represent different magnitudes of exhaust gas pressure, while the fluid pressure varied continually along the flow line, as seen in the aforementioned figure. Mshelia (2015) had a comparable experience simulating fluid flow in the exhaust of a four-cylinder internal combustion engine and Venkata (2004) had a similar experience simulating fluid flow in internal combustion engines and engine manifolds using wave action modelling. This is a sign of the exhaust gas's non-uniform and erratic flow characteristics. Before imparting on the turbine blade, the exhaust gas pressure fell. This is due to the converging nozzle in the flow line having an effect. The simulated average. The average simulated pressure of the exhaust gas was found to be 130830.97N/m^2. Fig. 4 shows the velocity of exhaust gas contour of the simulated flow through the prototype engine.

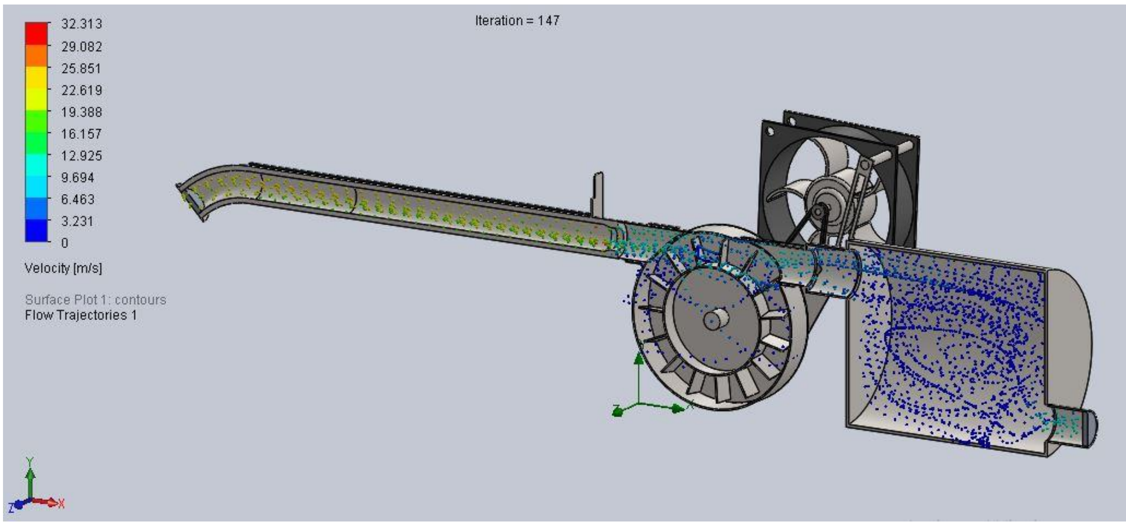

Fig. 4 Velocity of exhaust gas contour of the simulated flow through the prototype engine

The colour illustration in Fig.4 shows that the velocity of the exhaust gas varied throughout the flow line, indicating that the flow was non-uniform and unstable. This fluctuation in exhaust gas velocity was also observed by Kong (2021) in his 1D–3D approach for quick numerical flow analysis and more specifically by Aziz *et al.* (2021) who did flow characterization in an exhaust manifold of a single cylinder internal combustion engine.

The exhaust flow velocity was high before impacting the turbine blades, then fell and increased at the exhaust system's tailpipe. In the prototype engine, the average simulated velocity of the exhaust gas or stream was determined to be 10.58m/s. Fig. 5 depicts the density of exhaust gas contour of the simulated flow through the prototype engine.

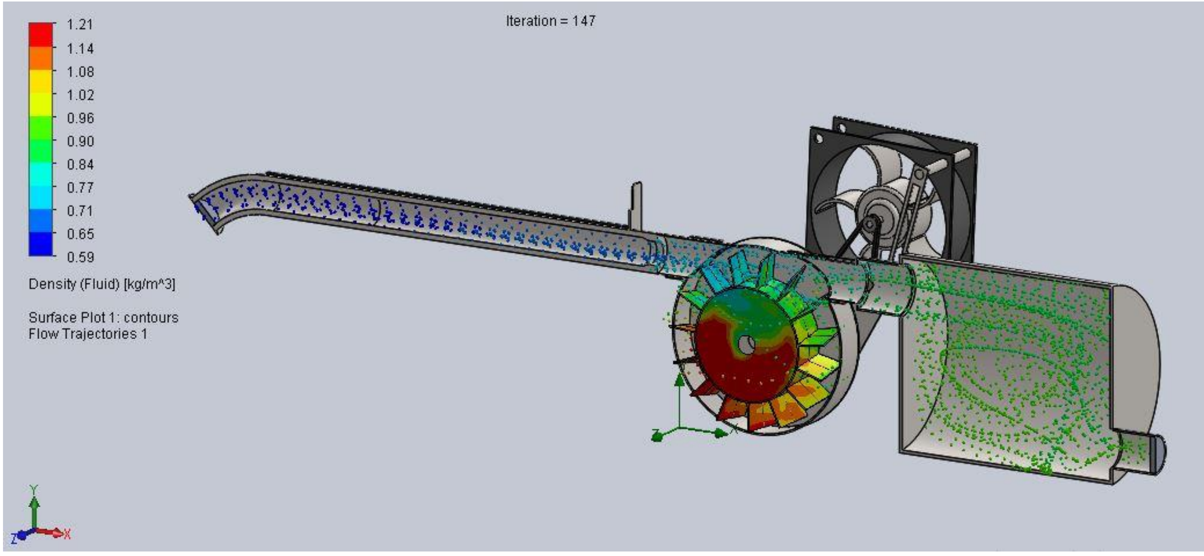

Fig. 5 Density of exhaust gas contour of the simulated flow through the prototype engine.

The density of the exhaust gas fluctuated throughout the flow line, as shown in Fig. 5, indicating that the exhaust gas flow in the prototype engine was non-uniform and non-steady. At the inlet of the exhaust, the density was very low, but it grew before the exhaust gas impinged on the turbine blades, as well as at the blade, and then declined steadily through to the tail pipe. This is comparable to what Mshelia (2015) experienced when simulating fluid flow in a four-cylinder internal combustion engine's exhaust. The prototype engine's average simulated density of exhaust gas was determined to be 0.897kg/m^3. Fig. 6 shows the temperature of exhaust gas contour of the simulated flow through the prototype engine.

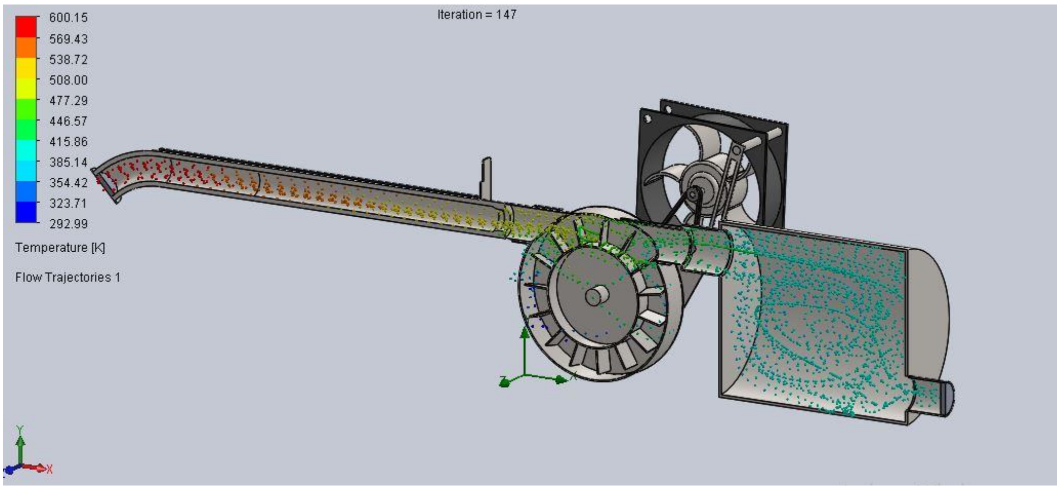

Fig. 6 Temperature of exhaust gas contour of the simulated flow through the prototype engine

The temperature of the exhaust gas was very high at the input of the flow line, fell before hitting the turbine blade, and dropped even more in the exhaust system's tailpipe, as shown in Fig. 6. This change in exhaust gas temperature is consistent with what Brito *et al*. (2015) found in his research. The average simulated temperature of the exhaust gas in the prototype engine was 447K (174^0C).

CONTRIBUTION TO KNOWLEDGE

This study has established the potentials of employing the pressure energy of exhaust gas of gasoline generator for generating electricity which can serve as a boost to the output power of gasoline generator.

CONCLUSION

The flow through the prototype engine for power generation from gasoline generator exhaust gas has been simulated. The pressure, velocity, density, and temperature of the flow varied along the flow line, indicating that the flow was non-uniform and unstable. It is possible to generate a reasonable amount of power from the exhaust gas of a gasoline generator, based on the magnitude of average pressure and velocity of flow of the exhaust gas, on which the power generated is dependent. The power that will provided by the prototype engine will supplement the electricity produced by the gasoline generator.

ecosystems in the area. The economic benefits of the dam will largely offset the aggregate cost of assets that will be consumed by the dam construction.

CONFLICT OF INTEREST

There is no conflict of interest for this research work.

REFERENCES

Aziz, N.A., Rahman, M.T.A., Amin, N.A.M., BinMohamad,M.S., Mohamad A., Izham, M and Saad, M. A.M.(2021). Flow characterisation in an exhaust manifold of a single cylinder internal combustion engine (ICE). ***Journal of Physics: Conference Series*****, 2129 (2021):1-7**

Bajpai, k., Chandrakar, A., Agrawal, A and Shekhar, S. (2017). CFD analysis of exhaust manifold of SI engine and comparison of back pressure using alternative fuels. ***IOSR Journal of Mechanical and Civil Engineering*****, 14(1): 23-29**

Bodin, O. (2013). Simulations of compressible flows associated with internal combustion engines. Technical Reports from Royal Institute of Technology KTH Mechanics SE-100 44 Stockholm, Sweden. (Published): 1-100

Brito, C.H.G., Maia, C.B and Sodré, J. R. (2015). A Mathematical Model for the Exhaust Gas Temperature Profile of a Diesel Engine. ***Journal of Physics: Conference Series,*** **633 (2015):1-6**

Chatterjee, S. (2017). Computational fluid dynamic analysis of the exhaust gas flow through absorptive and reactive mufflers: some case studies Proceedings of the institution of mechanical engineers, Part D. ***Journal of Automobile Engineering,*** **(231)11 : 1568-1588**

Durat, M., Parlak, Z., Kapsiz, M., Parlak, A and Ficici, F. (2013). CFD and experimental analysis on thermal performance of exhaust system of a spark ignition engine. *Journal of Thermal Sciences and Technology*, 33(2):89-99.

Fajardo, D. P.(2012). Methodology for the NumericalCharacterization of a Radial Turbine under Steady and Pulsating Flow". PhD Thesis. Universitat Politecnica De Valencia

Fuc, P., Lijewski, P., Sokolnicka, B., Siedlecki, M and Szymlet, N.(2019).Analysis of Exhaust Gas Flow Through a Particulate Filter in the Exhaust of the Spark Ignition Direct Injection Engine. AIP Conference Proceedings 2078, 020088 (2019).

Jang, J., Woo, Y., Jung, Y., Cho, C., Kim, G., Pyo, Y., Han, M and Lee, S. (2018). Research for intake and exhaust system parameterization of 2-cylinder gasoline engine for RE-EV. *International Journal of Energy Research,* 42(13): 4256-4256.

Jianmin X and Shuiting, Z. (2014). The Analysis for flow field of automotive exhaust system based on computational fluid dynamics. *The Open Mechanical Engineering Journal,* 20(8): 587-593.

Kong, K. J. (2021). A 1D–3D Approach for fast numerical analysis of the flow characteristics of a diesel engine exhaust gas. *Machines,* 9(10):1-14.

Lungu, J., Siwale, L., Kashinga, R., Chama, S. and Bereczky, A. (2021). Correlation of performance, exhaust gas temperature and speed of a spark ignition engine using Kiva4. *Journal of Power and Energy Engineering,* 9, 53-78.

Mantilla, J, M., Falla, C, A and Gomez, J. A.(2009). Internal combustion engine exhaust pipe flow simulation: Part I: theoretical aspects. Ing. Investig. 29(1):115-125.

Mshelia, U. B. (2015). Simulation of fluid flow in the exhaust of an internal combustion engine. *International Journal of Engineering Research and Technology*, 8(1): 13-21

Mohamad B., Szepesi, G and Bolló B. (2017.Review article: Modelling and analysis of a gasoline engine exhaust gas systems. International Scientific Conference on Advances in Mechanical Engineering, 12-14 October 2017, Debrecen, Hungary

Munjal. M. L. (2006). Acoustics of ducts and mufflers with application to exhaust and ventilation system design. Wiley-Interscience,

Peat, K.S., Torregrosa, A.J., Broatch, A and Fernandez. T. (2006). An investigation into the passive acoustic effect of the turbine in an automotiveturbocharger. In: *Journal of Sound and Vibration*, 295(1-2):60-75

Vamsi, M and Chuen-Sen, L. (2019) Numerical study of an exhaust heat recovery system using corrugated tube heat exchanger with modified twisted tape inserts. *International Journal of Energy Engineering* ,9(1): 12-24.

Venkata S. M. (2004). Simulation of fluid flow in internal combustion engines using wave action simulation. B.Tech. Thesis. Graduate Faculty of Texas Tech University.

Xu, P., Jiang, H and Zhao, X. (2016). CFD analysis of a gasoline engine exhaust pipe. *International Journal of Simulation: Systems, Science & Technology*,17(20): 7.1-7.7

Zubair, A.M. (2022). Development of a prototype engine for generation of power from exhaust gas of gasoline generator. Seminar paper presented to Mechanical Engineering Department, Federal University of Technology, Minna, Nigeria.