E3S Web of Conferences **723**, 04002 (2026) https://doi.org/10.1051/e3sconf/202672304002
*AIEI 2026*

# A Physics-Informed CFD–Machine Learning Framework for Piston Bowl Optimization Toward Enhanced Tumble in a Small Diesel Engine

Giang Luong Huynh
Faculty of Automotive Engineering Technology
Industrial University of Ho Chi Minh City (IUH), Vietnam
Ho Chi Minh City, Vietnam
luonghuynhgiang@iuh.edu.vn

Luan Phan Vu
Faculty of Automotive Engineering Technology
Industrial University of Ho Chi Minh City (IUH), Vietnam
Ho Chi Minh City, Vietnam
phanvuluan1410@gmail.com

Chau Vo Tan
Faculty of Automotive Engineering Technology
Industrial University of Ho Chi Minh City (IUH), Vietnam
Ho Chi Minh City, Vietnam
* votanchau@iuh.edu.vn

Duong Phan Nguyen Quoc
Faculty of Automotive Engineering Technology
Industrial University of Ho Chi Minh City (IUH), Vietnam
Ho Chi Minh City, Vietnam
21043931.duong@student.iuh.edu.vn

***Abstract*—Strong in-cylinder tumble motion enhances turbulent mixing and combustion efficiency in compression ignition engines. However, the geometry–tumble relationship remains poorly quantified, particularly in small agricultural diesel engines where design is largely empirical. This study introduces a physics-informed, data-driven framework for combustion chamber optimization in a Vikyno RV165-2 diesel engine operating within a speed range of 1600–2200 rpm, with explicit emphasis on tumble enhancement. A parametric design space defined by six non-conventional geometric variables was systematically explored through 100 high-fidelity CFD simulations using ANSYS ICE. The resulting dataset was used to train a machine-learning-based surrogate model, enabling rapid prediction of the tumble ratio and efficient identification of the optimal design. The results indicate that the optimized configuration (A = 59 mm, B = 35 mm, H = 14.38 mm, L = 8 mm, X = 10 mm, Y = 10 mm) achieved a 56.81% increase in average tumble ratio relative to the baseline. Flow field analysis reveals that this improvement is driven by enhanced flow redirection toward the piston bowl and intensified large-scale vortex structures during the late compression stroke. The proposed CFD–machine learning framework provides an efficient strategy for combustion chamber design, improving geometry–flow coupling and air–fuel mixing.**

***Keywords-Piston bowl, Ansys-ICE, Tumble ratio, Machine Learning***

## I. INTRODUCTION

The advantages of diesel engines in terms of efficiency and durability are not sufficient to drive broader application, as they remain among the primary sources of soot and NOx emissions [1], [2]. Therefore, improving both the performance and emission characteristics of diesel engines, particularly small, single-cylinder engines widely used in agricultural applications, is essential.

Previous studies have proposed various approaches to enhance engine performance. Some researchers have focused on the use of alternative fuels such as biodiesel and pyrolysis oil to improve combustion characteristics [3], [4]. Although these approaches show promise, they often face challenges related to fuel processing, cost, scalability, and instability in fuel properties. Other studies have aimed to improve in-cylinder flow characteristics by modifying the intake system or applying techniques to enhance turbulence intensity, tumble ratio, and swirl ratio [5]-[10]. However, these solutions typically require complex designs and high costs, limiting their applicability to low-cost agricultural engine systems.

In addition, several studies have explored the optimization of combustion chamber geometry using Computational Fluid Dynamics (CFD) tools such as ANSYS and KIVA [11]-[15]. While CFD methods provide detailed insights into in-cylinder flow and heat transfer, their high computational cost and trial-and-error nature restrict the comprehensive exploration of large geometric designs.

In recent years, Machine Learning (ML) techniques, including Artificial Neural Networks (ANN), XGBoost, Deep Learning, and Fuzzy Neural Networks, have been increasingly applied in the field of internal combustion engines [16]-[21]. These methods have demonstrated effectiveness in predicting fuel consumption, emissions, and engine performance through the adjustment of operating parameters and conditions. However, most existing studies primarily focus on optimizing engine operating conditions, while the application of Machine Learning to the optimization of the geometric designs of engine components remains limited.

Therefore, a significant research gap exists in the integration of CFD simulations and Machine Learning to efficiently explore complex geometric design spaces. To address this issue, this study proposes a data-driven approach for combustion chamber optimization. Specifically, the piston bowl geometry is parameterized using six design variables, forming a multidimensional design space. CFD simulations are conducted to generate a dataset representing in-cylinder flow characteristics, in which the tumble ratio is selected as the primary evaluation metric. Based on this dataset, a Machine Learning model is developed to learn the nonlinear relationships between geometric parameters and flow characteristics, enabling rapid prediction and optimization. This integrated approach not only reduces computational cost but also enhances the effectiveness of combustion chamber design, ultimately aiming to enhance fuel–air mixing and combustion performance.

## II. RESEARCH METHOD

### A. *Effect of the piston bowl on the air entrainment into the combustion chamber*

Understanding in-cylinder gas motion is crucial due to its significant influence on fuel–air mixing, flame propagation, and overall engine performance. The combustion chamber geometry, particularly the piston bowl, plays a dominant role in governing in-cylinder flow behavior. Among various flow characteristics, the tumble ratio is highly sensitive to the piston bowl geometry, as its shape directly controls the formation, intensity, and evolution of large-scale flow structures within the cylinder. Key flow characteristics commonly considered include turbulence intensity, swirl ratio, and tumble ratio, with particular emphasis on the tumble ratio in this study due to its strong dependence on piston bowl design [22].

#### 1) *Turbulence*

Flame propagation in non-turbulent mixtures is relatively slow but increases significantly with turbulence intensity due to enhanced mixing at the flame front, which accelerates chemical reactions by increasing the contact rate between reactants. Small-scale turbulence, characterized by numerous fine-scale vortices, is more effective in enhancing reaction rates and flame propagation than larger, less frequent flow structures [7], [23]. Various design modifications, including tumble flaps, shrouded or unshrouded valve configurations, and optimized piston bowl geometries, have been proposed as effective strategies to enhance turbulent flow characteristics within the cylinder [24], [25]. Computational Fluid Dynamics (CFD) simulations are widely employed to analyze such flow behaviors, in which turbulent kinetic energy is commonly used as a key parameter to quantify turbulence intensity.

#### 2) *Tumble*

As the piston nears the Top dead center (TDC) at the end of the compression stroke, the mixture within the engine experiences a radial inward or transverse motion known as squish. Tumble, a secondary rotational flow, occurs as a result of this squish motion when the piston is positioned close to TDC. Tumble is also described as rotational flow around a circumferential axis near the outer edge of the piston bowl, as depicted in Fig. 1 [24], [26].

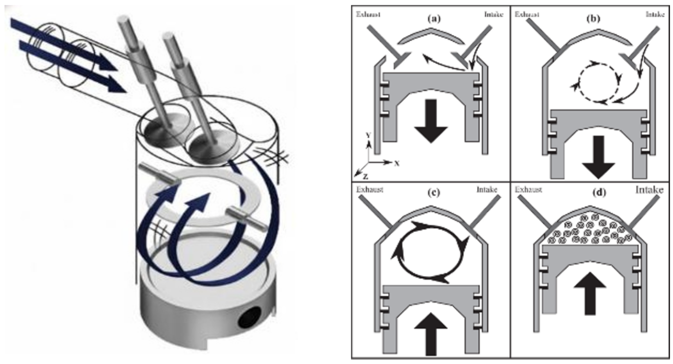


Figure 1. The formation process of Tumble flow [27], [28].

In ANSYS IC Engine, the tumble ratio is automatically computed under the appropriate simulation command. CFD simulation by ANSYS-ICE computed tumble ratio as.

$$T_R = \frac{L_{.ta}/I_{.ta}}{2\pi.N/60} \quad (1)$$

Where L.ta is the magnitude of fluid angular momentum for the tumble axis, and I.ta denotes the moment of inertia of fluid mass about the tumble axis [29]. In addition to the tumble ratio, ANSYS-ICE introduces another parameter called the cross-tumble ratio, which involves the computation of rotational flow around an axis perpendicular to the tumble axis, known as the cross-tumble axis.

### B. *Operation process*

High-fidelity CFD simulations were conducted using ANSYS ICE to evaluate 100 combustion chamber configurations and obtain the corresponding tumble ratio (TR) values. The resulting dataset was employed to develop a machine learning-based surrogate model within the RapidMiner platform, approximating the functional relationship Y=f(x), where Y denotes the dependent variable (TR), and x represents a multidimensional vector of geometric design variables. The surrogate model was subsequently treated as the objective function for optimization. A Random Forest algorithm was applied to efficiently explore the design space and identify the optimal parameter set that maximizes the tumble ratio. The overall workflow is illustrated in Fig. 2.

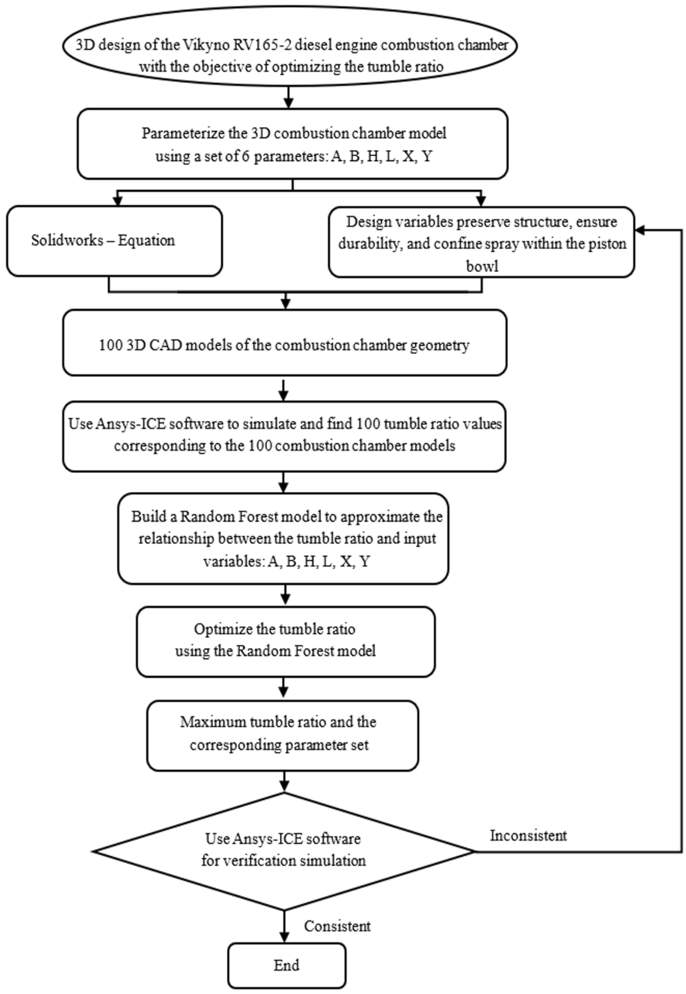


Figure 2. Integrated CFD–Machine learning workflow for Tumble ratio optimization.

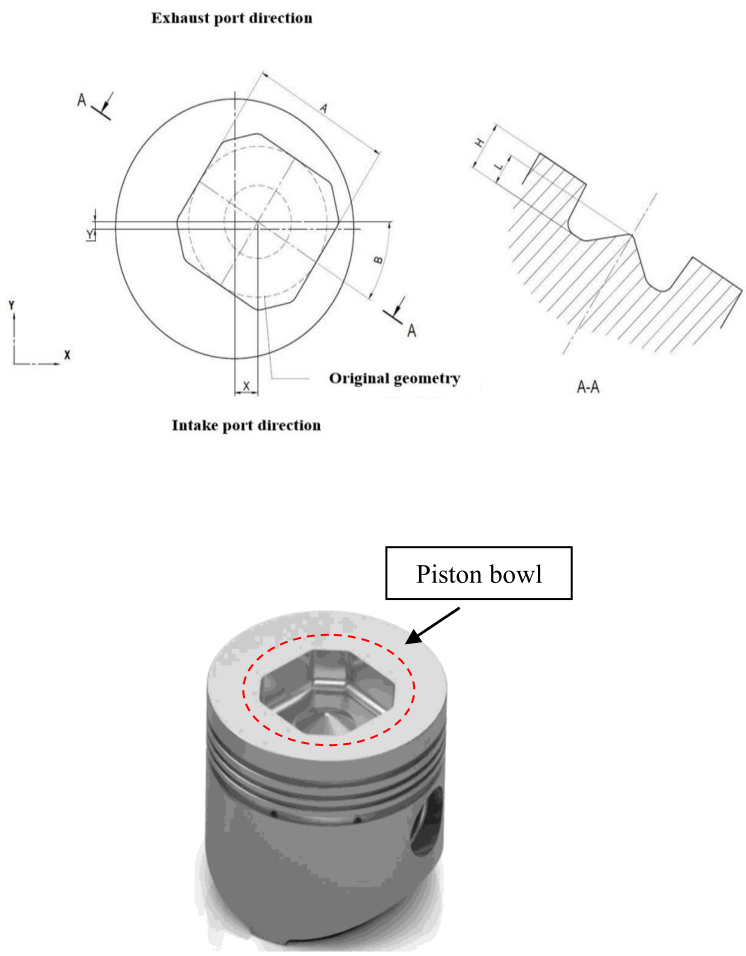


Figure 3. Parameterization of the combustion chamber geometry.

### C. *Parameterization of the 3D Geometric Profile of the Piston Bowl*

In this study, the geometric 3D profile of the piston bowl is defined by six variables: A, B, H, L, X, Y, as shown in Fig. 3 and detailed in Table 1. These parameter variables represent critical dimensions that significantly contribute to shaping the profile of this component.

TABLE I. COMBUSTION CHAMBER DESIGN PARAMETERS

| **No.** | **Parameter details** | | | | |
|---|---|---|---|---|---|
| | ***Parameter*** | ***Profile variables*** | ***Unit*** | ***Original value*** | ***Constraints*** |
| 1 | A | Combustion chamber width | mm | 61 | $50 \le A \le 60$ |
| 2 | B | Polygon rotation angle | degree | 0 | $0 \le B < 90$ |
| 3 | H | Combustion chamber depth | mm | 16 | $13{,}67 \le H \le 20{,}7$ |
| 4 | L | Cone height | mm | 7,5 | $5 \le L \le 9$ |
| 5 | X | Combustion chamber X-coordinate on the piston | mm | 4.5 | $-10 \le X \le 10$ |
| 6 | Y | Combustion chamber Y-coordinate on the piston | mm | 2.5 | $-10 \le Y \le 10$ |

The design variables were defined to generate new combustion chamber configurations under the following stringent constraints: the original combustion chamber structure must not be altered or compromised; structural integrity must be maintained under operating conditions; and the fuel spray must be fully confined within the piston bowl to ensure effective air–fuel mixing and to prevent wall impingement.

https://doi.org/10.1051/e3sconf/202672304002

*D. CFD simulation Procedure for Tumble ratio evaluation using ANSYS-ICE*

*1) Modeling of the engine*

Fig. 4 illustrates one of the 100 3D Piston bowl models utilized in this study. The specifications of the Vikyno RV165-2 engine, used for cold flow analysis, are detailed in Table 2.


Figure 4. Partitioned regions and their corresponding component labels in Ansys-ICE.

TABLE II. VIKYNO RV165-2 ENGINE SPECIFICATIONS

| No. | Engine technical details | | |
|---|---|---|---|
| | ***Specifications*** | ***Unit*** | ***Value*** |
| 1 | Connecting rod length | mm | 158 |
| 2 | Cylinder bore | mm | 105 |
| 3 | Crank radius | mm | 48.5 |
| 4 | Piston stroke | mm | 97 |
| 5 | Piston offset | mm | 0 |
| 6 | Compression ratio | - | 20:1 |
| 7 | Engine speed | rpm | 2200 |
| 8 | Minimum valve lift | mm | 0.2 |

The piston motion profile and valve lift characteristics are taken from the engine and serve as key inputs for the initial cold flow simulation.

*2) Decomposition and meshing*

To precisely depict the dynamics of the RV165-2 engine in a transient simulation, the engine model was partitioned into distinct zones. In ANSYS Meshing, each partitioned zone is denoted by a distinct label corresponding to its specific components, illustrated in Fig. 5.

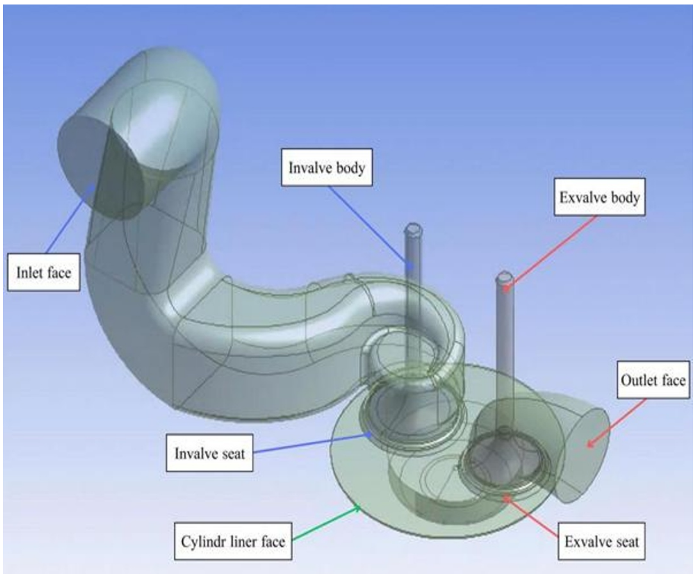


Figure 5. Partitioned regions and their corresponding component labels in Ansys-ICE.

The computational domain was discretized into high-quality meshes to ensure accurate CFD predictions. The mesh quality is confirmed by a low average skewness (0.2319), a high orthogonal quality (0.78035), and an acceptable average aspect ratio (3.7206). Although a small number of elements exhibit relatively high skewness and aspect ratio, their influence is limited. Overall, the mesh is considered suitable for stable and reliable simulations.

*3) Ansys-ICE Boundary conditions of the ANSYS-ICE simulation process*

The fundamental boundary conditions for the simulation process in ANSYS-ICE, designed to determine the tumble ratio value and other relevant parameters for each Piston bowl Model, are shown in Fig. 6.

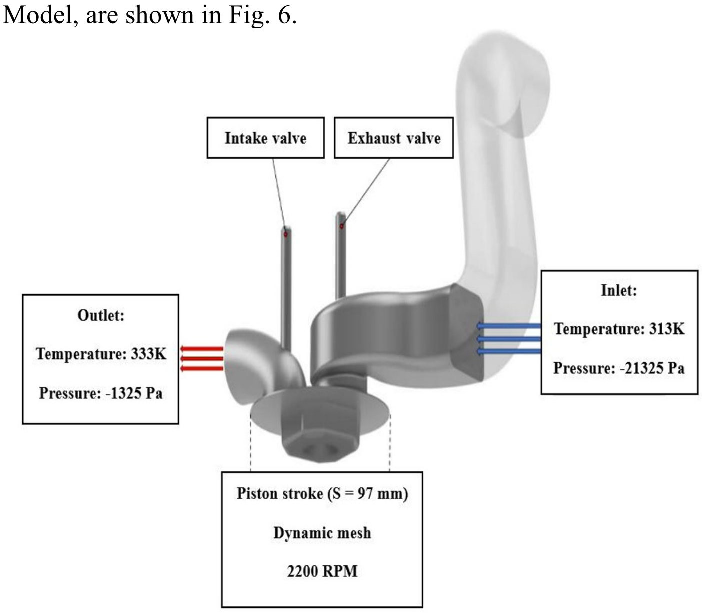


Figure 6. Boundary conditions simulated in Ansys-ICE.

The analysis in this study begins at a crank angle of 350 CAD, where the intake valve opening coincides with the exhaust valve closing. This phase continues until the end of combustion at 716 CAD. It is assumed that, with the influence of residual gases on in-cylinder conditions neglected, the exhaust valve remains fully closed from 532 CAD to 716 CAD.

*4) Convergence Assessment*

The convergence of the simulation model was evaluated through the following procedure. In the first timestep, convergence was achieved using the maximum number of iterations, resulting in an average residual on the order of $10^{-4}$ . The solution from this timestep was then used as the initial condition for the subsequent timestep. Additionally, the mesh was updated dynamically at every crank angle increment of 0.25 CAD, corresponding to the selected timestep size. The scaled residuals from the second timestep were monitored alongside those from the first timestep. Convergence was considered satisfactory when the average residuals decreased to the range of $10^{-5}$ to 10⁻6. Based on the CFD simulations performed using ANSYS ICE, the corresponding tumble ratio (TR) values were obtained for each combustion chamber configuration.

*E. Combustion chamber optimization of the VIKYNO RV165-2 engine using RapidMiner and Random Forest*

A machine learning workflow was developed in RapidMiner using a drag-and-drop interface to train a Random Forest regression model based on simulation data stored in Data.xlsx. The process includes data loading and role assignment, where the tumble ratio (TR) is defined as the target variable, while the remaining parameters serve as input features. The dataset is duplicated to enable both model training and subsequent validation. The Random Forest algorithm, configured with 100 trees and a maximum depth of 65, is employed to capture the nonlinear relationship between geometric parameters and TR. After training, the model is stored for reuse and simultaneously applied to the original dataset to generate predictions. This approach enhances model accuracy and generalization while enabling efficient evaluation of the predictive performance.

## III. RESULTS AND DISCUSSION

*A. In-cylinder Flow characteristics*

The CFD results indicate that the piston bowl geometry has a noticeable influence on the in-cylinder airflow during the intake and compression processes. As the piston approaches top dead center (TDC), the air motion is intensified due to the squish effect, contributing to the formation of tumble motion.

The velocity field distributions suggest that different piston bowl geometries lead to variations in airflow direction and vortex structures. In some configurations, the airflow appears to be more effectively redirected toward the central region of the combustion chamber, which may support the development of stronger tumble motion.

*B. Effect of geometric parameters on Tumble ratio*

A total of 100 piston bowl configurations were generated by varying six geometric parameters (A, B, H, L, X, and Y). The CFD results show that the tumble ratio varies significantly across the investigated design space, indicating a strong sensitivity of in-cylinder airflow to piston bowl geometry. This variation suggests that even relatively small changes in bowl dimensions can influence the direction, confinement, and redistribution of the airflow during the intake and compression processes.

Among the investigated parameters, X, L, and B are observed to appear more frequently in the decision rules derived from the machine learning model, suggesting their relatively stronger influence on the tumble ratio. These parameters are associated with key geometric features such as the bowl profile, edge location, and flow-guiding surfaces. Variations in these dimensions may affect how the incoming air is directed toward the piston bowl and how it is compressed near top dead center (TDC). As a result, they can contribute to either the enhancement or the weakening of the tumble vortex structure.

In addition, the results indicate that the effect of each parameter is not independent but depends on the interaction with other variables. A parameter that improves tumble motion under certain geometric conditions may become less effective when combined with different configurations. This highlights the nonlinear nature of the relationship between piston bowl geometry and tumble ratio, where simple one-factor analysis may not adequately capture the underlying flow behavior. Therefore, the use of machine learning provides a useful complementary approach for identifying optimal parameter combinations based on CFD-generated data.

*C. Optimization results*

A total of 100 piston bowl configurations were generated by varying six geometric parameters (A, B, H, L, X, and Y) and evaluated using CFD simulations. The results indicate that the tumble ratio varies significantly across the investigated design space, highlighting the strong sensitivity of in-cylinder airflow to piston bowl geometry. To support the analysis, a machine learning model based on a regression tree approach was developed to approximate the relationship between the input parameters and the tumble ratio. The model results show that parameters such as X, L, and B appear frequently in the decision structure, suggesting their relatively important role in influencing the airflow behavior. However, the effect of individual parameters is not independent, but rather depends on their interactions, indicating a nonlinear relationship between piston bowl geometry and tumble ratio.

Based on the model predictions, an optimized piston bowl configuration was identified with the parameter set A = 59 mm, B = 35 mm, H = 14.38 mm, L = 8 mm, X = 10 mm, and Y = 10 mm. CFD verification shows that this configuration achieves an increase of approximately 56.81% in the average tumble ratio compared to the baseline case. This improvement can be attributed to the combined effect of the optimized geometric parameters, which enhance airflow redirection and promote squish-induced vortex formation near top dead center (TDC). Furthermore, the selected configuration appears to facilitate a more coherent large-scale tumble structure while reducing flow dissipation, likely due to an appropriate balance between bowl depth and flow-guiding features.

As illustrated in Fig. 7, a comparison between the predicted and baseline tumble ratios clearly demonstrates the improvement achieved by the optimized configuration. These findings suggest that the simultaneous optimization of multiple geometric parameters is essential for effectively enhancing tumble motion within the cylinder.

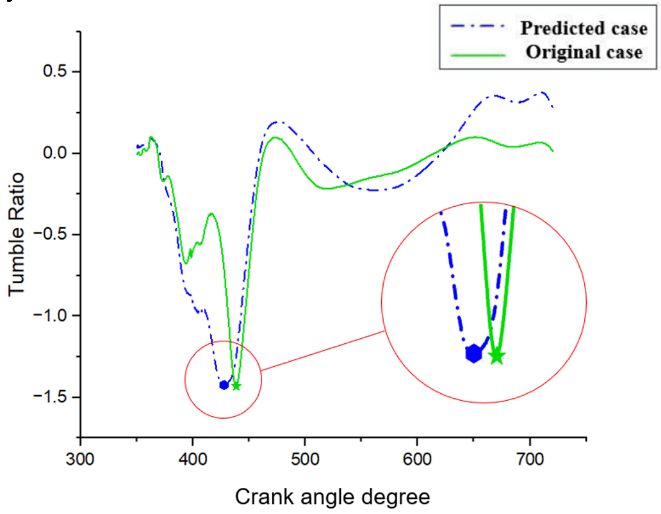


Figure 7. ICE Comparison of predicted and original Tumble results.

### D. Discussion

The results obtained in this study are generally consistent with previous findings confirming that combustion chamber geometry plays a significant role in influencing in-cylinder airflow characteristics [13], [22]. The integration of CFD simulations and machine learning provides a practical and efficient approach for exploring the design space. Compared to conventional trial-and-error methods, this combined approach enables a more systematic and comprehensive evaluation of multiple geometric configurations.

However, it should be noted that the present study primarily focuses on cold flow analysis. Further investigations are required to assess the impact of the optimized piston bowl geometry on combustion performance and emission characteristics. In addition, the results obtained from the simulation and modeling approach should be validated through experimental studies to ensure their reliability under real operating conditions. In particular, the effectiveness of the optimized configuration may be influenced by the actual fuel injection characteristics, including spray targeting and injector positioning. Therefore, future work should consider the interaction between piston bowl geometry and fuel injection parameters to provide a more comprehensive evaluation.

## IV. CONCLUSION

This study investigated the influence of piston bowl geometry on tumble motion in a diesel engine using a combination of CFD simulations and machine learning techniques. The results show that variations in geometric parameters lead to noticeable changes in the tumble ratio, indicating a strong dependency on piston bowl design. A machine learning model was used to approximate the relationship between the design variables and the tumble ratio, providing useful insights into the parameter influence. An optimized configuration was identified, which achieved an increase of approximately 56.81% in tumble ratio compared to the baseline case. This result suggests that the proposed approach can be used as a reference for improving in-cylinder flow characteristics.

## REFERENCES

[1] J. Kelly, K. Chen, V. Tattari, E. Uusivuori, and H. Ekberg, "Health impacts of diesel vehicles suspected of using prohibited defeat devices in the EU and UK," 2025.

[2] B. Wang, B. Wang, B. Lv, and R. Wang, "Impact of Motor Vehicle Exhaust on the Air Quality of an Urban City," Aerosol Air Qual. Res., vol. 22, no. 8, Aug. 2022, doi: 10.4209/aaqr.220213.

[3] W. Anggono et al., "Effects of extraction methods on the fuel characteristics and diesel engine performances of jatropha curcas biodiesel," J. Teknol., vol. 84, no. 4, pp. 29–39, Jul. 2022, doi: 10.11113/jurnalteknologiv 84.15079.

[4] P. V. Rao, S. Venkateswarulu, S. Venkataiah, G. Narendar, J. Pavani, and G. I. Mahith Simha Reddy, "Evaluating the performance of a diesel engine with a diesel-plastic fuel blend as an alternate fuel," in Journal of Physics: Conference Series, Institute of Physics, 2025. doi: 10.1088/1742-6596/3066/1/012013.

[5] G. J. Micklow and W. D. Gong, "Intake and in-cylinder flowfield modelling of a four-valve diesel engine," Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering, vol. 221, no. 11, pp. 1425–1440, 2007, doi: 10.1243/09544070JAUTO504.

[6] N. H. Anand, A. A. Hairuddin, K. A. Md Rezali, M. S. Mohd Alias, and A. N. Nizarimi, "CFD based parametric study of swirl intake manifold geometry and its effect on airflow characteristics on CI engines," Engineering Research Express, vol. 7, no. 3, Sep. 2025, doi: 10.1088/2631-8695/ae0000.

[7] S. G. Choe et al., "Effect of the geometrical shapes of the helical-spiral shroud intake valve on swirl generation in cylinder of diesel engine," Results in Engineering, vol. 18, Jun. 2023, doi: 10.1016/j.rineng.2023.101132.

[8] T. Wang, D. Liu, G. Wang, B. Tan, and Z. Peng, "Effects of variable valve lift on in-cylinder air motion," Energies (Basel), vol. 8, no. 12, pp. 13778–13795, 2015, doi: 10.3390/en81212397.

[9] S. H. Abidi and M. M. Hasan, "Variation of turbulent kinetic energy due to dimples in intake manifold for diesel engine," Case Studies in Thermal Engineering, vol. 27, Oct. 2021, doi: 10.1016/j.csite.2021.101299.

[10] F. Liu, B. Liu, J. Zhang, P. Wan, and B. Li, "Study on a Novel Variable Valve Timing and Lift Mechanism for a Miller Cycle Diesel Engine," Energies (Basel), vol. 15, no. 22 Nov. 2022, doi: 10.3390/en15228521.

[11] Z. Tong, H. Wu, Z. Shi, and L. Myagkov, "A review on the influence of in-cylinder turbulence type on combustion characteristics of high power-density diesel engine," Oct. 01, 2025, KeAi Communications Co. doi: 10.1016/j.jtte.2024.09.008.

[12] T. Wang et al., "Influence of In-Cylinder Turbulence Kinetic Energy on the Mixing Uniformity within Gaseous-Fuel Engines under Various Intake Pressure Conditions," Energies, vol. 17, no. 13, Jul. 2024, doi: 10.3390/en17133321.

[13] *T. Q. Nguyen and A. Y. Dunin, "Investigation into the Impact of Piston Bowl Size on Diesel Engine Characteristics with Changes in Fuel Injection Pressure and Boost Pressure," Applied Sciences (Switzerland), vol. 14, no. 10, May 2024, doi: 10.3390/app14104334.*

[14] *R. Kumar, S. Nath, Atul, V. Ranjan, S. Khan, and S. K. Ghoshal, "Influence of Piston Bowl Geometry and Hydrogen–Diesel Mixing Ratios on Dual-Fuel Engine Combustion," ACS Omega, Feb. 2026, doi: 10.1021/acsomega.5c09077.*

[15] *N. K. F. Al-Abboodi, "The Combined Effect of the Piston Bowl Geometry and Injection Fuel Pressure on the Compression Ignition Engine Characteristics," CFD Letters, vol. 14, no. 8, pp. 43–62, Aug. 2022, doi: 10.37934/cfdl.14.8.4362.*

[16] *M. B. Sanjeevannavar et al., "Machine Learning Prediction and Optimization of Performance and Emissions Characteristics of IC Engine," Sustainability (Switzerland), vol. 15, no. 18, Sep. 2023, doi: 10.3390/su151813825.*

[17] *M. Nachippan et al., "Machine learning predictions for enhancing engine performance and emission using aluminum oxide nano additives in castor biodiesel," Sci. Rep., vol. 15, no. 1, Dec. 2025, doi: 10.1038/s41598-025-02388-y.*

[18] *M. Zbikowski and A. Teodorczyk, "Machine Learning for Internal Combustion Engine Optimization with Hydrogen-Blended Fuels: A Literature Review," Mar. 01, 2025, Multidisciplinary Digital Publishing Institute (MDPI). doi: 10.3390/en18061391.*

[19] *Ü. Ağbulut, A. E. Gürel, and S. Sarıdemir, "Experimental investigation and prediction of performance and emission responses of a CI engine fuelled with different metal-oxide based nanoparticles–diesel blends using different machine learning algorithms," Energy, vol. 215, Jan. 2021, doi: 10.1016/j.energy.2020.119076.*

[20] *Y. Yu, Y. Wang, J. Li, M. Fu, A. N. Shah, and C. He, "A novel deep learning approach to predict the instantaneous NOx emissions from diesel engine," IEEE Access, vol. 9, pp. 11002–11013, 2021, doi: 10.1109/ACCESS.2021.3050165.*

[21] *M. Yue, L. Li, R. Zou, T. Su, N. Wang, and H. Wen, "Optimization of Combustion Chamber Structure in a Rotary Engine Based on a Fuzzy Neural Network and a Genetic Algorithm," Processes, vol. 13, no. 1, Jan. 2025, doi: 10.3390/pr13010122.*

[22] *V. H. Lê, "Nghiên Cứu Cải Tiến Đường Nạp Động Cơ Diesel Một Xilanh 16,5 Hp Sử Dụng Trong Nông – Lâm – Ngư Nghiệp", Trường Đại Học Sư Phạm Kỹ Thuật Thành Phố Hồ Chí Minh, Tp. Hồ Chí Minh, 2019.*

[23] *Zhu, Y., Zhao, H., Melas, D. A., & Ladommatos, N. (2004). Computational study of the effects of the re-entrant lip shape and toroidal radii of piston bowl on a HSDI diesel engine's performance and emissions (No. 2004-01-0118). SAE Technical Paper.*

[24] *Heywood, J.B. (1988). Internal combustion engine fundamentals (1st ed.). United States: McGraw-Hill, Inc.*

[25] *Shafie, N. M., & Said, M. M. (2017). Cold flow analysis on an internal combustion engine with different piston bowl configurations. Journal of Engineering Science and Technology, 12(4), 1048-1066.*

[26] *Han, S.B.; Chung, Y.J.; and Lee, S. (1995). Effect of engine variables on the turbulent flow of a spark ignition engine. Journal of Mechanical Science and Technology, 9(4), 492-501*

[27] *FEV, "Flow Test Bench." [Online]. Available: https://www.fev-sts.com/fileadmin/user_upload/Flow_Test_Bench-EN-DE.pdf*

[28] *P. Olmeda, J. Martín, R. Novella, và R. Carreño, "An adapted heat transfer model for engines with tumble motion", Appl Energy, vol 158, tr 190–202, 2015, doi: https://doi.org/10.1016/j.apenergy.2015.08.051.*

[29] *ANSYS, Inc. (2013). Internal combustion engine in the workbench. Retrieved March 10, 2015, from* http://www.ansys.com