International Journal of Engineering Trends and Technology
ISSN: 2231–5381 / https://doi.org/10.14445/22315381/IJETT-V71I10P215

Volume 71 Issue 10, 163-172, October 2023


*Original Article*

# CFD Study of the Effect of Engine Speed on the Combustion Process and the Formation of Pollutants in a Diesel Engine

Fabrice Parfait Nang Nkol[1,2*], Nelson Junior Issondj Banta[1,2,] Claude Valery Ngayihi Abbe[1,2], Ruben Mouangue[1,2]

[1]*National Higher Polytechnic School of Douala, University of Douala, Cameroon.*
[2]*Laboratory of Energy, Materials, Modeling and Method of the University of Douala, Cameroon*.

**Corresponding Author : parfaitnang@yahoo.com*

Received: 23 July 2023 Revised: 12 September 2023 Accepted: 13 September 2023 Published: 03 October 2023

***Abstract*** *- This article presents a numerical study on the influence of engine speed on the combustion process and the formation of pollutant emissions in a compression ignition engine. The literature shows that the movement of the air impacts the combustion process and, therefore, pollutants. Experimental studies have shown that at higher engine speeds, an increase in turbulence is created in the cylinders, thus improving the air/fuel mixture. However, few studies address the fundamental factors of varied engine speed at reduced steps for a certain range. This study develops and validates a CFD model with experimental data to predict the combustion scenario. Zeldovich extended mechanism, Hiroyasu model, and Kelvin-Helmohtz model are adopted to calculate NOx, soot, and spray quality, respectively. The engine speed was varied between 1500 and 2000 RPM with an increment of 100 RPM to better analyze and optimize combustion parameters and pollutant variations. The results of this research show that at higher engine speeds, fuel consumption is reduced in a shorter time by improving the air/fuel mix. The combustion time is shorter, which means less time for NOx and soot emissions. The improved air/fuel mix significantly reduces NOx and soot by about 38% and 40%, respectively. The results of these calculations show that the combustion process and the formation of pollutants strongly depend on the engine speed and load as a function of the crankshaft angle under ignition conditions. The results also show that engine speed and injection timing can be adopted in order to establish optimal engine power conditions based on low pollutant emissions and correct engine and exhaust temperatures in a compression ignition engine. This study blocks the benefits of using computational fluid dynamics (CFD) to better predict a diesel engine's combustion process and pollutant emissions.*

***Keywords*** *- CFD, Engine speed, Combustion, Pollutants, Diesel engine.*

## 1. Introduction

The motorization sector is one of the main contributors to energy consumption and polluting environmental emissions. These emissions from internal combustion engines have a negative impact on the environment and human health. Among the polluting emissions that derive from the combustion of engines, we have soot and NOx particles, which are the result of the unfinished combustion of engines. They are considered the main climate change factors, right after greenhouse gases. Essentially composed of black carbon, soot particles absorb solar radiation, thus contributing to nature's warming and rapid degradation. Soot particles and NOx seriously harm human health; once absorbed, they can cause respiratory and cardiovascular diseases [1]. This is how the very strict measures were launched for their reduction, both for their mass and the number of particles [2–3]. Although many studies have been carried out on the understanding and reduction of soot particles, NOx, and many other pollutants from internal combustion engines, these phenomena remain dominant in Africa because their formation involves a complex physical and chemical process in the environment. To better understand the phenomenon of pollution that derives from internal combustion engines, research has been carried out in several automobile sectors following well-defined strategies to optimize these performances and revise downward its pollutant rate, one of which is to act on the engine speed [4].

The following experimental studies [5–9] have shown that at high engine speeds, the engine experiences an increase in the phenomenon of turbulence in the combustion chamber, which implies an improvement of the mixture air–fuel, which means that engine speed and injection timing can be applied wisely to determine the optimal conditions for a good engine power range with low emissions and acceptable engine and exhaust temperatures in compression ignition engines. Tests performed at different engine speeds on a single-cycle version of the Caterpillar 3406 production engine by Daniel A. Nehmer et al. [10] show that flow and fractional injection can affect soot and NOx emissions from a heavy diesel engine. Another recent study [11] shows that low-temperature combustion in compression-ignition engines has the ability to produce ultra-low NOx and soot emissions while maintaining good thermal efficiency. This study shows that to obtain a low-temperature combustion, a proportionally defined mixing time between air and fuel is necessary to avoid fuel-rich regions and reduce the maximum combustion temperatures, considerably reducing the formation of polluting emissions. In order to better predict the combustion process and the phenomenon of pollution in diesel engines, a numerical study based on fluid dynamics (CFD) is appropriate because it is an alternative to the experimental study. Although the execution times of the scenarios are relatively longer than the 0D models, this allows a better approach to the combustion process and pollutants [12].

Since the 1990s, several articles have been published based on the application of CFD to internal combustion engines to better understand the phenomena in the cylinders and propose good engine optimization [13–15]. Nowadays, several research topics based on the dynamics of digital fluids are conducted to optimize the engines [16–18]. Christian Angelberger et al. [19] show in their work that digital fluid mechanics is a key method to further improve internal combustion engines in terms of performance and environmental preservation. Computational fluid dynamics has the unique potential to enable the consolidation of research knowledge in the fields of turbulence, chemistry, combustion, thermodynamics, and heat transfer and then integrate it into different phases of engine design processes to understand, control, and better optimize combustion in engines [20–22]. In this study, a CFD model is developed and experimentally validated [23] for the work done on a diesel engine at Sandia National Laboratories.

The above-mentioned research is based on a variation of the engine speed with a pitch difference of about 500 RPM; this certainly allows for an understanding of the effect of this strategy, but reducing this step of the engine speed could lead to a better understanding and optimization of the characteristics of the engine. The research of [24] explains that the best range for a good study of combustion phenomena and polluting emissions of a compression ignition engine is between 1500 RPM and 2000 RPM. The objective of this study is to analyze through the CFD calculation code the phenomenology of diesel combustion and pollutant emissions under engine speed conditions variants between 1500 RPM and 2000 RPM with a pitch difference of the engine speed of more than 100 RPM.

## 2. Materials and Methods

### *2.1. Materials*

Good combustion improves the efficiency and optimization of the diesel engine. In medium-weight engines, the interactions between the fuel spray and the piston tank walls play a fundamental role in defining the heat release rate. Stepped lip pistons promote large-scale turbulence phenomena resulting from faster and more efficient heat release. However, it should be noted that this behavior is widely observed for late injection settings where the engine does not operate at its maximum efficiency and, therefore, at low rpm [25–27]. It is in this sense that a new medium-weight diesel search engine was built at the Sandia National Laboratory to allow advanced research on the combustion of pollutants and methods of heat loss through walls to improve efficiency. Our research topic was validated experimentally based on the previous study's data.

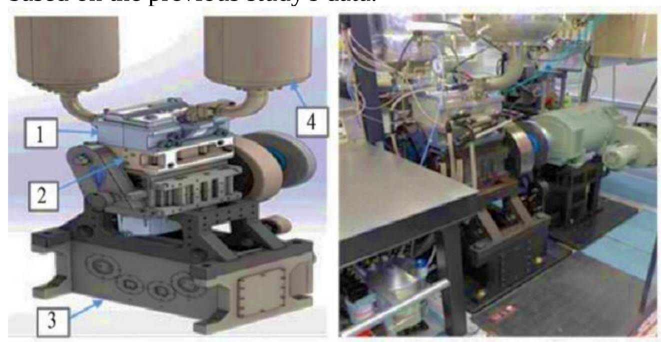



**Fig. 1 Experimentation with light optical diesel engine [23]**

1. Cast aluminum cylinder head.
2. Custom deck adapter facilitates conversion to optimal engine.
3. Lanchester balancing box.
4. Control of intake flow rate, composition, and temperature.

**Table 1. Engine parameters**

| | |
|---|---|
| Boron × Stoke | 99 × 108 mm |
| Displacement engine | 0.477L |
| Compression ratio | 16.2 |
| Nozzle diameter | 0.254 mm |
| Fuel Decane | C10H22 |
| Fuel injected per orifice | 29.58 mg/cycle |
| Injection pressure | 800 bars |
| Injection start timing | $691^0$CA |
| Injection duration | $20^0$CA |
| Spray direction | 700 with the cylinder axis |
| Coordinates of spray emanation point | x=0, y=0, z=2e-5m |
| Engine speed | 1500 rpm |
| Number of nozzles | 7 |
| Intake valve closed (IVC) | $570^0$ CA |
| Exhaust valve opened (EVO) | $833^0$ CA |
| Swirl number at IVC | 1.3 |

### *2.2. Methodology*

The mesh in ANSYS Forte is generated automatically and does not require enough input or processing. After having defined surfaces and their refinement methods, the only important input is the overall size of the mesh. The resolution of the mesh and the detail of the results depend strongly on the size of the mesh. Accurate results require a finer mesh, although computation times and power have greatly increased. This is a very important factor because time and energy resources are very limited.

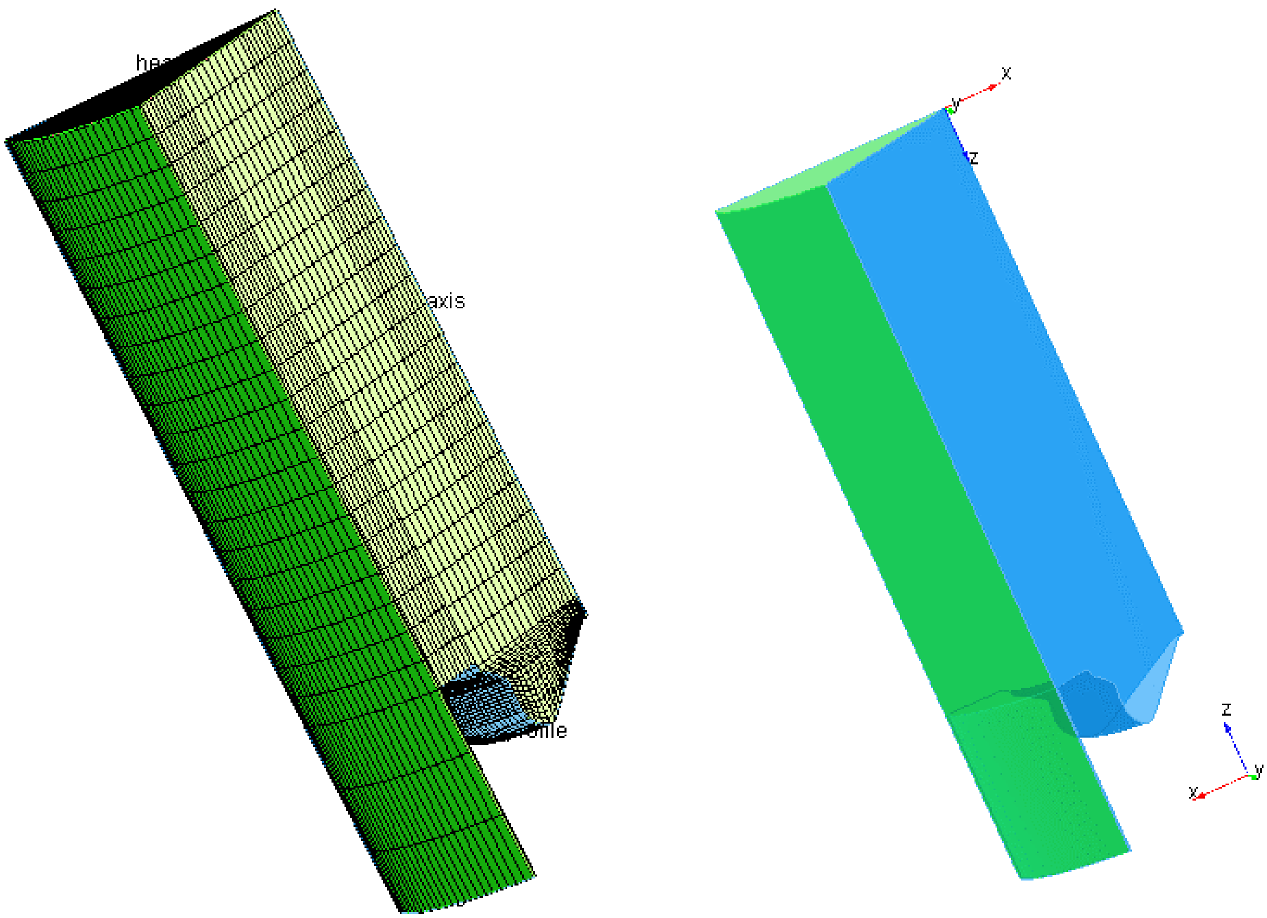



**Fig. 2 Cylinder sector geometry and mesh**

#### *2.2.1. Boundary and Initial Condition*

ANSYS Forte uses the finite volume method to solve the governing algebraic equations. The boundary conditions, initial conditions, and set-up settings are listed in the table-2.

#### *2.2.2. Model CFD*

Numerical Fluid Dynamics is a means of applying numerical methods to analyze equations governing heat transfer and fluid dynamics to obtain solutions to fluid flow problems [28]. For this, computers need discretized algebraic equations derived from continuous differential equations to obtain solutions. In the ANSYS Forte package solvers, there are governing equations that govern the calculation process. These equations are, more precisely, conservation equations that derive from the driving equations (1), the Navier-Stokes equation for the conservation of momentum (2), the space conservation equations (3), and energy conservation (4) [26].

The above equations are necessary for flow simulations and modelling. As for simulations, they consist of Direct Numerical Simulation (DNS) and large turbulence simulation (LES). DNS and LES have specific applications with relatively higher calculation costs.

**Table 2. Boundary and initial conditions**

| **Turbulence model** | **SST-k Epsilon** |
|---|---|
| Inlet boundary conditions | Inlet pressure 1bar |
| Outlet boundary conditions | Outlet pressure 1.23bar |
| Time step | $10^{-5}$ sec |
| Initial gas temperature | 372.12 K |
| Initial swirl ratio | 1.5 |
| Initial swirl profile factor | 3.11 |
| Sector angle | 51.42 deg |
| Head temperature | 470K |
| Liner temperature | 420 K |

However, a simpler approach is adopted for numerical flow studies, the Reynolds Averaged Navier-Stokes (RANS) method. The RANS method is recognized for its seniority in terms of CFD analysis. It is very simple, requires little time for calculations, and is very effective for the analysis of turbulent flows. ANSYS Forte, which is the solver used for this work, integrates the RANS approach and is capable of simulating the average flow field. This approach uses the RANS equations themselves (5), (6), and the Reynolds stress equation (7) presented below [29–30].

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = 0 \tag{1}$$

$$\partial\left(\frac{\partial \vec{u}}{\partial t} + \vec{u}.\nabla\vec{u}\right) = -\nabla P + v\Delta u \tag{2}$$

$$\frac{\partial}{\partial t}\int_V dV = \int_S \vec{V}_b dS \tag{3}$$

$$\frac{DV}{Dt} = -\nabla q - p(\nabla.V) + \rho\dot{Q} \tag{4}$$

$$\frac{D\bar{u}}{Dt} + \frac{1}{\rho}\nabla\bar{p} - (\nabla.\nu\nabla)\bar{u} = \nabla.r \tag{5}$$

$$\frac{\partial \rho u_i}{\partial x_i} = 0 \tag{6}$$

$$\tau_{i,j} = \rho(u'_i u'_j) = \rho(u_i u_j) - \rho(u_i u_j) \tag{7}$$

*2.2.3. The NOx Formation Model*

The mechanism of NO formation has been studied by many researchers. Zeldovich et al., however, have shown in their work the particular role of certain reactions in the formation of NO. The concentration of NO is calculated decoupled from the combustion phenomenon by a post-treatment procedure based on the reversible reactions of the Zeldovich mechanism:

$$\frac{d[NO]}{dt} = \frac{2R\{1-([NO]/[NO]_e)^2\}}{1+(\frac{[NO]}{[NO]_e)R_1}/(R_2+R_3)} \tag{8}$$

Where the following notations have been Introduced, designating by $[NO]_e$ the concentrations. The concentration of NO in equation (7) can be converted into a mass fraction as:

$$\frac{dX_{NO}}{dt} = \frac{2\left(\frac{M_{NO}}{\rho c.v}\right)R_1\{1-([NO]/[NO]_e)^2\}}{1+(\frac{[NO]}{[NO]_e)R_1}/(R_1+R_3)} \tag{9}$$

*Soot Formation*

For soot, an empirical model is used, considering two concurrent reactions: soot formation and its oxidation. It is easier to implement with the CFD program as it provides empirical equations that need to be adjusted to match the experimental profile of soot. One of these models most commonly used in the literature was proposed in 1983 by Hiroyasu et al., which is directly applicable to the simulation of internal combustion engines. This model was implemented by Feiyang et al. [1] to develop a diesel engine and is confronted with other models. It follows the equations that calculate the rate of soot formation using the rate of soot formation and the rate of oxidation in the Arrhenius-type equations:

$$\frac{dm_{soot}}{dt} = (\frac{dm_{soot}}{dt})_{formation} - (\frac{dm_{soot}}{dt})_{oxydation} \tag{10}$$

$$(\frac{dm_{soot}}{dt})_{formation} = A_f m_{fuel} p^{0.5} exp(-\frac{E_f}{RT}) \tag{11}$$

$$(\frac{dm_{soot}}{dt})_{oxydation} = A_O m_{soot} X_{o_2} p^{1.8} exp(-\frac{E_o}{RT}) \tag{12}$$

Where

$m_{soot}$ is the mass of net soot formed, $m_{fuel}$ is the mass of fuel vaporized, $X_0$ is the molar fraction of oxygen, $E_f$ and $E_0$ are the activation energies of soot formation and oxidation, respectively, $A_f$ and $A_0$ are parameters that can be adjusted to match the simulation to the experiment.

*Injection Model*

The KH-RT model estimates that the disruption of the liquid is due to two types of instabilities: the first is the Kelvin-Helmholtz type. In his study, Reitz gets the wavelength $A_{KH}$ and the rate of increase $\Omega_{KH}$ of the fastest-growing wave. Based on the dimensionless numbers of the problem, Reitz obtains the following correlations from the results of [31]:

$$A_{KH} = \frac{9.02r_0(1+0.45\sqrt{Z})(1+0.4T_a^{0.7})}{(1+0.865We^{1.67})^{0.7}} \tag{13}$$

$$\Omega_{K.H.} = \frac{0.34+0.38We^{1.5}}{(1+Z)(1+1.4T^{0.6})}\sqrt{\frac{\sigma}{\rho_l r^3}} \tag{14}$$

*Model of Heat Transfer in the Engine Walls*

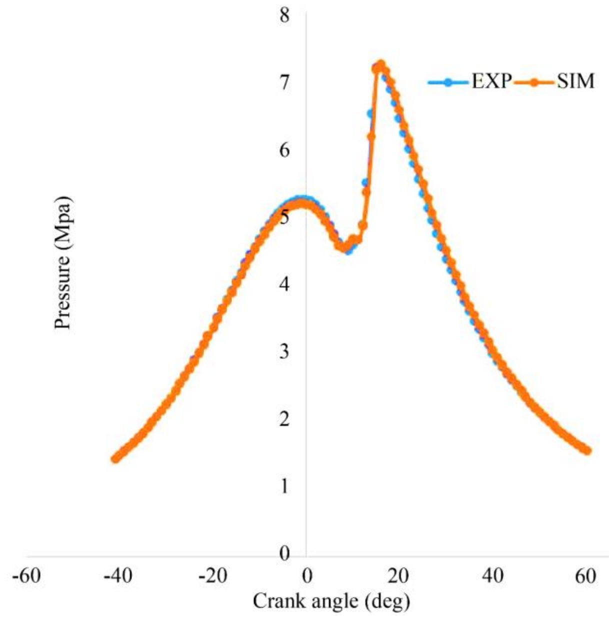
The multidimensional analysis of the heat transfer inside the engine is performed by solving the following heat diffusion equation:

$$\nabla(k\nabla T) = \rho c_p \frac{\partial T}{\partial t} \tag{15}$$

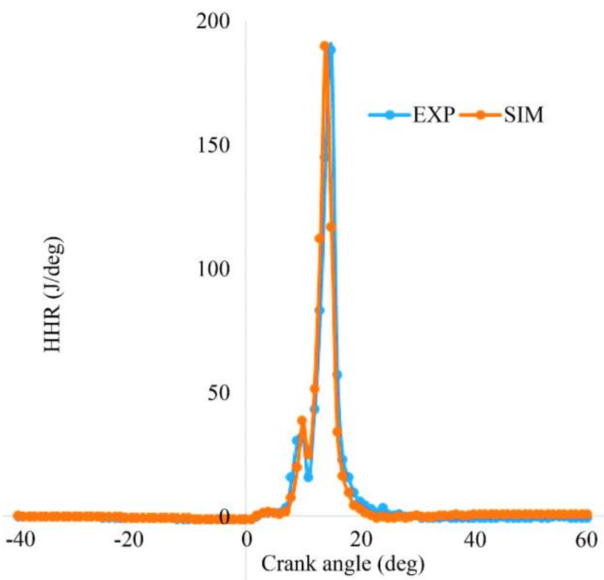
Several methods of solving these equations are discussed in the literature, all of which have as their main objective the calculation of the distribution of temperature and heat flux through the combustion engine parts [32]. Jafari et al. [33] perform an iterative analysis between the 3D KIVA II CFD code and a finite element thermal conduction code to estimate the engine operating characteristics based on heat losses. Then Trujilo et al. [34] present a methodology to predict the temperature of the inner surface of the cylinder using a finite element model.

## 3. Results and Discussion

### *3.1. Model Validation*




**(a)**




**(b)**

**Fig. 3 Model validation: (a) Validation of the model for pressure (b) Validation of the model for heat release rate**

The experimental data mentioned above validated the CFD model simulated in this study. The condition of this engine operating at an engine speed of 1500 rpm with an IMEP of 1 MPa was chosen for this numerical study. Figure 3 below shows the pressure and heat release validations between the measured and calculated values for the diesel engine. We can see that the concordance of the pressure traces is good for validation. Although it overestimates the initial heat release, this model predicts the pressure and heat release rate trend better and then captures the overall combustion characteristics. The expected ignition delay period and the duration of diesel combustion are observed.

### *3.2. Analysis and Discussion*

#### *3.2.1. Effect of Engine Speed on Engine Parameters and Pollutants*

In full variation of the engine speed, the injection of the fuel can, in some cases, cause a quantity of unburned fuel at the end of the cycle in the combustion chamber. By delaying the injection, the time required for the air/fuel mixture to achieve better combustion is reduced, leading to a considerable reduction in the engine consumption rate. This reduced fuel consumption associated with delayed injection will result in a large amount of unburned fuel. From then on, the heat accumulated at the end of the cycle will be released, reducing the injection delay, as shown in Fig. 4b. It is noted that to obtain a large part of the energy of the pulverized fuel, the injection must be done at about 15 degrees ATDC for the higher engine speed and about 10 degrees ATDC for the low engine speed.

Figures 4e and 4f show that the fuel injection timing probably affects the pollutant emissions at varying engine speeds, more precisely, NOx and soot. When fuel injection occurs before 15 degrees ATDC at maximum engine speed, which is 2000 RPM for this study, we observe a large reduction in the level of NOx and soot particles of about 38% and 40%, respectively, between the minimum speed (1500 RPM) and the maximum speed (2000 rpm). This is due to the fact that the ignition time is long enough for the fuel to be fully injected before ignition takes place and has premised combustion, as shown in Fig. 4b. In addition, the evaporation of the fuel before ATDC causes a decrease in the temperature in the cylinder, causing a delay in the ignition of the fuel. This delayed ignition time is certainly responsible for reducing NOx and soot emissions. The reduction of these pollutants can also be explained by the fact that a longer time for a mixture with a precipitated injection will lead to a poor mixture.

The variation of the average temperatures in the cylinder (Fig. 4c) reflects the exact behavior of the cylinder pressures (Fig. 4a). Most of the fuel's energy is released when it is injected earlier before the ATDC, which will result in higher engine temperature peaks at high rpm variations of about 25% between minimum and maximum. At low engine speeds, the exhaust temperature increases because the fuel injection is delayed. By varying the engine speed, the work can reach its maximum. There is a well-defined fuel injection range, so this work is at its maximum at maximum engine speed. The cylinder pressure visible in Fig. 4a clearly indicates that at 1500 RPM, the fuel injects in advance and ignites faster, hence the increase in cylinder pressure before the piston reaches the ATDC. This will lead to a loss of much of the work generated at low engine speeds.

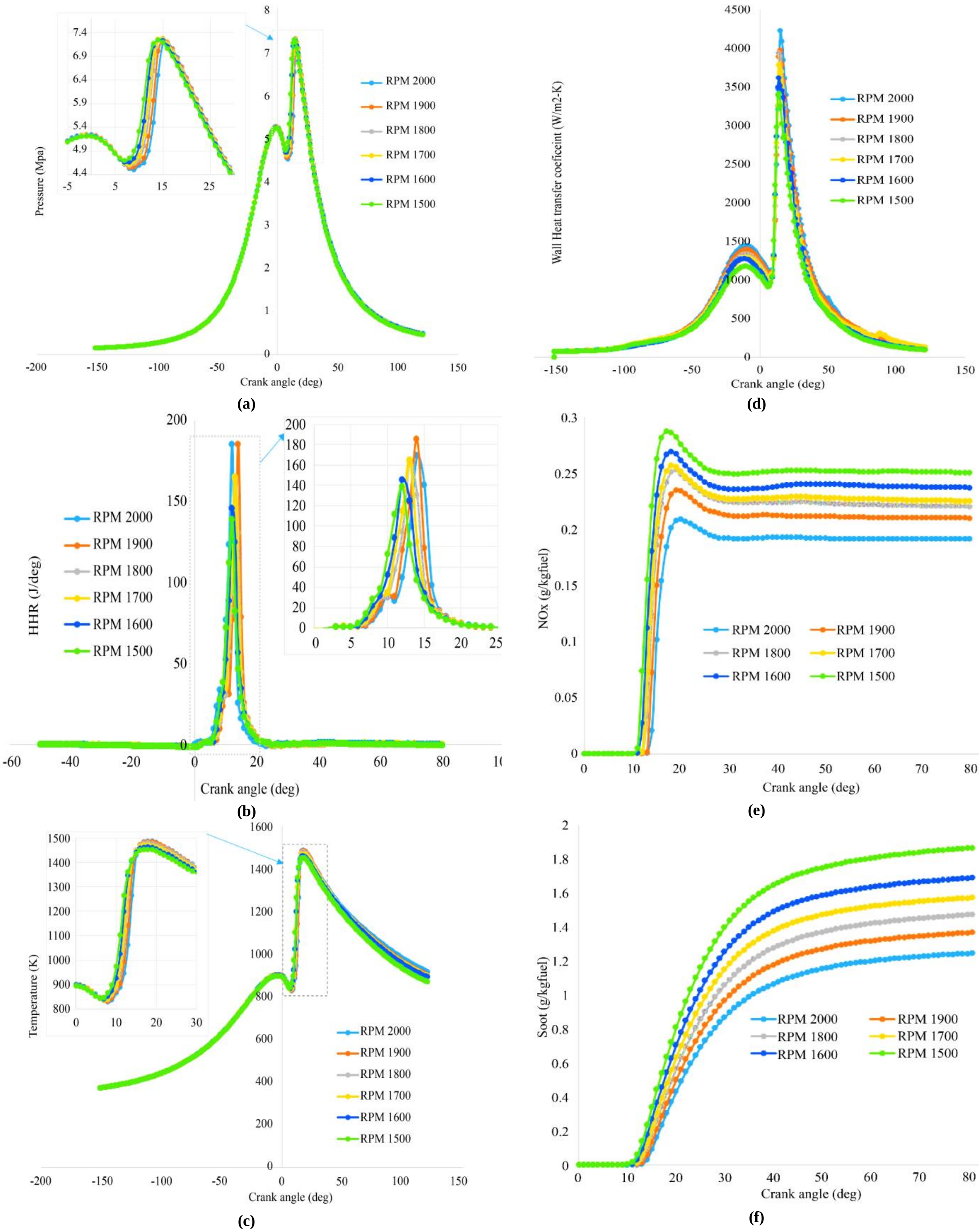



**Fig. 4 Variation: Pressure (a), Heat release rate (b), Temperature (c), Wall heat transfer (d), Nox (e), soot (f).**

Previous studies show that the movement of air affects the combustion process and thus significantly influences the formation of pollutant emissions [35–37]. In this study, the engine's operation was simulated over a wide range of engine speeds (1500 to 2000 RPM) with a pitch of 100 RPM, and a significant variation of different parameters was observed. All these parametric variations in combustion and pollutant formation mean that, at higher engine speeds, fuel is consumed in a much shorter period of time by the improved mixture of air and fuel, according to [37] research. The shorter combustion time allows more time for the formation of pollutants (soot and NOx).

In addition, the right air/fuel mix reduces soot and NOx by reducing the portion of fuel-rich regions. Therefore, high engine speed is expected to significantly reduce Nox and Soot emissions, as shown in Figs. 4e and 4f.

These results are consistent with the study of [38], which shows that when the engine speed increases, the maximum torque decreases because the ratio of excess air continues to increase due to the appearance of the flame. As a result, the exhaust gas temperature rises immediately, which could lead to an increase in heat transfer in the engine walls. According to Fig. 4d, this increase is about 30% for this study. Harsh Goyal et al. [39] show in their research that compression ignition with a homogeneous load offers both high efficiency and very low NOx and soot emissions. During the engine's operating range, this must be limited by an excessive pressure rate that increases the high load area, which is the main reason for the engine clicking. For this problem, illustrated by combustion, the ignition time must be delayed after the ATDC, while the correct control of the gas temperature and combustion time must be adequately ensured.

**Table 3. Contours of combustion parameters**

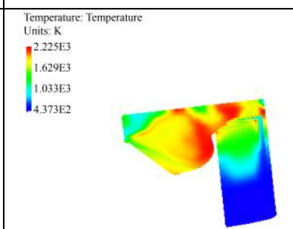
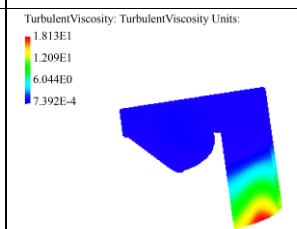
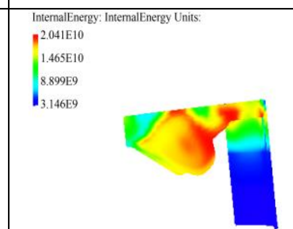
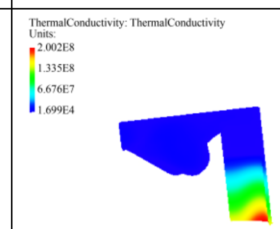
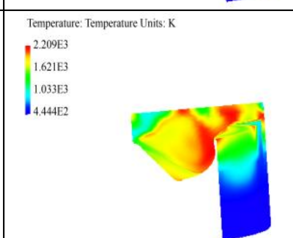
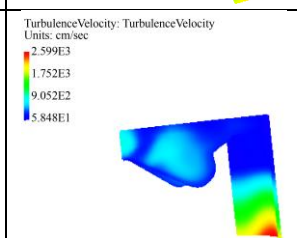
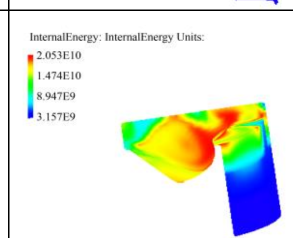
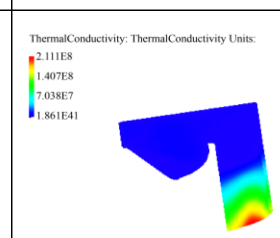
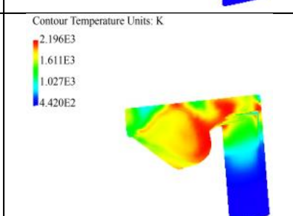
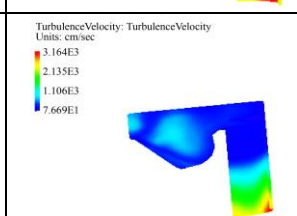
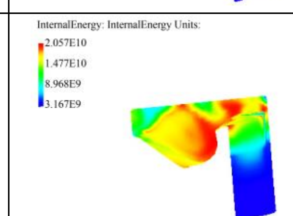
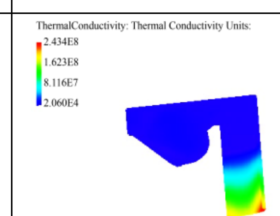
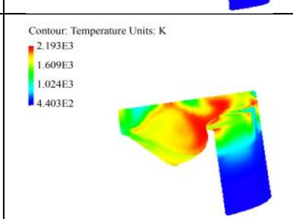
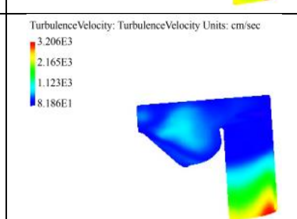
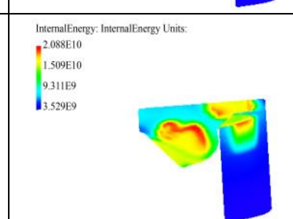
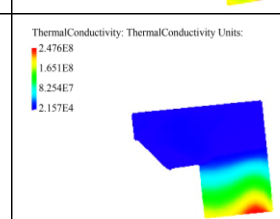
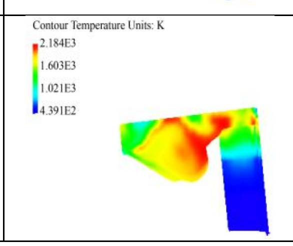
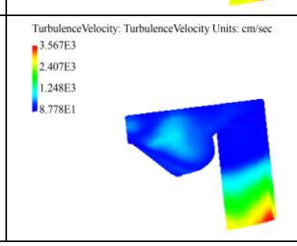
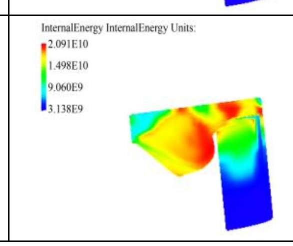
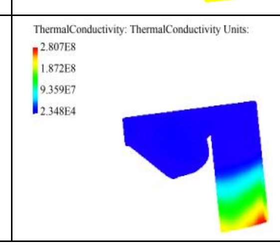
| Engine Speed rpm | Temperature | Turbulence Velocity | Internal Energy | Thermal Conductivity |
|---|---|---|---|---|
| 1500 |  |  |  |  |
| 1700 |  |  |  |  |
| 1800 |  |  |  |  |
| 1900 |  |  |  |  |
| 2000 |  |  |  |  |

Table 3 above provides a more detailed view of some of the diesel engine CFD simulation parameters operating at speeds ranging from 1500 to 2000 RPM. It is noted that there is a temperature variation that decreases with the increase in engine speed before the piston reaches the ATDC, an increase in heat transfer, the speed of the turbulence phenomenon, and the internal energy of the engine. Studies conducted by [40, 41] show that increasing engine speed has a direct effect on turbulence, which in turn greatly influences engine combustion.

The main effect of turbulence is to affect the parietal heat transfer and thus to change the mixture's temperature, which in turn influences the moment of ignition and the burning time, which results in a reduction of pollutant emissions if the combustion phenomenon remains ideal.

## 4. Conclusion

For this study, a CFD mathematical model was adopted, validated, and executed using ANSYS FORTE software to analyze the effect of engine speed on the combustion and pollutant formation processes. Emphasis was placed on the variation of NOx, soot, and key combustion parameters, namely heat release, cylinder pressure, temperature, and heat transfer. This research shows that the varying effects of injection timing and velocity have a significant impact on the combustion process and, therefore, also on pollutants. For simulations carried out at 1500 AND 2000 RPM, we recorded an increase in cylinder pressure (10%), heat release (20%), temperature (8%), and heat transfer coefficient (10%). Then, there was a significant reduction in NOx (25%) and soot (40%) emissions for the same range.

## References


[1] Feiyang Zhao, Wenming Yang, and Wenbin Yu, “A Progress Review of Practical Soot Modeling Development in Diesel Engine Combustion,” *Journal of Traffic and Transportation Engineering (English Edition)*, vol. 7, no. 3, pp. 269-281, 2020. [CrossRef] [Google Scholar] [Publisher Link]

[2] E. Jiaqiang et al., “Effects of Injection Timing and Injection Pressure on Performance and Exhaust Emissions of a Common Rail Diesel Engine Fueled by Various Concentrations of Fish-Oil Biodiesel Blends,” *Energy*, vol. 149, pp. 979-989, 2018. [CrossRef] [Google Scholar] [Publisher Link]

[3] Hamid Omidvarborna, Ashok Kumar, and Dong-Shik Kim, “Recent Studies on Soot Modeling for Diesel Combustion,” *Renewable and Sustainable Energy Reviews*, vol. 48, pp. 635-647, 2015. [CroosRef] [Google Scholar] [Publisher Link]

[4] Haroun A.K. Shahad, and Saad K. Wabdan, “Effect of Operating Conditions on Pollutants Concentration Emitted from a Spark Ignition Engine Fueled with Gasoline Bioethanol Blends,” *Journal of Renewable Energy*, vol. 2015, 2015. [CrossRef] [Google Scholar] [Publisher Link]

[5] A. Uludogan, David E. Foster, and Rolf Reitz, “Modeling the Effect of Engine Speed on the Combustion Process and Emissions in a DI Diesel Engine,” *SAE Technical Paper*, 1996. [CrossRef] [Google Scholar] [Publisher Link]

[6] M.A. Patterson et al., “Modeling the Effects of Fuel Injection Characteristics on Diesel Engine Soot and NOx Emission,” *SAE Transactions*, pp. 836-852, 1994. [Google Scholar] [Publisher Link]

[7] Malin Alrikson, and Ingemar Denbratt, “Low Temperature Combustion in a Heavy-Duty Diesel Engine using High Levels of EGR,” *SAE Technical Paper*, 2006. [CrossRef ] [Google Scholar] [Publisher Link]

[8] Bui Van Ga et al., “Numerical Simulation Studies on Performance, Soot and NOx Emissions of Dual-Fuel Engine Fuelled with Hydrogen Enriched Biogas Mixtures,” *IET Renewable Power Generation*, vol. 12, no. 10, pp. 1111-1118, 2018. [CrossRef] [Google Scholar] [Publisher Link]

[9] Yong Sun, and Rolf D. Reitz, “Modeling Diesel Engine NOx and Soot Reduction with Optimized Two-Stage Combustion,” *SAE Technical paper*, 2006. [CrossRef] [Google Scholar] [Publisher Link]

[10] Daniel A. Nehmer, and Rolf D. Reitz, “Measurement of the Effect of Injection Rate and Split Injections on Diesel Engine Soot and NOx Emissions,” *SAE Transactions*, pp. 1030-1041, 1994. [Google Scholar] [Publisher Link]

[11] Adam B. Dempsey, Scott J. Curran, and Robert M. Wagner, “A Perspective on the Range of Gasoline Compression Ignition Combustion Strategies for High Engine Efficiency and Low NOx and Soot Emissions: Effects of in-Cylinder Fuel Stratification,” *International Journal of Engine Research*, vol. 17, no. 8, pp. 897-917, 2016. [CrossRef] [Googler Scholar] [Publisher Link]

[12] Victor lorga-Siman, “*Study by Numerical Simulation of Flows in the Intake Duct of an Engine with Variable Intake Valve Lift*,” Doctoral Thesis in Energy, 2012. [Google Scholar] [Publisher Link]

[13] H. Weller et al., “Prediction of Combustion in Homogeneous-Charge Spark-Ignition Engine,” *International Symposium COMODIA 94*, pp. 163-169, 1994. [Googler Scholar] [Publisher Link]

[14] Heywood B. John, “Combustion and its Modeling in Spark-Ignition Engine,” *International Symposium COMODIA 94*, pp. 1-15, 1994. [Googler Scholar] [Publisher Link]

[15] David L. Reuss et al., “Particule Image Velocimetry Measurements in a High-Swirl Engine used for Evaluation of Computational Fluid Dynamics Calculations,” *SAE Transactions*, vol. 104, pp. 2073-2092, 1995. [CrossRef] [Google Scholar] [Publisher Link]

[16] G.M. Kosmadakis, E.G. Pariotis, and C.D. Rakopoulos, "Heat Transfer and Crevice Flow in a Hydrogen-Fueled Spark-Ignition Engine: Effect on the Engine Performance and No Exhaust Emissions," *International Journal of Hydrogen Energy*, vol. 38, no. 18, pp. 7477–7489, 2013. [CrossRef] [Googler Scholar] [Publisher Link]

[17] Zhi Wang et al., "A Computational Study of Direct Injection Gasoline HCCI Engine with Secondary Injection," *Fuel*, vol. 85, no. 12–13, pp. 1831–1841, 2006. [CrossRef] [Google Scholar] [Publisher Link]

[18] Francesca Di Mare, Robert Knappstein, and Michael Baumann, "Application of LES Quality Criteria to Internal Combustion Engine Flows," *Computers & Fluids*, vol. 89, pp. 200–213, 2014. [CrossRef] [Google Scholar] [Publisher Link]

[19] Christian Angelberger, and Sebastien Candel, "Editorial-IFP Energies Nouvelles Scientific Meeting LES4ICE 2012-Large-scale Simulation for Flows in Internal Combustion Engines," *Oil & Gas Science and Technology–IFP Energies Nouvelles Review*, vol. 69, no. 1, pp. 3-9, 2014. [CrossRef] [Google Scholar] [Publisher Link]

[20] A. Imren, and D.C. Haworth, "On the Merits of Extrapolation-based ODE Solvers for Combustion CFD, *Combustion and Flame*, vol. 174, pp. 1-15, 2016. [CrossRef] [Google Scholar] [Publisher Link].

[21] Marco Mehl et al., "An Approach for Formulating Surrogates for Gasoline with Application toward a Reduced Surrogate Mechanism for Engine Modeling," *Energy Fuels*, vol. 25, no. 11, pp. 5215-5223, 2011. [CrossRef] [Google Scholar] [Publisher Link]

[22] Lei Zhang, "Parallel Simulation of Engine in-Cylinder Processes with Conjugate Heat Transfer Modeling," *Applied Thermal Engineering*, vol. 142, pp. 232-240, 2018. [CrossRef] [Google Scholar] [Publisher Link]

[23] Stephen Bush, "Sandia's New Medium-Duty Diesel Research Engine and what we're going to do with it," *Sandia National Laboratories*, 2020. [Google Scholar] [Publisher Link]

[24] Mohamed Ali Djemal, "Development of a Simulator for the Diesel Engine to Study Performance and Dynamic Behavior, " *Semaphor, Electronic Document Collections*, pp. 1-108, 2020. [Google Scholar] [Publisher Link]

[25] Stephen Busch et al., "Experimental and Numerical Studies of Bowl Geometry Impacts on Thermal Efficiency in a Light-Duty Diesel Engine," *SAE Technical Paper*, 2018. [CrossRef] [Google Scholar] [Publisher Link]

[26] Stephen Busch et al., "Bowl Geometry Effects on Turbulent Flow Structure in a Direct Injection Diesel Engine," *SAE Technical Paper*, 2018, [CrossRef] [Google Scholar] [Pulisher link].

[27] Zha Kan et al., "A Study of Piston Geometry Effects on Late-Stage Combustion in a Light-Duty Optical Diesel Engine using Combustion Image Velocimetry," *SAE International Journal of Engines*, vol. 11, no. 6, pp. 783-804, 2018. [Google Scholar] [Publisher Link]

[28] K. Theofanidis, Y. Lu, and A. Kovacevic, "CFD Analysis on the Effect of Discharge Port Geometry of the Hook and Claw Vacuum Pumps," *IOP Conference Series: Materials Science and Engineering*, vol. 1180, pp. 1-10, 2021. [CrossRef] [Google Scholar] [Publisher Link]

[29] ANSYS Forte Simulate Guide, Forte Tutorials, Forte Theory Manual, 2020. [Online]. Available: https://academica-e.unavarra.es/xmlui/bitstream/handle/2454/37481/Memoria%20TFG_Sergio%20Pe%26%23769%3Brez%20de%20Albe%26%23769%3Bniz%20Azqueta_ANSYS%20Forte%20Simulate%20Guide.pdf?sequence=1&isAllowed=y

[30] Oleg Zikanov, *Essential Computational fluid Dynamics*, 2nd ed., John Wiley Sons, 2019. [Google Scholar] [Publisher Link]

[31] B.B. Sahoo, N. Sahoo, and U.K. Saha, "Effect of Engine Parameters and Type of Gaseous Fuel on the Performance of Dual-Fuel Gas Diesel Engines—A Critical Review," *Renewable and Sustainable Energy Reviews*, vol. 13, no. 6-7, pp. 1151-1184, 2009. [CrossRef] [Google Scholar] [Publisher Link]

[32] Vipin Mehta, Sanjay Kateray, and P.L. Verma, "Calculation Analysis of Heat Transfer on the Surface of Engine Cylinders with Unequal Size of Material and Expanded Surfaces," *SSRG International Journal of Mechanical Engineering*, vol. 9, no. 2, pp. 1-8, 2022. [CrossRef] [Publisher Link]

[33] Ali Jafari, and Siamak Kazemzadeh Hannani, "Effect of Fuel and Engine Operational Characteristics on the Heat Loss from Combustion Chamber Surfaces of SI Engines," *International Communications in Heat and Mass Transfer*, vol. 33, no. 1, pp. 122–134, 2006. [CrossRef] [Google Scholar] [Publisher Link]

[34] Elisa Carvajal Trujillo et al., "Methodology for the Estimation of Cylinder Inner Surface Temperature in an Air-Cooled Engine," *Applied Thermal Engineering*, vol. 31, pp. 1474–1481, 2011. [CrossRef] [Publisher Link]

[35] Magnus Sjoberg, and John E. Dec, "Effects of Engine Speed, Fueling Rate and Combustion Phasing on the Thermal Stratification Required to Limit HCCI Knocking Intensity," *SAE Transactions*, vol. 114, pp. 1472-1486, 2005. [Google Scholar] [Publisher Link]

[36] B. Murali Krishna, and J.M. Mallikarjuna, "Effect of Engine Speed on In-Cylinder Tumble Flows in A Motored Internal Combustion Engine-an Experimental Investigation using Particule Image Velocimetry," *Journal of Applied Fluid Mechanics*, vol. 4, no. 1, pp. 1-14, 2011. [CrossRef] [Google Scholar] [Publisher Link]

[37] Magnus Sjoberg, and John E. Dec, "Combined Effect of Fuel-Type and Engine Speed on Intake Temperature Requirements and Completeness of Bulk-Gas Reactions for HCCI Combustion," *SAE Technical Paper*, 2003. [CrossRef] [Google Scholar] [Publisher Link]

[38] Cheolwoong Park et al., "The Effect of Engine Speed and Cylinder-to-Cylinder Variations on Backfire in a Hydrogen-Fueled Internal Combustion Engine," *International Journal of Hydrogen Energy*, vol. 44, no. 39, pp. 22223-22230, 2019. [CrossRef] [Google Scholar]

[Publisher Link]
[39] Harsk Goyal, Sanghoon Kook, and Evatt Hawkes, "Influence of Engine Speed on Gasoline Compression Ignition (GCI) Combustion in a Single-Cylinder Light-Duty Diesel Engine," *SAE Technical Paper*, 2017. [CrossRef] [Google Scholar] [Publisher Link]
[40] Taehoon Kim, Jingeum Song, and Parc Sunwook, "Effects of Turbulence Enhancement on Combustion Process using a Double Injection Strategy in Direct Injection Spark-Ignition Gasoline Engine," *International Journal of Heat and Fluid Flow*, vol. 566, pp. 124-136, 2015. [CrossRef] [Google Scholar] [Publisher Link]
[41] Song-Charng Kong et al., "Modeling the Effects of Geometry Generated Turbulence on HCCI Engine Combustion," *SAE Transaction*, vol. 112, pp. 1511-1521, 2003. [CrossRef] [Google Scholar] [Publisher Link]