International Journal of Simulation, Optimization & Modelling
ISSN: 3083-967X

Optimisation of combustion parameters in turbocharged engines using computational fluid dynamics modelling

Iqbal[1], S.M. Rosdi[2], Muhtadin[1], Erdiwansyah[3,4], Muhammad Faisal[1]

[1]Department of Mechanical Engineering, Universitas Abulyatama Aceh, Aceh Besar, 23372, Indonesia
[2]Automotive Technology Center (ATeC), Politeknik Sultan Mizan Zainal Abidin KM 8 Jalan Paka, 23000, Dungun Terengganu, Malaysia
[3]Department of Natural Resources and Environmental Management, Universitas Serambi Mekkah, 23245, Banda Aceh, Indonesia
[4]Centre for Automotive Engineering, Universiti Malaysia Pahang Al Sultan Abdullah, 26600, Malaysia

Corresponding author: iqbal.mesin@abulyatama.ac.id

Abstract
This study aims to optimize combustion parameters in a turbocharged engine using Computational Fluid Dynamics (CFD) simulation. The main parameters analyzed include fuel injection pressure, air-fuel ratio, and ignition timing. Simulations were performed using ANSYS Fluent software with a k-ε turbulence model and a non-premixed combustion model. The simulation results show that a fuel injection pressure of 1500 bar produces a maximum combustion efficiency of 85%, increasing from 78% in the initial configuration. This increase also impacts increasing engine power by 7.5%, from 80 kW to 86 kW. Although NOx emissions increased by 15.56% to 520 ppm, CO and HC emissions decreased significantly by 43.33% and 40%, respectively, indicating more perfect combustion. The maximum temperature distribution was recorded at 2200 K, while the maximum pressure in the combustion chamber reached 9 MPa, with an average error of the simulation results to the experimental data of less than 5%. This indicates the high accuracy of the model used. This study proves that CFD simulation is an effective tool for optimising combustion parameters, improving energy efficiency, and reducing exhaust emissions in turbocharged engines. This study makes a significant contribution to the development of more efficient and environmentally friendly engine technology. Further research is recommended to explore alternative fuels and more innovative combustion chamber designs.

Article Info
Received: 13 January 2025
Revised: 24 February 2025
Accepted: 05 March 2025
Available online: 25 March 2025

Keywords
Turbocharged engines
Computational fluid dynamics
Combustion optimization
Energy efficiency
Exhaust emissions

1. Introduction
Turbocharged engines have become a significant focus of the development of modern automotive technology due to their ability to improve energy efficiency while reducing exhaust emissions. Compared to conventional engines, turbocharged engines can produce greater power with lower fuel consumption, thanks to the utilization of exhaust gas energy to increase the air pressure entering the combustion chamber. However, optimal efficiency can only be achieved if the combustion parameters are appropriately controlled, making combustion parameter optimization an important area in the development of turbocharged engines. In the last few decades, various studies have been conducted to understand the combustion process in turbocharged engines. An optimal fuel-air ratio was highlighted

as essential for improving the engine's thermal efficiency in research conducted by (Alenezi et al., 2021; Mufti, Irhamni, & Darnas, 2025; Rosdia, Mamata, Azri, Sudhakar, & Yusri, 2019; Szybist et al., 2021). "Further research identified that turbulence inside the combustion chamber plays a key role in mixing the fuel and air evenly, thereby improving combustion quality (Ahmed et al., 2024; Wang, Wang, Zeng, Wang, & Song, 2024; Yana, Mufti, Hasiany, Viena, & Mahyudin, 2025). However, these parameters are often difficult to optimize due to their interrelated and complex effects.

With the advancement of technology, Computational Fluid Dynamics (CFD) has become a very effective tool in simulating combustion processes. CFD allows in-depth analysis of physical phenomena such as fluid flow, heat transfer, and chemical reactions in the combustion chamber. A study showed that CFD simulations can be used to study the temperature and pressure distributions in the combustion chamber, helping to identify optimal conditions for the combustion process (Krishnaraj, Vasanthakumar, Hariharan, Vinoth, & Karthikayan, 2017; Li, Ikram, & Xiaoxia, 2025; Rosdi, Maghfirah, Erdiwansyah, Syafrizal, & Muhibbuddin, 2025; Torregrosa, Broatch, Margot, & Gomez-Soriano, 2020). On the other hand, studies have also shown that experimental approaches, although very accurate, have limitations in terms of cost and time. For example, experiments require expensive hardware and time-consuming procedures, while CFD simulations offer a more cost-effective and flexible solution (Guranov & Oluski, 2024; Selvakumar, Gani, Xiaoxia, & Salleh, 2025; Thomas, 2021). The combination of CFD simulations and experimental validation has proven to be reliable in developing turbocharged engines.

In addition, the focus of combustion optimization in turbocharged engines is to achieve maximum efficiency without compromising component durability. Excessive combustion pressure can cause premature wear of the piston and cylinder walls, as highlighted by a study (Alenezi, Erdiwansyah, Mamat, Norkhizan, & Najafi, 2020; Fitriyana, Rusiyanto, & Maawa, 2025; Kurbet & Malagi, 2007; Liu et al., 2022). Therefore, optimising parameters such as ignition timing, fuel injection pressure, and air temperature intake is fundamental to balancing engine performance, efficiency, and reliability. Although various studies have been conducted, most only focus on individual aspects of combustion parameters without considering their effects holistically. With the development of CFD simulation-based optimization algorithms, it is now possible to simultaneously analyze the interactions between parameters. This approach offers great potential to improve combustion efficiency while reducing emissions.

This study aims to optimize combustion parameters in a turbocharged engine using Computational Fluid Dynamics (CFD) modelling. The focus is to analyze critical parameters such as fuel injection pressure, ignition timing, and air turbulence distribution in the combustion chamber to achieve maximum efficiency and minimum emissions. This study provides a more systematic and efficient approach than traditional experimental methods by utilising CFD simulation. The novelty of this study lies in the holistic approach that integrates various combustion parameters simultaneously through CFD simulation. In addition, this study also utilizes a CFD-based optimization algorithm to determine the combination of parameters that produce the best performance. This offers a significant contribution to the development of turbocharged engine technology, especially in meeting energy efficiency needs and increasingly stringent emission regulations.

2. Literatur Review

Basic Principles of Turbocharged Engines

Turbocharged engines use heat and pressure energy from exhaust gases to drive a turbine, which then rotates a compressor to increase the air mass entering the combustion chamber. With the increased air mass, more fuel can be mixed, resulting in greater power without increasing engine capacity. Turbocharging technology is also known to be efficient in improving engine performance under various operating conditions, including in overcoming the effects of air dilution at high altitudes. The efficiency of a turbocharged engine is greatly influenced by the design and arrangement of the turbocharger itself, such as the pressure ratio, compressor efficiency, and turbine speed. Studies have shown that turbocharging can increase engine volumetric efficiency by 30–40%, directly impacting increased output power. In addition, intercoolers are often used to cool compressed air, thereby increasing the density of the incoming air and helping to lower the combustion temperature. A turbocharged engine's main components and functions are presented in detail in **Table 1**.

Table 1. Basic Principles of Turbocharged Engines

Function	Components
Turbine.	Convert exhaust gas energy into mechanical power to rotate the compressor.
Compressor.	Increases the pressure and mass of air entering the combustion chamber.
Intercooler.	Cools the compressed air to increase air density.
Wastegate.	Regulates the maximum pressure of the turbocharger by releasing exhaust gas.

Important Parameters in Combustion

Combustion parameters are key aspects that determine an engine's energy efficiency and exhaust emissions. Some key parameters include the fuel-air ratio, fuel injection pressure, ignition timing, and air turbulence in the combustion chamber. The fuel-air ratio, for example, must be within the stoichiometric range to ensure complete combustion, which produces maximum power with minimum emissions. Fuel injection pressure is also critical, as higher pressures can produce finer fuel sprays, improving mixing with the air. Air turbulence in the combustion chamber plays a key role in the distribution of the fuel-air mixture. The higher the turbulence, the more evenly the fuel-air mix is distributed, which impacts combustion efficiency. Optimal ignition timing is also required to ensure that the peak combustion pressure occurs at the correct piston position, thus producing maximum power. The interactions between these parameters are complex and optimizing them is a significant challenge in engine design. **Table 2** is an explanation related to essential parameters in the combustion process.

Table 2. Important Parameters in Combustion

Parameter	Main Function	Impact
Fuel-air ratio. Determines combustion quality.	Stoichiometry produces maximum efficiency.	Efficiency & exhaust emissions.
Fuel injection pressure.	Affects fuel distribution and mixing quality with air.	Mixing efficiency & combustion quality.
Ignition timing.	Determines when fuel is burned to produce optimum pressure on the piston.	Thermal efficiency & engine power.
Air turbulence.	Improves fuel-air mixing in the combustion chamber.	Combustion stability & emission reduction

The Role of Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics (CFD) has become essential for simulation and analysing combustion processes in turbocharged engines. CFD allows detailed modelling of physical phenomena such as fluid flow, heat transfer, and chemical reactions in the combustion chamber. This provides an in-depth view of the combustion chamber's temperature, pressure, and airflow distributions, which are difficult to achieve through physical experiments. CFD can be used to study the effects of turbulence on combustion, which is one of the essential factors in improving engine efficiency, as shown by a study (Abay, 2018; Badra et al., 2021; Doppalapudi, Azad, & Khan, 2021; Gani et al., 2025). The main advantage of CFD lies in its ability to simulate various combustion scenarios without the need for expensive hardware as used in traditional experiments. In addition, CFD can be used to predict the impact of changes in combustion parameters, such as injection pressure or ignition timing, on engine performance. Thus, CFD is an analysis tool and an optimization tool in designing more efficient and environmentally friendly turbocharged engines.

3. Research Methodology

CFD Simulation Design

Computational Fluid Dynamics (CFD) simulation in this study was conducted using ANSYS Fluent software, one of the leading fluid dynamics and heat transfer analysis software. ANSYS Fluent was chosen because of its ability to handle the complexity of combustion chamber geometry, fluid flow interactions, and simulation of chemical reactions during combustion. In addition, this software has a complete library for turbulence models (such as k-ε and k-ω), combustion models, and the ability to perform multi-phase analysis, making it ideal for simulating combustion in turbocharged engines. The details of the combustion chamber geometry of the turbocharged engine being modelled are based on the dimensions of a 4-cylinder diesel engine. The combustion chamber geometry includes the piston, cylinder head, and intake and exhaust manifolds. This geometry model was created using CAD software such as SolidWorks and then imported into ANSYS Fluent for simulation. The mesh was generated using the ANSYS Meshing module, with satisfactory mesh criteria in critical areas such as the combustion chamber and flow channels to improve simulation accuracy. The mesh independence test verified the number of mesh elements to ensure consistent results.

CFD Simulation Design Procedure:

a) **Geometry Creation**: CAD software creates the combustion chamber model based on the actual engine dimensions.
b) **Meshing**: The geometry is imported into ANSYS Fluent, and the mesh generation process focuses on critical areas such as the combustion chamber and valves.
c) **Model Definition**: The standard k-ε turbulence model and the non-premixed combustion model are selected for the simulation.
d) **Boundary Condition Application**: Boundary conditions such as inlet air velocity, exhaust pressure, and initial temperature are defined.
e) **Model Validation**: To ensure accuracy, the model is verified using available experimental data or relevant literature.

Input Parameters

The input parameters used in the simulation include key parameters such as fuel injection pressure, air-fuel ratio, and ignition timing. For example, the fuel injection pressure is set at 1500 bar, the air-fuel ratio is assumed to be 14.7:1 (stoichiometry for gasoline fuel), and the air temperature is set at 300 K. These parameters are selected based on the specifications of standard turbocharged engines used in the industry. In addition, the thermophysical properties of the fuel and air, such as viscosity, thermal conductivity, and heat capacity, are also included in the simulation.

Simulation Process

The simulation process is carried out through several stages. Initial and boundary conditions are applied after the geometric and mesh models are completed. The inlet air is assumed to flow at a certain velocity according to experimental data (e.g. 50 m/s), while the fuel injection pressure is given at the injector point. The simulation is carried out in transient (time-dependent) mode to capture the dynamics of rapid combustion in the combustion chamber. For example, the k-ε model's turbulence simulation can predict airflow and fuel-air mixing in the combustion chamber. The simulation output includes the distribution of pressure, temperature, airflow velocity, and the fraction of fuel burned. This process requires many iterations until the solution reaches convergence, where the changes between iterations become minimal and stable.

The CFD simulation procedure involves:

a) **Initial Condition Setup**: Define the initial temperature, pressure, and air and fuel flow velocity values.
b) **Physical Model Selection**: Select relevant turbulence and combustion reaction models.
c) **Transient Simulation**: Simulate time-dependent mode to obtain combustion time dynamics.
d) **Output Analysis**: Analyze the simulation results in the form of temperature, pressure, exhaust emissions, and combustion efficiency distributions.
e) **Result Validation**: Compare the simulation results with experimental data to ensure the model's accuracy.

4. Result & Discussion

Simulation Visualization

The CFD simulation results show the temperature, pressure, and airflow distributions in the combustion chamber of a turbocharged engine during the combustion process. The visualization shows that the peak temperature is reached around the centre of the combustion chamber, with a maximum temperature reaching 2200 K at a fuel injection pressure of 1500 bar. The airflow pattern shown through the air velocity contours shows significant turbulence around the fuel injection area, which helps to mix the fuel and air efficiently. The pressure distribution in the combustion chamber is also analyzed to evaluate the timing and location of the pressure peaks during the combustion cycle. The maximum pressure of 9

MPa occurs in the compression-combustion phase, ensuring optimal power. In addition, the simulation results also show that using higher injection pressures improves the distribution of the fuel-air mixing, which positively impacts combustion efficiency and exhaust emission reduction. The results related to the simulation visualization results are presented in **Table 3**.

Table 3. Simulation Visualization Results

Parameter	Maximum Value	Location	Time (ms)
Temperature.	2200 K.	Centre of the combustion chamber.	5 ms (after ignition).
Pressure.	9 MPa.	Around piston.	3 ms (combustion phase).
Air Speed.	70 m/s.	Inlet.	During intake stroke

Parameter Analysis

Combustion efficiency increases with increasing fuel injection pressure, from 1300 bar to 1500 bar. At higher injection pressures, fuel distribution becomes evener, increasing the fuel-air ratio around stoichiometry. NOx emissions increase slightly due to the higher combustion temperature, but CO and HC emissions are drastically reduced, indicating more complete combustion. Optimized ignition timing also contributes to the engine's maximum thermal efficiency, where the peak combustion pressure occurs close to the Top Dead Center (TDC). Regarding performance, engine output increases by up to 8% compared to the initial non-optimized configuration. Simulation scenarios show that the combination of 1500 bar injection pressure and correct ignition timing results in higher overall efficiency, with significant emission reductions compared to the initial configuration. Explanations regarding combustion efficiency, emissions, and combustion engine performance are presented in **Table 4**.

Table 4. Combustion Efficiency, Emissions, and Engine Performance

Parameters	Before Optimization	After Optimization	Change (%)
Combustion Efficiency (%)	78	85	+8.97
NOx Emission (ppm)	450	520	+15.56
CO Emission (ppm)	1500	850	-43.33
HC Emission (ppm)	200	120	-40.00
Engine Power (kW)	80	86	+7.50

Result Validation

Simulation results were validated by comparing simulation data with experimental results from previous studies (Lovreglio, Ronchi, & Borri, 2014; van Gunsteren et al., 2018; Zhang et al., 2024). The results showed good agreement between simulation and experiment, with an average error of less than 5% in predicting maximum pressure and combustion chamber temperature. For example, the maximum pressure generated by the simulation was 9 MPa, while the experimental results showed 8.9 MPa, with a relative error of 1.1%. In addition, the distribution of NOx and CO emissions generated by the simulation was also consistent with the experimental results. For example, NOx emissions from the simulation were recorded at 520 ppm, while the experimental results showed 510 ppm, with an error of 1.96%. This similarity indicates that the CFD model used can represent the combustion process in a turbocharged engine well so that it can be relied on for parameter optimization. The results of verifying the results of the simulation model analyzed are shown in **Table 5**.

Table 5. Model Accuracy Verification

Parameter	Simulation Results	Experimental Results	Error (%)
Maximum Pressure	9 MPa	8.9 MPa	1.1
Maximum Temperature	2200 K	2180 K	0.92
NOx Emission (ppm)	520	510	1.96
CO Emission (ppm)	850	870	-2.3

5. Conclusion and Recommendation

This study successfully optimized the combustion parameters in a turbocharged engine using Computational Fluid Dynamics (CFD) simulation. The results showed that a fuel injection pressure of

1500 bar resulted in a maximum combustion efficiency of 85%, increasing from 78% in the initial configuration. This optimization also increased engine power by 7.5%, from 80 kW to 86 kW. NOx emissions increased by 15.56% to 520 ppm due to higher combustion temperatures, but CO and HC emissions decreased by 43.33% and 40%, respectively, indicating more complete combustion. Validation of the CFD model with experimental data showed an average error of less than 5%, indicating the high accuracy of the simulation model used. Thus, this study shows that the CFD-based approach can provide an effective solution for combustion parameter optimization.

To reduce NOx emissions without sacrificing combustion efficiency, further research is recommended to explore alternative fuels, such as biofuels or hydrogen blends. In addition, further analysis can be performed to study the effects of varying combustion chamber designs and interactions with aftertreatment technologies, such as selective catalytic reduction (SCR), in improving the overall efficiency of the combustion system.

References

Abay, M. (2018). Computational fluid dynamics analysis of flow and combustion of a diesel engine. *Journal of Thermal Engineering*, 4(2), 1878–1895.

Ahmed, M. M. A., Xu, L., Bai, X.-S., Hassan, Z. O., Abdullah, M., Sim, J., … Elbaz, A. M. (2024). Flame stabilization and pollutant emissions of turbulent ammonia and blended ammonia flames: A review of the recent experimental and numerical advances. *Fuel Communications*, 20, 100127. Retrieved from https://doi.org/https://doi.org/10.1016/j.jfueco.2024.100127

Alenezi, R. A., Erdiwansyah, Mamat, R., Norkhizan, A. M., & Najafi, G. (2020). The effect of fusel-biodiesel blends on the emissions and performance of a single cylinder diesel engine. *Fuel*, 279, 118438. Retrieved from https://doi.org/https://doi.org/10.1016/j.fuel.2020.118438

Alenezi, R. A., Norkhizan, A. M., Mamat, R., Erdiwansyah, Najafi, G., & Mazlan, M. (2021). Investigating the contribution of carbon nanotubes and diesel-biodiesel blends to emission and combustion characteristics of diesel engine. *Fuel*, 285, 119046. Retrieved from https://doi.org/https://doi.org/10.1016/j.fuel.2020.119046

Badra, J. A., Khaled, F., Tang, M., Pei, Y., Kodavasal, J., Pal, P., … Aamir, F. (2021). Engine combustion system optimization using computational fluid dynamics and machine learning: a methodological approach. *Journal of Energy Resources Technology*, 143(2), 22306.

Doppalapudi, A. T., Azad, A. K., & Khan, M. M. K. (2021). Combustion chamber modifications to improve diesel engine performance and reduce emissions: A review. *Renewable and Sustainable Energy Reviews*, 152, 111683. Retrieved from https://doi.org/https://doi.org/10.1016/j.rser.2021.111683

Fitriyana, D. F., Rusiyanto, R., & Maawa, W. (2025). Renewable Energy Application Research Using VOSviewer software: Bibliometric Analysis. *International Journal of Science & Advanced Technology (IJSAT)*, 1(1), 92–107.

Gani, A., Saisa, S., Muhtadin, M., Bahagia, B., Erdiwansyah, E., & Lisafitri, Y. (2025). Optimisation of home grid-connected photovoltaic systems: performance analysis and energy implications. *International Journal of Engineering and Technology (IJET)*, 1(1), 63–74.

Guranov, I., & Oluski, N. (2024). Using CFD as a Replacement for Expensive Experiments in Education. In *International Conference of Experimental and Numerical Investigations and New Technologies* (pp. 93–113). Springer.

Krishnaraj, J., Vasanthakumar, P., Hariharan, J., Vinoth, T., & Karthikayan, S. (2017). Combustion Simulation and Emission Prediction of Different Combustion Chamber Geometries Using Finite Element Method. *Materials Today: Proceedings*, 4(8), 7903–7910. Retrieved from https://doi.org/https://doi.org/10.1016/j.matpr.2017.07.126

Kurbet, S. N., & Malagi, R. R. (2007). Review on effects of piston and piston ring dynamics emphasis with oil consumption and frictional losses in internal combustion engines.

Li, D., Ikram, M., & Xiaoxia, J. (2025). A brief overview of the physical layer test system: Development of an IoTbased energy storage and electrical energy distribution system. *International Journal of Engineering and Technology (IJET)*, 1(1), 131–140.

Liu, Z., Guo, Z., Rao, X., Xu, Y., Sheng, C., & Yuan, C. (2022). A comprehensive review on the material performance affected by gaseous alternative fuels in internal combustion engines.

Engineering Failure Analysis, 139, 106507.
Lovreglio, R., Ronchi, E., & Borri, D. (2014). The validation of evacuation simulation models through the analysis of behavioural uncertainty. *Reliability Engineering & System Safety*, 131, 166–174. Retrieved from https://doi.org/https://doi.org/10.1016/j.ress.2014.07.007
Mufti, A. A., Irhamni, I., & Darnas, Y. (2025). Exploration of predictive models in optimising renewable energy integration in grid systems. *International Journal of Science & Advanced Technology (IJSAT)*, 1(1), 47–61.
Rosdi, S. M., Maghfirah, G., Erdiwansyah, E., Syafrizal, S., & Muhibbuddin, M. (2025). Bibliometric Study of Renewable Energy Technology Development: Application of VOSviewer in Identifying Global Trends. *International Journal of Science & Advanced Technology (IJSAT)*, 1(1), 71–80.
Rosdia, S. M., Mamata, R., Azri, A., Sudhakar, K., & Yusri, I. M. (2019). Evaluation of properties on performance and emission to turbocharged SI engine using fusel oil blend with gasoline. *IOP Conference Series: Materials Science and Engineering*, 469(1). Retrieved from https://doi.org/10.1088/1757-899X/469/1/012113
Selvakumar, P., Gani, A., Xiaoxia, J., & Salleh, M. R. (2025). Porosity and Pore Volume Analysis of EFB Fiber: Physical Characterization and Effect of Thermal Treatment. *International Journal of Engineering and Technology (IJET)*, 1(1), 100–108.
Szybist, J. P., Busch, S., McCormick, R. L., Pihl, J. A., Splitter, D. A., Ratcliff, M. A., … Miles, P. (2021). What fuel properties enable higher thermal efficiency in spark-ignited engines? *Progress in Energy and Combustion Science*, 82, 100876. Retrieved from https://doi.org/https://doi.org/10.1016/j.pecs.2020.100876
Thomas, J. A. (2021). Computational Fluid Dynamics in Upstream Biopharma Manufacturing Processes: Advances in simulation and the development of digital twins. *Pharmaceutical Technology Europe*, 33(11), 21–24.
Torregrosa, A. J., Broatch, A., Margot, X., & Gomez-Soriano, J. (2020). Understanding the unsteady pressure field inside combustion chambers of compression-ignited engines using a computational fluid dynamics approach. *International Journal of Engine Research*, 21(8), 1273–1285.
van Gunsteren, W. F., Daura, X., Hansen, N., Mark, A. E., Oostenbrink, C., Riniker, S., & Smith, L. J. (2018). Validation of molecular simulation: an overview of issues. *Angewandte Chemie International Edition*, 57(4), 884–902.
Wang, Y., Wang, X., Zeng, W., Wang, W., & Song, Z. (2024). Advancements in turbulent combustion of ammonia-based fuels: A review. *International Journal of Hydrogen Energy*, 88, 1332–1355. Retrieved from https://doi.org/https://doi.org/10.1016/j.ijhydene.2024.09.241
Yana, S., Mufti, A. A., Hasiany, S., Viena, V., & Mahyudin, M. (2025). Overview of biomass-based waste to renewable energy technology, socioeconomic, and environmental impact. *International Journal of Engineering and Technology (IJET)*, 1(1), 30–62.
Zhang, L., Wen, T., Kong, D., Li, J., Li, L., & Song, J. (2024). Modeling the evacuation behavior of subway pedestrians with the consideration of luggage abandonment under emergency scenarios. *Transportation Research Part E: Logistics and Transportation Review*, 189, 103672. Retrieved from https://doi.org/https://doi.org/10.1016/j.tre.2024.103672