GSJ: Volume 8, Issue 3, March 2020, Online: ISSN 2320-9186
www.globalscientificjournal.com

Design Optimization of the Combustion Chamber of a Diesel-Plant for Power Generation

Ajoko, Tolumoye John[1]; Biebelemo. Jethrow. Jonathan[2], UmorenGoodnews Godwin[3]

1. **PhD Research Scholar**, Department of Mechanical Engineering, Faculty of Engineering, Niger Delta University, Wilberforce Island, Bayelsa State, Nigeria. Email: johntolumye@yahoo.co.uk, or tjajoko@ndu.edu.ng
2. **PhD Research Scholar**Department of Agricultural/Environmental Engineering, Faculty of Engineering, Niger Delta University, Wilberforce Bayelsa State, Nigeria. Email: jetbie4real@gmail.com
3. **Research Student**, Department of Mechanical Engineering, Faculty of Engineering, Niger Delta University, Wilberforce Island, Bayelsa State, Nigeria. Email:goodnewsgodwin@gmail.com

KeyWords
Biodiesel, Combustion, gravimetric, Optimization, stoichiometric, volumetric

ABSTRACT

Diversification of power generation is an important factor to energize a growing economy of any nation in the world. Thus, diesel engines are one of the most common reciprocating engines for power generation applications. Although, the chemical combustion of diesel and air in the combustion chamber (CC) of the engine due to its long carbon chain produces unburnt gas and air resulting to the production of heavy carbon(IV)oxide (CO_2). This concern is the reason of optimizing the CC of diesel engine perhaps with biodiesel as fuel for power generation. The study was carried out in computational fluid dynamics (CFD) interface, while analysis was numerically analyzed with the aid of stoichiometric combustion equations. Results attained attests that excess air addition in the combustion process reduces the production CO_2 both gravimetrically and volumetrically from (23.8 – 16.61)%, (18.56 – 12.25)% and (15.56 – 10.91)% respectively. Other results are increase in the air production at the end of the combustion exercise from (0 – 7.05)% and (66.45 - 69.55)% for oxygen gas (O_2) and nitrogen (N_2) gas respectively. This affirms that biodiesel is a clear substitute of diesel for power generation in diesel engine after optimization of the CC.

I. INTRODUCTION

The search for cleaner energy for power generation is the concern for scientific and engineering research. Thus, the presentation of this paper is focused to bridge the gap and discontinue the use of fossil fuels in power generating plants like diesel engines due to its exhaust emissions. This is imperative because the over dependence on the use of fossil fuels in machineries has caused more danger to the environment than been safe which cannot be over emphasized. Research over the years confirms that the quality of air circulating in the environment has deteriorated as a result of exhaust gas emissions from fossil fuelled engines. This has increased respiratory diseases significantly in the environment. Also the toxicity of these gases are notably the cause of global warming, ozone layer depletion, etc, a threat to global catastrophe seeking for amends [1. 2]. However, the set of solution in restructuring and redesigning of a diesel plant for power generation is the consideration of optimizing the CC to suite a biofuel in carrying out the same purpose.

Meanwhile, optimization requires the definition of control parameters determining the search space over which the studied configuration has to be improved. In the simple mechanical power generating context, the set of design parameters are very large and cannot be used as a whole. For simplicity, only geometrical and inflow conditions are chosen as possible optimization criteria. This means that a given CC is improved acting on a limited set of parameters and not totally designed from scratch. The user defines cost functions on the search space to assess the quality of a given design in that space. All the state variables and the functions are evaluated from CFD. In the last decade researchers have been concerned with optimizing internal combustion engine (ICE) performances using CFD tool. Recent experimental presentation by [3] offered analysis on turbulence modelling of ICEs using proper orthogonal decomposition (POD) method to both CFD and particle imaging velocimetry data of simplified motored engine flows. [4] adopted the use of CFD models to determine overall engine performance using design and operational variables of inlet and exhaust systems. This models were basically adopting zero dimensional phenomena within the engine cylinder and one dimensional phenomena at the engine inlet and exhaust considering adjusted valve timing, valve diameter and diameter of exhaust manifold. A similar contribution by a scholar made a progressive modelling of a four stroke engine with the use of open field operation and manipulation (Open-FOAM) software which is also an industrially adopted software to simulate ICEs [5]. This model was able to maintain a platform for comparison in the areas of pressure distribution, turbulent kinetic energy as well as heat transfer in the combustion.

However, another interest of concern is the optimization methods used on genetic algorithm (GA) in the automotive industry over the last few years due to the wide range of solutions they offer in combination with CFD. Meanwhile, a similar study was carried out by group of researchers in combining GA and artificial neural network (ANN) for optimizing the intake port design of a spark-ignited (SI) engine with four control parameters [6]. Conversely, several studies have applied this technique to vary engine applications in which the number of optimized parameters are relatively high [7]. In view of this, the optimization of CC design of a compression-ignition (CI) of diesel engine with six design parameters considering emissions and performance was carried out [8]. Nevertheless, a simulation study conducted by [9] reveals that piston bowl geometry affects the combustion and performance characteristics of direct-injection (DI) and CI engines. Furthermore, these studies have been used to optimize the piston bowl geometry and spray angle in order to improve the performance and reduce the production of emissions. Piston bowl geometry plays an important role in the motion of air and fuel inside the cylinder. A high swirl ratio developed from piston geometry may produce better air–fuel mixture [10]. This will control the turbulence in the CC hence reducing the production of NOx, soot, hydrocarbons and carbon-monoxide. However, simulating the CC of diesel engine with CFD tool to enable the use of biofuels in the same set of engines will reduce emissions production and thus increase performance. Also simulation will take care of combustion noise control which is a criterion in engine design optimization studies.

II. CFD OPTIMIZATION

The combustion system of a Perkins 403D-15G diesel engine model shown in figure 1 is analyzed with the aid of CFD Ansys Fluent. The study is aimed to carryout CFD optimization test to enhance improvement in the engine CC for the conversion of the system to the use of biofuels. The model is a compact of 3 cylinders, 1.1 litres diesel motor that runs at1800 rpm**.** It works at full or part-time for days or weeks at a time which needs only a change of oil at every 500 hours of use. The diesel engine uses diesel as its fuel in general and it is more efficient than gasoline engines. The Perkins 403D-15G diesel engine model specification is presented in table 1.

Figure 1: Perkins 403D-15G Diesel Generator
Sources: [11]

Table 1: Perkins 403D-15G Diesel Engine Specifications

Model	403D-15G
Parameters	Specifications
Mechanical Output	12-16 kWm
Electrical Output	13-17 kVA (10-14 kWe)
Rated Speed	1800 rpm
Bore	84.0 mm
Stroke	90.0mm
Displacement	1.5 Litres
Aspiration	Natural Aspiration
Coolant Capacity	6.0 litres
Oil Capacity	6.0 litres
Rotation from Flywheel End	Anti-clockwise
Numberof Cylinders	3
Compression Ratio	22.5:1
Combustion System	Indirect injection
Cycle	4 stroke
Cooling System	Liquid (Water Cooled)
Width	497.0 mm
Length	820.0 mm
Height	791.0 mm
Dry Weight	197.0kg

Meanwhile, the thermodynamic variables involved in the optimization performance of ICEs can be streamlined into a particular section of improvement. However, these parameters captured in this study include the inlet velocity, engine temperature, density, pressure, turbulent viscosity, etc. Thus, establishing the relationship between the objective functions and control variables is paramount to the parameterization. Therefore, to achieve this aim the important basic steps in quest of the CFD analysis include the following.

III. PRE-PROCESSING, COMPUTATION AND POST-PROCESSING

The geometrical modelling of the domain for which the CFD is to be conducted is carried out in Solidworks CAD modelling tool with suitable scale and imported to Ansys Fluent domain for analysis. The pre-processing also involves mesh generation by using suitable finite element models. Finite volume models are also used for three dimensional modelling. However, accuracy of results can depend on meshing technique adopted for simulation. Boundary conditions are important in CFD runs where the condition variables are computed and boundary characteristics are defined for the geometry. Off course, the computation is the solver operations performed by the computer. Iterative methods are used while the system calculates and computes the solution. Thus, the parameters of computation in study include stability, convergence and consistency. The post-processing **e**xperiments are performed to achieve results. Therefore, post-processing is the extraction of results from CFD runs but results are presented in visual modes and colour legend for analysis. In understanding the principle of operation for conducting optimization regarding the CC it requires the knowledge of turbulent flow characteristics and parameters. Hence, equations have been developed to assign variables in the CFD experimentation as given in equations 1 – 7. They are the continuity, momentum and enthalpy Equations, others are temperature, mass fraction, turbulent kinetic energy and turbulent dissipation rate equations respectively.

$$\frac{\delta \rho U}{\delta t} + \nabla \cdot \rho U = 0 \quad \text{......} 1$$

$$\frac{\delta \rho U}{\delta t} + \nabla \cdot \rho UU = -\nabla p + \nabla \tau + \rho g \quad \text{......} 2$$

$$\frac{\delta \rho h}{\delta t} + \nabla \cdot \rho U = \nabla \cdot \lambda_e \nabla T - \nabla q_r + \nabla \cdot \sum_l p h_l (T) D_e \nabla m_l \quad \text{......} 3$$

$$\rho C_p \frac{dT}{dt} = \nabla \cdot \lambda_e \nabla T - V \cdot \sum_l \rho h_l(T) D_e \nabla m_l - \rho \sum_l \frac{dm_l}{dt} h_l(T) \quad \ldots 4$$

$$\frac{\delta \rho m_l}{\delta t} + \nabla \cdot \rho U m_l = \nabla \cdot D_e \rho \nabla m_l - R_l \quad \ldots 5$$

$$\frac{\delta(\rho k)}{\delta t} + \frac{\delta(\rho k u_i)}{\delta x_i} = \frac{\delta}{\delta x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\delta k}{\delta x_j}\right] + P_k + P_b + \rho \in - Y_M + S_k \quad \ldots 6$$

$$\frac{\delta(\rho \in)}{\delta t} + \frac{\delta(\rho \in u_i)}{\delta x_i} = \frac{\delta}{\delta x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\in}\right)\frac{\delta \in}{\delta x_j}\right] + C_{1\in}\frac{\in}{k}(P_k + C_{3\in} P_b) - \rho C_{2\in}\frac{\in^2}{k} + S_\in \quad \ldots 7$$

IV PRESENTATION OF RESULTS AND DISCUSSION

CFD employs virtual laboratory to simulate experiments to determine approximate solutions. This is done in the early stage of the design to test the methodology without actually having to build a physical model or gadget. The operation of the CFD software simulation employ mathematical models that characterizes different phenomena occurring in the system, and boundary conditions to forecast experimental results most convergent to actual physically conducted experiment results. The accuracy of the CFD simulation is dependent on the appropriate selection of the model as well as the specification of its boundary and loading conditions. The sequence of simulation carried out is the geometry creation, discretization (meshing), model selection and specification of boundary and loading condition in setup, solution initialization, solving (Solution), and then reading of results. Some of the process results are presented in figures 2 – 5 while table 2 is geometric parameters used for the geometry creation.

Tables 2: Geometric Parameters

Description	**Value**
Bore	84.00 mm
Stroke	90.00 mm
Compression ratio, C_r	22.5:1
Inlet size	33.40 mm
Exhaust Outlet Size	38.10 mm
Clearance Height, C_h	4.19 mm

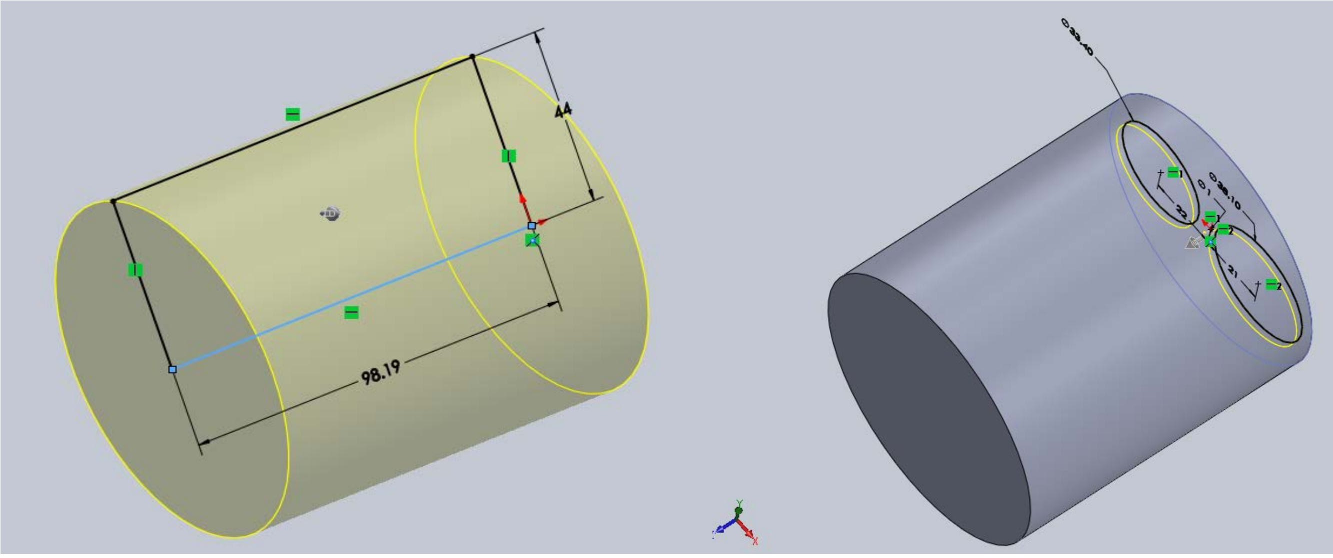


Figure 2: Geometry modelling

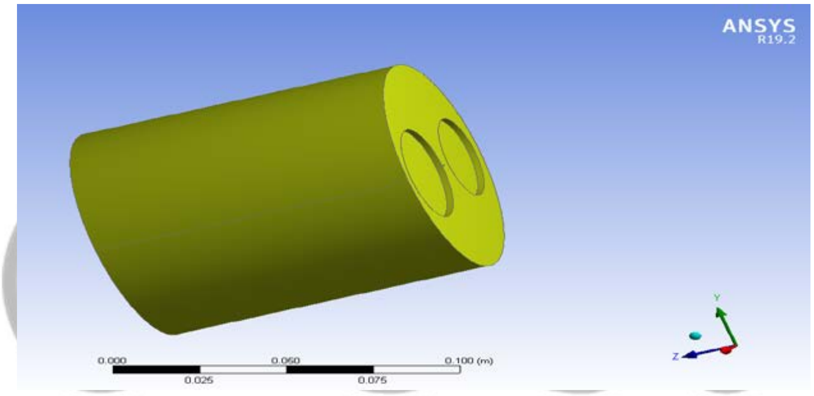


Figure 3: Finite Element Model – Fluid

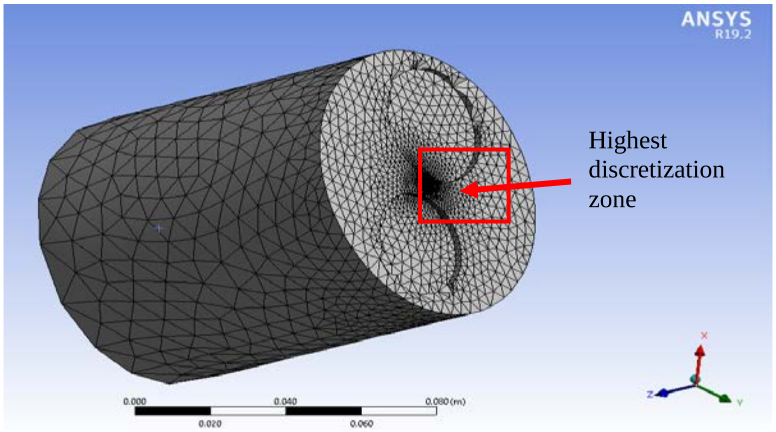


Figure 4: Discretized Model of the Combustion Chamber

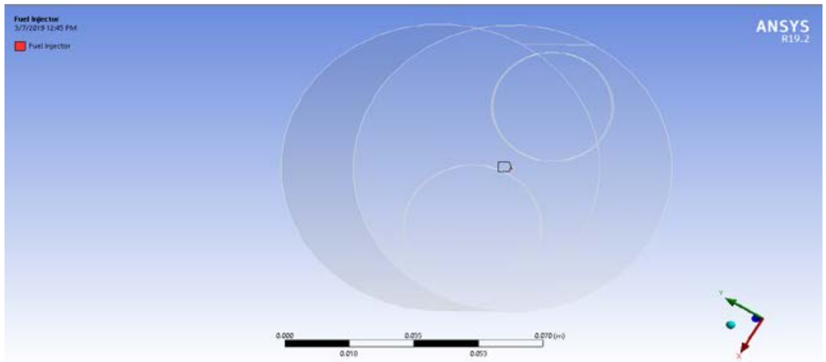
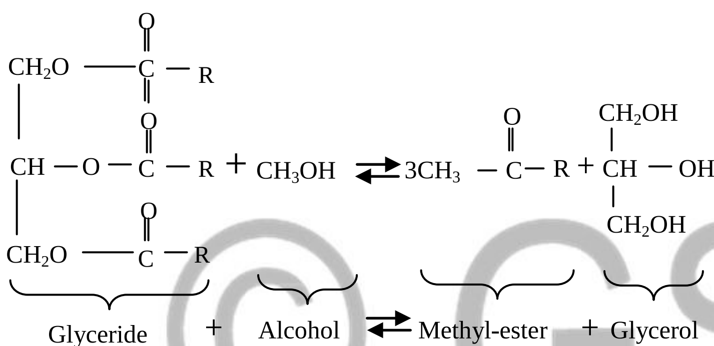
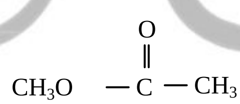


Figure 5: Selection of Fuel Injector

Meanwhile, position for the fuel injector is specified on the CC and the biodiesel fuel used is analyzed. The chemical formation of the biofuel (methyl-ester) is a product of glyceride and alcohol as shown in the reversible and structural equations as presented in equations 8 and 9 respectively.



……………………………………….8



……………………………………………………….9

where R is a methyl radical.

Meanwhile, the condensed formula of the methyl-ester (methyl-acetate) can be written as:- CH_3COOCH_3 or $C_3H_6O_2$ which is used for the stoichiometric chemical combustion equation as presented in equation 10.

$$2\mathrm{CH_3COOCH_3} + 7O_2 \rightarrow 6\mathrm{CO_2} + 6\mathrm{H_2O} \quad \ldots\ldots 10$$

$$\underbrace{2\mathrm{CH_3COOCH_3}}_{\text{Fuel}} + \underbrace{7O_2 + 7\left(\frac{79}{21}\right)N_2}_{\text{Air}} \rightarrow 6\mathrm{CO_2} + 6\mathrm{H_2O} + 7\left(\frac{79}{21}\right)N_2$$

The mole and mass ratio of fuel and air reaction in the equation above for 100% theoretical air is given as:-

$$2\text{mol fuel} + 33.33mol\ Air \rightarrow 6mol\ CO_2 + 6mol\ H_2O + 26.33molN_2$$

$$148\text{Kgfuel} + 961.33KgAir \rightarrow 264KgCO_2 + 108KgH_2O + 737.33KgN_2$$

$$1\text{Kgfuel} + 6.496KgAir \rightarrow 1.784KgCO_2 + 0.73KgH_2O + 4.982KgN_2$$

This yields 7.496kg of the reactants giving the same value of the products, confirming the law of conservation of matter. Subsequent results from the gravimetric and volumetric analysis of the combustion process of this study is presented in figures 6 – 8 graphically.

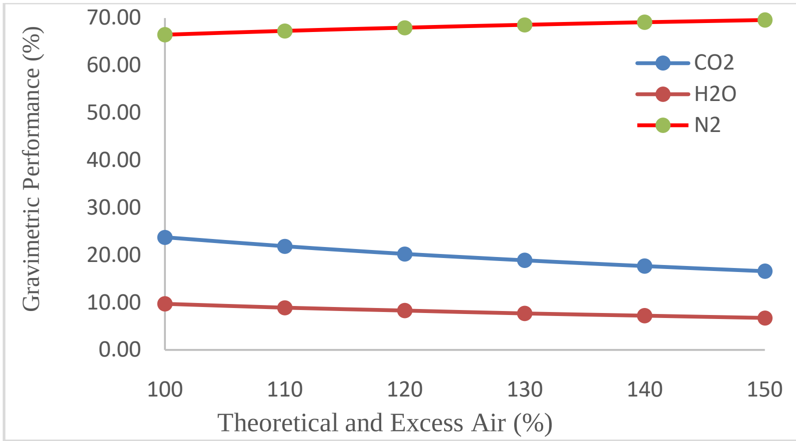


Figure 6: Gravimetric Analysis Result

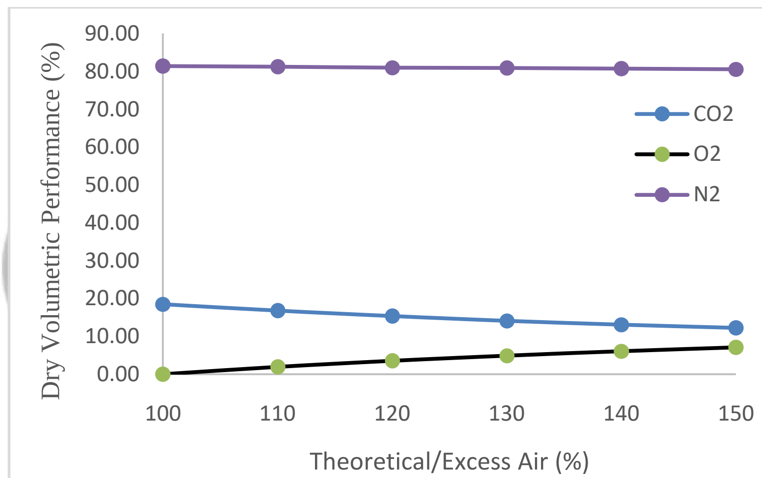


Figure 7: Dry Volumetric Result Analysis

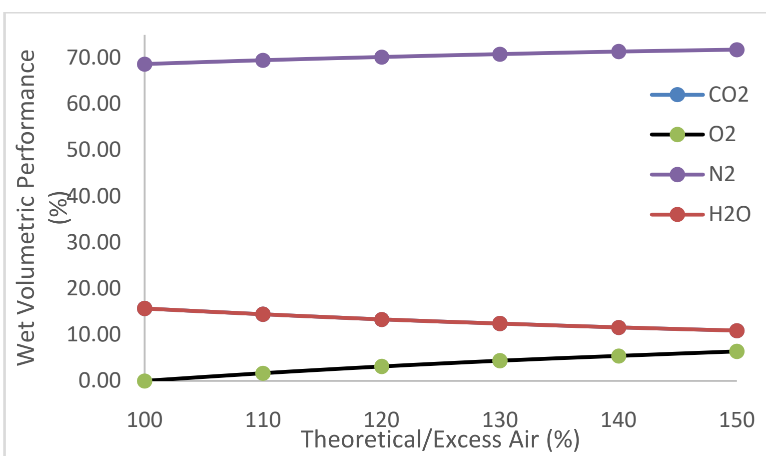


Figure 8: Wet Volumetric Result Analysis

The graphical results are presentation of the products of the stoichiometric combustion equations which by all indication are complete combustion process. It is obvious that complete combustion process always gives off CO_2 and H_2O as products of the reaction of fuel and air. Thus, the results from this study confirms this fact. However, the results shows reduction of CO_2 production from (23.8 – 16.61)%, (18.56 – 12.25)% and (15.56 – 10.91)% in figures 6, 7 and 8 respectively as excess air is supplied to the combustion process. Meanwhile, the analytical values of CO_2 and H_2O for figure 8 are almost the same hence, the plots in the graph appears as one entity. Similarly, the exhaust water from the engine at the end of the combustion also decreases in percentage from (9.74 – 6.79)% and (15.56 – 10.91)% in terms of gravimetric and wet volumetric analyses. Contrarily, the formation of air with its two key components (oxygen and nitrogen) at the end of the combustion process improves in their percentage performance. The combustion of biodiesel with theoretical air of (100 – 150)% gives an increment of O_2 and N_2 from (0 – 7.05)% and (66.45 - 69.55)% gravimetrically and it is shown in figure 6. Likewise, these components of air in the wet volumetric analysis as shown in figure 8 increases from (0 – 6.36)% and (68.70 − 71.82)% for O_2 and N_2 respectively. These results attests that the use of biodiesel in replacement of fossil fuel will not in any form cause degradation and poisonous emission to the environment.

IV CONCLUSION

The study of optimizing the CC of a diesel engine for power generation in order to produce little or no emission can be concluded with the following conclusive points.

- Analysis from the study affirms that biodiesel can serve as alternative fuel in diesel power generating plant after the optimization of the CC.
- The chemical combustion of biodiesel such as methyl-acetate with air in the CC of diesel engine produces harmless products to the environment.
- The process of generating biodiesel is safe, more economical compared to its rival, environmentally friendly and readily available everywhere since its raw materials is mostly agricultural crops.

Therefore, carrying out design optimization for the CC of a diesel engine to generate electric power is one means of putting off fossil fuelled plants. Hence, the continuous embracing of the alternative sources and keeping the environment free from any form of emissive pollutant(s) should be the target of above research.

REFERENCES

[1] M. Kampa and E. Castanas, "Human Health Effects of Air Pollution: 4th International Workshop on Bio-monitoring of Atmospheric Pollution".*Environmental Pollution–Elsevier*, Vol.151, No.2, pp. 362-367, 2008.

[2] W.L. Hardy and R.D Reitz, "A Study of the Effects of High EGR, High Equivalence Ratio, and Mixing Time on Emissions Levels in a Heavy-Duty Diesel Engine for PCCI Combustion" *SAE Technical Paper*, 2006-01-0026, 2006.

[3] E.A.M Fogleman, "Application of the proper orthogonal decomposition to datasets of internal combustion engine flows". *Journal of Turbulence* Vol.5, pp.1-23, 2004.

[4] K. Ugur, "Study on the design of inlet and exhaust system of a stationary internal combustion engine". *Elsevier-Energy Conversion and Management*, Vol. 46, No.13-14, pp.2258–2287, 2005.

[5] G. Stefan, "CFD modeling of a four stroke S. I. engine for motorcycle application". Unpublished MSc. Thesis, KTH Industrial Engineering and Management, Stockholm, Sweden, 2009

[6] Y. Sun, T. Wang, Z. Lu, and L. Cui, "The Optimization of Intake Port using Genetic Algorithm and Artificial Neural Network for Gasoline Engines". *SAE Technical Paper* 2015-01-1353, 2015.

[7] S.W. Park, "Optimization of Combustion Chamber Geometry for Stoichiometric Diesel Combustion Using a Micro Genetic Algorithm". *Elsevier-Fuel Processing Technology*, Vol.91, No.11, pp. 1742-1752, 2010.

[8] P. Senecal and R. Reitz, "Simultaneous Reduction of Engine Emissions and Fuel Consumption Using Genetic Algorithms and Multi-Dimensional Spray and Combustion Modeling". *SAE Technical Paper* 2000-01-1890, 2000.

[9] J. Lim and K. Min, "The effects of spray angle and piston bowl shape on diesel engine soot emissions using 3-D CFD simulation". *SAE Technical Paper* 2005-01-2117, 2005.

[10] J. Song, C. Yao, Y. Liu, and Z. Jiang, "Investigation on flow field in simplified piston bowls for DI diesel engine". *Eng. Appl. Comp. Fluid* 2(3), 354–365, 2008.

[11] Perkins, "Perkins 403D-15G Diesel Engine" [Online] Available: https://www.perkins.com › Perkins › Electric Power Generation › Diesel. Accessed 27th December, 2019