Computational Sciences and Engineering 5(1) (2025) 25-38

University of Guilan

Computational Sciences and Engineering

journal homepage: https://cse.guilan.ac.ir/

Computational Fluid Dynamics Study of Combustion Chamber Geometry Effects on In-Cylinder Flow and Emissions Formation in a Diesel Engine

Ali Mohammad Taheri [a], Bahram Jafari [a], Morteza Fathi [b,*]

[a] Faculty of Engineering Modern Technologies, Amol University of Special Modern Technologies (AUSMT), Amol, Iran
[b] Department of Mechanical Engineering, Technical and Vocational University (TVU), Tehran, Iran

ARTICLE INFO

Article history:
Received 10 August 2025
Received in revised form 19 August 2025
Accepted 27 August 2025
Available online 27 August 2025

Keywords:
Diesel Engine
Chamber Geometry
In-Cylinder Flow
CFD
Emissions

ABSTRACT

This study examines how piston-bowl geometry influences in-cylinder flow and pollutant formation in a large diesel engine. High-fidelity simulations were performed for the baseline combustion chamber and three modified bowl designs (A, B, and C). The model replicates experimental peak pressure within 3%, soot levels with ≤1% error, and NOx emissions within ±3%. The modified bowls increase in-cylinder turbulence and induce stronger squish flows, leading to longer combustion duration but more uniform mixing. As a result, peak cylinder pressures are slightly lower in the re-designed bowls than in the baseline, and the onset of combustion is delayed. Notably, the most highly squish-inducing chamber (A) produced higher peak temperatures but also exhibited the lowest soot emissions, consistent with enhanced mixing. Across the modified chambers, indicated work and cycle efficiency increased relative to the baseline (due to reduced negative work in compression). Emissions of NOx and soot showed opposing trends: chamber A (highest turbulence) generated more NOx (owing to its higher local temperatures) but significantly less soot (owing to more complete combustion), whereas the baseline chamber had higher soot due to local fuel-rich pockets. These results indicate that combustion chamber shape can be tuned to improve mixing and efficiency, at the cost of shifting the NOx–soot trade-off. However, combustion chamber B, simultaneously improves power output by 10%, NOx emissions by 41 %, and soot emissions by 33%.

1. Introduction

High-compression diesel designs are favored for their superior efficiency and reliability. In transport and power generation, these large diesel engines are valued for their ability to convert fuel energy into mechanical power with high thermal efficiency. However, they also emit

* Corresponding author.
E-mail addresses: m-fathi@tvu.ac.ir; morteza.fathi@yahoo.com (M. Fathi)

https://doi.org/10.22124/cse.2025.31368.1116

significant pollutants, which are increasingly regulated by international standards [1]. To meet strict emission limits, various strategies are employed: fuel treatment, advanced injection control, exhaust gas recirculation (EGR), after-treatment, and combustion chamber optimization. Unlike fuel injection parameters that directly control temporal aspects of combustion, geometric modifications alter spatial fuel-air distribution through complex interactions between swirl, squish, and turbulence generation mechanisms. A persistent challenge involves reconciling conflicting requirements: enhancing air utilization for soot reduction while avoiding thermal conditions that promote nitrogen oxide formation [2].

Diesel engines rely on in-cylinder air motion to achieve efficient combustion. In particular, the piston bowl (combustion chamber) shape strongly influences in-cylinder flow (swirl and squish motions) and turbulence, which in turn governs fuel–air mixing, ignition, and pollutant formation [3]. This geometric optimization problem has gained renewed urgency with the advent of carbon-neutral fuels exhibiting distinct combustion behaviors, necessitating chamber redesigns for optimal performance [4]. Classic studies demonstrate that modifying bowl geometry can dramatically alter swirl and turbulence levels [5, 6]. For example, in [5], computational fluid dynamics (CFD) is used to show that a center-bowl on a flat piston delivered the highest tumble ratio and turbulent kinetic energy, indicating very strong in-cylinder motion. These internal flow patterns (swirl, squish, turbulence) are key to mixing quality. Better mixing reduces local fuel–air inhomogeneities, shortening ignition delay and improving combustion completeness [7]. Conversely, poor mixing leaves fuel pockets that burn in diffusion flames, increasing soot. Consequently, chamber design directly affects combustion efficiency and emissions, and has become a vital topic in diesel research [7].

Early diesel combustion chambers evolved from open bowls to re-entrant designs that enhance radial momentum preservation. Seminal work by Dent and Derham [8] first quantified squish velocity profiles, demonstrating how peripheral flow accelerates toward bowl lips during compression. The transition to direct injection systems in the 1990s prompted systematic studies of bowl aspect ratios, with Miles and co-authors [9] revealing that shallow bowls increase spray-wall impingement while deep bowls enhance swirl decay. Computational breakthroughs by Gonzalez and co-workers [10] enabled three-dimensional (3D) analysis of vortex-driven combustion, linking bowl curvature to turbulent kinetic energy distribution. Early work by Saito and co-workers experimentally compared conventional versus re-entrant bowls and found re-entrant (toroidal) bowls significantly intensified near-top dead center (TDC) swirl and turbulence, enhancing combustion and dramatically reducing soot [11]. Numerical studies have since confirmed these trends. In [12], CFD is used to evaluate a high-swirl re-entrant bowl and found it raised swirl and turbulent kinetic energy while simultaneously lowering NOx and soot emissions. Similarly, Abdul Gafoor and Gupta [13] numerically varied bowl shape and swirl ratio, reporting that bowls inducing stronger swirl generally gave better mixing and lower soot, albeit with some NOx penalty at high swirl. Comparison of Omega versus toroidal bowl profiles showed that a toroidal bowl gave higher turbulence intensity and combustion efficiency, reducing soot [6]. These studies collectively highlight that even modest bowl re-entrance or lip shaping can greatly enhance mixing and alter the NOx–soot tradeoff [12]. Yan and co-workers [14] examined three modern chamber shapes (nebula, cross, re-entrant) in a stoichiometric natural-gas engine and found that the re-entrant design achieved the highest pre-TDC turbulence intensity and most symmetric flame spread. Their nebula chamber suffered from asymmetric turbulence and delayed flame

development, leading to longer burn duration, whereas the re-entrant shape sustained a larger flame surface (and thus faster combustion) after TDC. This observation suggests that increased turbulence before ignition (as generated by re-entrant bowls) can improve combustion completeness and efficiency. Pham and co-authors [1] specifically examined heavy-duty marine diesel engines and reported that omega-type and re-entrant bowls increased indicated work and efficiency while significantly lowering NO emissions (by 25–46%) compared to a baseline U-type bowl. They attribute the NOx reduction to lower peak combustion temperatures in those designs. Interestingly, their modified bowls had negligible effect on soot in their case, implying complete mixing (and possibly offset by lower flame temperatures).

Bowl shape chiefly governs two in-cylinder motions: swirl, which rotates flow around the axis, and squish, which radially expels gases as the piston nears TDC. The momentum conservation principles governing in-cylinder flows create intrinsic trade-offs between swirl and squish energies [15]. Intake port design often sets the baseline swirl ratio, but the piston bowl lip and shape can amplify or damp swirl during compression. Raj and co-authors [5] showed that bowl geometry profoundly changed swirl and tumble ratios: for one case, a central flat bowl produced the highest tumble ratio and turbulence. Likewise, re-entrant bowls sustain stronger swirl and turbulence near TDC than shallow or hemispherical bowls [12]. A review of various bowl modifications showed that designs promoting more piston thrust flows (squish jets) and swirl led to higher overall turbulence, improving mixing [16].

Squish is particularly influenced by the bowl's throat diameter and lip height. A narrow throat or raised lip causes gas to squirt outwards as the piston closes, generating a high-speed squish jet. Li and co-workers [7] specifically simulated biodiesel combustion in Omega, shallow, and hemispherical bowls and found that the Omega-shaped bowl (with a narrow throat) created intense squish flows at high speed, markedly enhancing air–fuel mixing. Their CFD results showed the strong squish flow shortened ignition delay and favored premixed burning at high speed, even though it slightly increased NOx under some conditions. In general, strong squish flows promote better fuel atomization on the piston crown and accelerate burn, whereas inadequate squish leads to stratification and soot. Many authors report that bowls engineered for stronger squish yield lower soot concentrations and improved soot–NOx tradeoff [7].

The flow fields set by bowl geometry directly control combustion phasing and pollutant formation. Better air–fuel mixing generally shortens ignition delay and produces more uniform combustion. For instance, it is observed that the bowl shape with strongest turbulence also had the shortest ignition delay and fastest burn rate among the cases tested [5]. Similarly, computational studies show re-entrant bowls can accelerate initial combustion. It is found that the re-entrant bowl's higher velocities and turbulence increased mixing to the point that soot emissions dropped sharply [11]. In biodiesel simulations, Li and co-workers [7] noted that intense squish from a narrow Omega bowl enhanced mixing so much at high engine speeds that the engine actually needed advanced ignition timing to avoid knocking, implying a much faster burn, albeit with slightly higher peak temperature and NOx.

NOx and soot emissions are particularly sensitive to geometry-induced mixing. Better mixing reduces fuel-rich pockets but usually raises local flame temperatures. For example, it is reported that a deeper re-entrant bowl improved swirl and turbulence, which cut soot formation by enhancing fuel oxidation, but it also slightly increased NOx due to higher combustion efficiency

[12]. In [6], a toroidal bowl (which gave maximum turbulence) produced the cleanest burn, where soot levels fell substantially, but the cleaner combustion also elevated NOx unless EGR or timing were adjusted. Conversely, shallow, open bowls tend to have lower peak temperatures and NOx but suffer from incomplete mixing (more soot) [7, 11].

Kidoguchi and co-workers [17] observed that very high squish combined with a small throat diameter reduced both NOx and particulates, indicating an optimal geometry could improve both, but such gains usually require offsetting other factors. In practice, most studies find that bowls which strengthen mixing lower soot at the expense of moderate NOx rise [7, 11]. Doppalapudi and co-authors [18] reviewed chamber modifications and concluded that increased in-cylinder turbulence consistently boosts combustion quality and reduces HC/CO/soot emissions, while NOx often becomes harder to reduce without external measures.

Most investigations of chamber geometry use 3D CFD to resolve the complex flows and spray combustion [3, 19-21]. Traditional studies have used RANS-based CFD codes (e.g. KIVA, STAR-CD) with standard turbulence models (k-ε, k-ζ-f) to capture mean swirl and squish effects. Such studies (e.g., [5, 12, 13]) have been instrumental in linking bowl shape to flow metrics. They typically model injection with Lagrangian parcels and breakup (KH–RT models) and solve combustion chemistry with progress-variable or flamelet models. More recent work has begun employing high-fidelity large-eddy simulation (LES) to better capture the unsteady squish vortex and cycle-to-cycle variability [6]. For example, Xu and co-workers [22] used a second-order CFD solver to study a gasoline compression ignition (GCI) engine and highlighted that resolving transient flows is critical when co-optimizing bowl geometry with injection strategy. However, LES requires extremely fine computational grids.

While many prior studies examined one or two bowl shapes, often with varying injection parameters, this work systematically compares four bowls (one baseline plus three redesigned variants) under identical injection and intake conditions. By holding the injection profile and swirl ratio constant, the influence of the piston-bowl geometry on in-cylinder flow, combustion dynamics, and pollutant formation (NOx and soot) are isolated. Earlier works combined geometry changes with injection or EGR adjustments [6, 23, 24]. In contrast, this study maintains identical injection timing and operating conditions so that only geometry differences drive the results. Moreover, high-fidelity 3D CFD is employed to capture subtle flow-geometry interactions. This allows any performance or emission differences to be directly attributed to the geometry changes. Therefore, this study advances the literature by quantifying how incremental bowl modifications (denoted A, B, C) affect pressure, temperature, turbulent kinetic energy (TKE), equivalence ratio distribution, and emission indices.

2. Methodology

A heavy-duty, four-stroke marine diesel engine is modeled. Its key specifications are presented in ***Table 1***. The engine operates at steady full load and high speed; all cases assume identical operating conditions. Ambient pressure and temperature are 1 bar and 298.15 K, respectively. Injection timing and fuel quantity are fixed at the baseline values, and the intake air humidity is held constant at the nominal level. Only the piston bowl geometry is varied.

Table 1. Engine Specifications

Parameter	Value
Engine Type	Water-cooled Turbo-Charged Heavy-Duty Diesel Engine
Number of Cylinders	12
Bore (mm)	150
Stroke (mm)	180
Engine Displacement (L)	38.1
Compression ratio (-)	15:1
Fuel Injection System	Common Rail Direct Injection
Rated Power (kW)	1000
Rated Speed (RPM)	1500

The combustion chamber geometries studied are one baseline and three modifications (labeled A, B, and C). In Shape A, the piston crown has a deep re-entrant (Omega-type) profile; Shape B has a moderate re-entrant bowl; Shape C has a stepped shallow bowl with an inner pocket. The baseline chamber is a conventional shallow cylinder with a broad throat. ***Figure 1*** illustrates these geometries (viewed at TDC): the original (Base) and the three new designs (A, B, C). The modified bowls create stronger squish flows than the base, as illustrated by their deeper re-entrant profiles. The in-cylinder volume versus crank angle degrees (CAD) is nearly identical for all cases, as the bore, stroke, and clearance are unchanged. The key differences arise in the flow and combustion.

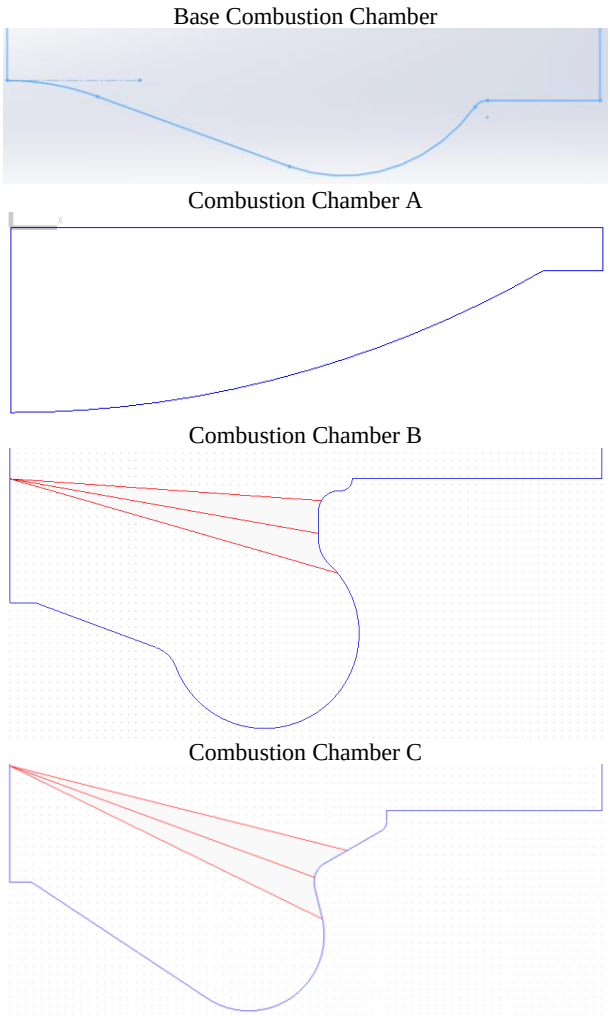

Figure 1. Geometries of the investigated combustion chambers

Computational simulations were carried out in two stages. First, a one-dimensional (1D) cycle simulation (AVL BOOST) was used to generate overall cylinder pressure, temperature, and mass flows for the baseline engine cycle. The 1D model accounts for intake/exhaust piping and heat transfer, and it provides boundary conditions and validated initial guesses for the CFD. Such 1D engine modeling is standard practice and helps accelerate the setup (especially since the engine is large) [1]. Next, a full 3D CFD simulation of a single cylinder was performed using the AVL FIRE code. To perform 3D simulation in AVL-FIRE software, the geometric model of the combustion chamber is created. After preparing the geometric model, dynamic meshing is performed in AVL FIRE software. The model's dynamic meshing is executed in the FAME ENGINE Plus environment. The 3D mesh is a moving mesh with adaptive remeshing of the piston bowl region. In the present study, the three-equation k-ζ-f model [25, 26] is used for turbulence modeling in flow simulations. This model is widely applied in internal combustion engine flow simulations [27-32]. Diesel combustion is simulated using Extended Coherent Flame Model (ECFM)-3Z [33-38]. NOx is modeled with an extended Zeldovich mechanism, and soot with a Turbulent Flame Closure sub-model. The same models and numerical settings (time step, mesh size, convergence criteria) are used for all chamber shapes. For each geometry, the model is run over several engine cycles until cyclic stability is reached; results from the final cycle are used for analysis.

It is assumed that fuel properties, injection pressure, and spray pattern remain fixed. Intake air mass flow and temperature at the start of compression are identical for all cases. All simulations include the effects of spray evaporation, ignition delay (based on the selected diesel model), and heat losses to the cylinder walls. The numerical approach follows established practices in engine CFD; for example, others have demonstrated that combined 1D–3D simulations can effectively capture diesel combustion and emissions under varied conditions [1].

Figure 2 and ***Table 2*** serve as a comprehensive validation of the model's predictive accuracy across geometry variations: pressure, NOx, and soot. The model replicates experimental peak pressure within 3%, soot levels with ≤1% error, and NOx emissions within ±3%. This performance aligns with findings from the combined-GCI engine validation where accuracy of pressure and emissions predictions also remained less than 6% using a similar 1D–3D CFD coupling and ECFM-3Z framework [39]. Achieving this level of correspondence affirms the model's capability to differentiate subtle effects introduced by geometric modifications. Therefore, this study compares four chamber designs in an otherwise fixed-engine, using validated combustion CFD to extract in-cylinder pressures, temperatures, turbulence energy, mixture distributions, and pollutant formation for each geometry.

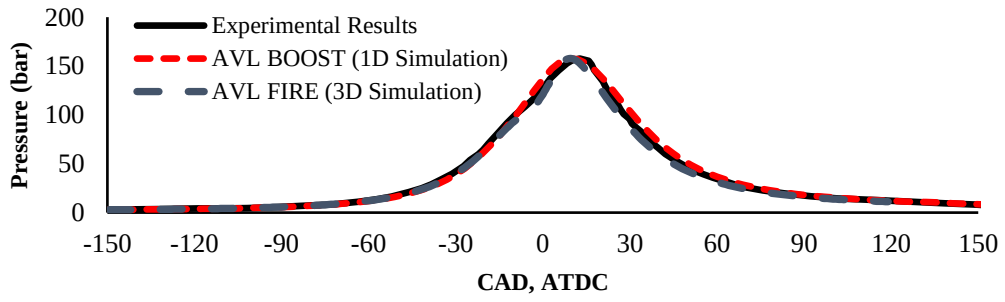

Figure 2. Comparison between experimental and numerical (1D/3D) modeling results of engine cylinder pressures

Table 2. Experimental vs. 3D simulated NOx and soot emissions comparison in the engine

Parameter	Experimental (ppm)	3D Simulation (ppm)	Relative Error (%)
Soot	12.35	12.40	+0.40%
NOx	4122.00	4008.00	−2.77%

3. Results and Discussion

Figure 3 and ***Figure 4*** show the time histories of cylinder pressure and mean gas temperature versus crank angle for each chamber shape. In the modified chambers A, B, and C, the peak cylinder pressure is observed to be somewhat lower than in the baseline case. This is mainly because the designs A–C increase turbulence and mixing, which delays the start of combustion and spreads it out. Among the designs, Chamber A (the deepest omega-shape) has the largest delay, so it shows the broadest pressure trace. Conversely, the baseline chamber (shallow bowl) ignites faster and produces a sharper pressure rise shortly after TDC. Despite the lower pressures, Chamber A produces the highest peak combustion temperature among all cases. Its intensified mixing and earlier fuel-wall impingement (due to the tall bowl walls) cause a more energetic premixed burn phase, raising the peak temperature. Chambers B and C yield peak temperatures between the baseline and A. These behaviors align with prior findings that re-entrant bowls increase heat release rates in the premixed phase, but reduce peak pressure due to faster expansion [11].

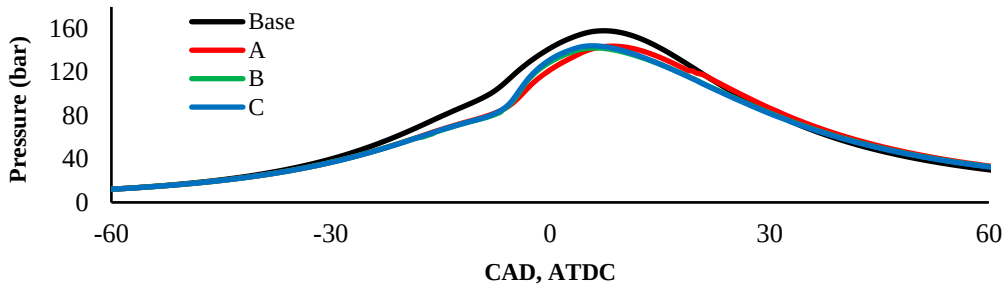

Figure. 3. Cylinder pressure vs. crank angle for the investigated combustion chambers

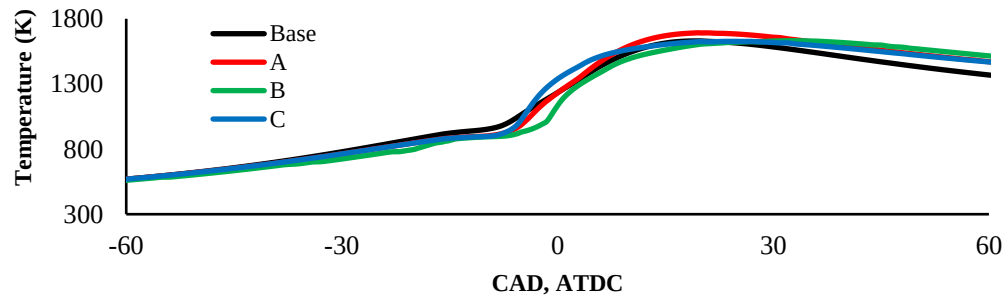

Figure 4. Mean cylinder temperature vs. crank angle for the investigated combustion chambers

Figure 5 shows a snapshot of the 3D temperature field at TDC for each design. In the baseline chamber, regions of higher temperature are more localized near the spray core, indicating stratification. In contrast, the modified chambers (especially A and B) show a more uniform

temperature distribution around the bowl. This reflects their higher turbulence level, which homogenizes the mixture by ignition. Likewise, ***Figure 6*** shows the equivalence ratio distribution at TDC. The baseline case has a larger volume of fuel-rich mixture ($\varphi > 1$) near the spray axis, whereas the modified shapes have narrower rich cores and more lean regions surrounding them. In other words, A–C produce a more uniform φ-distribution at ignition, due to enhanced mixing. This is consistent with the literature: higher-turbulence chambers tend to produce more uniform fuel–air mixing, reducing local fuel-rich pockets [1, 11].

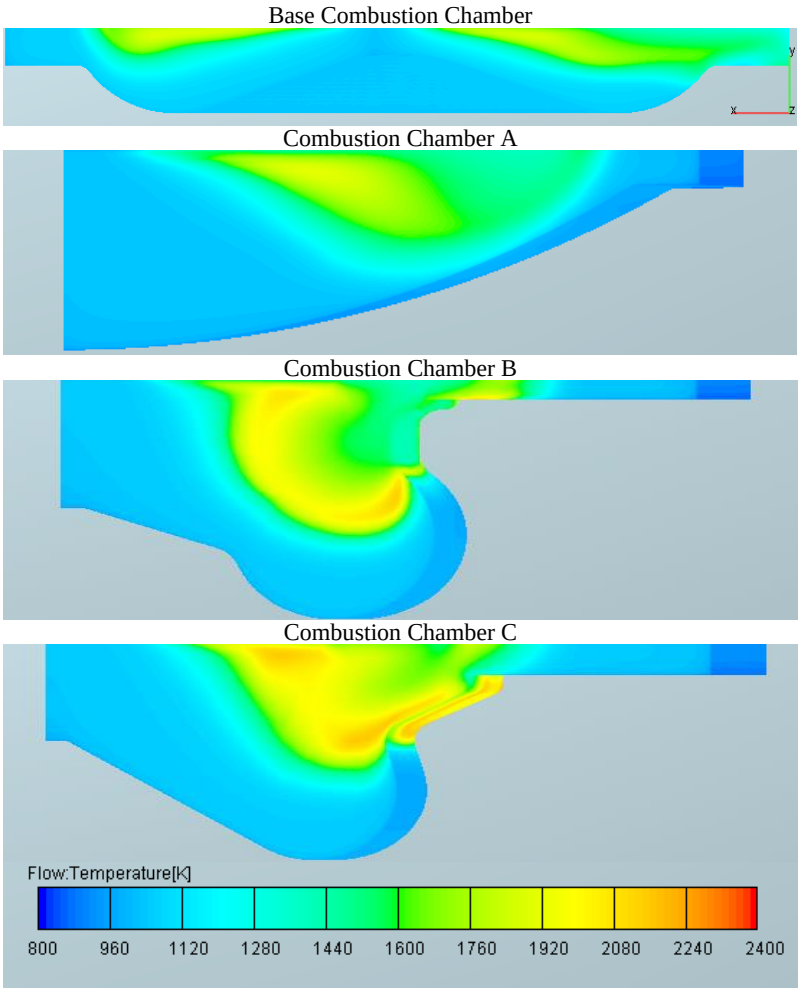

Figure 5. Spatial distribution of temperature across various combustion chamber geometries at TDC position

Figure 7 plots the time history of the mean TKE in the cylinder. All modified bowls (A, B, C) achieve significantly higher TKE prior to ignition than the baseline. Chamber A shows the highest TKE peak just before combustion. This confirms that the geometric changes substantially intensify in-cylinder turbulence and swirl. The stronger turbulence accelerates fuel evaporation and mixing, which in turn lengthens the burning duration but also suppresses soot.

Figure 8 shows the net heat-release rate (HRR) vs CAD. The baseline case has a sharp HRR peak near TDC, reflecting a rapid premixed burn. In contrast, shapes A–C have broader, lower-magnitude HRR peaks that begin later. The enhanced turbulence in A–C causes more combustion to occur in the diffusion phase over a longer period. As a consequence, the maximum pressure rise rate is smaller. These results are consistent with the findings of Saito and co-authors [11], who observed that chambers with higher turbulence (e.g. re-entrant bowls) exhibit reduced peak HRR and pressure-rise rate.

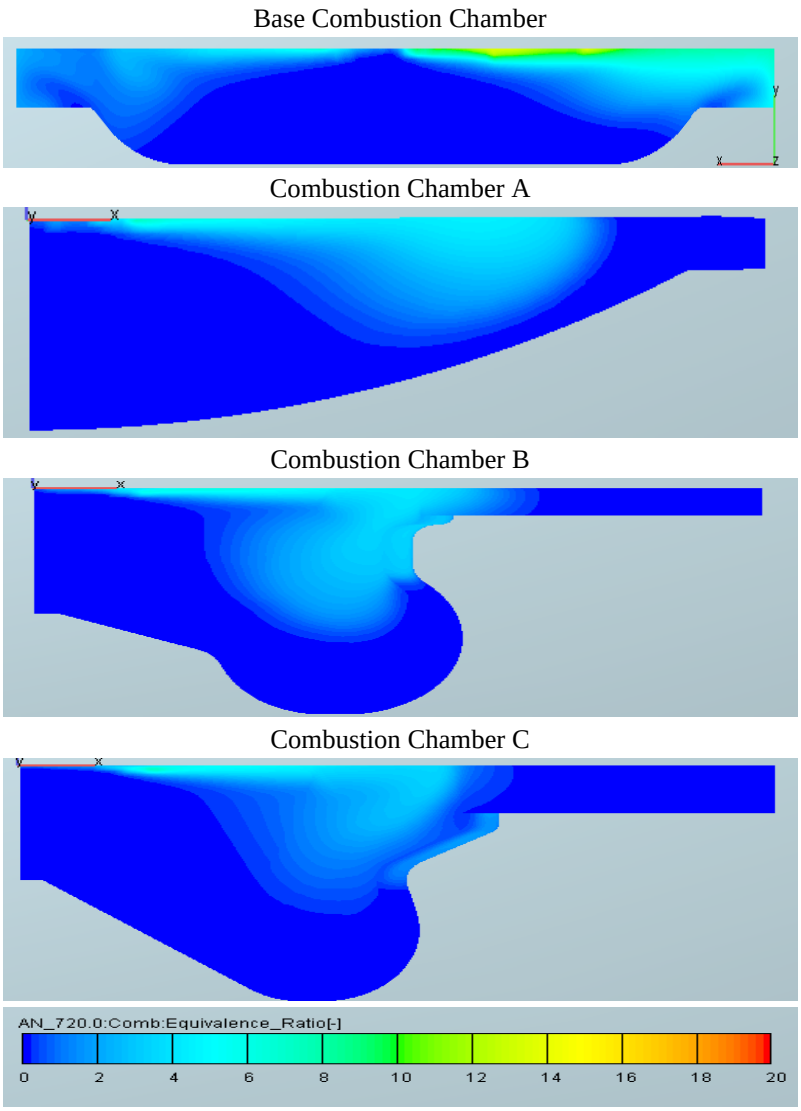

Figure 6. Spatial distribution of equivalence ratio across various combustion chamber geometries at TDC position

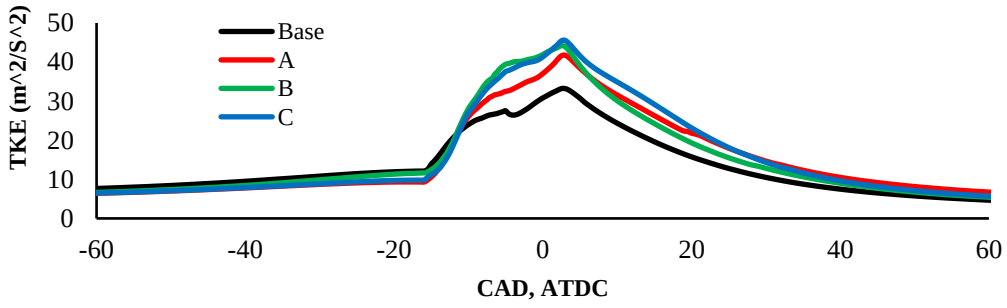

Figure 7. Mean turbulence kinetic energy vs. crank angle for the investigated combustion chambers

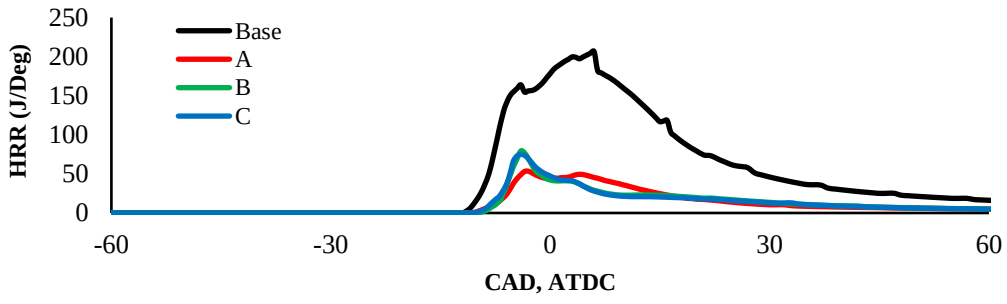

Figure 8. Heat release rate vs. crank angle for the investigated combustion chambers

Figure 9 summarize performance changes in terms of indicated work. All modified chambers yield higher indicated work and efficiency than the baseline. The reason is that delaying ignition (via enhanced turbulence) reduces the negative work done on the gas during compression, as observed in similar studies [1]. In the baseline case, some heat release begins in mid-compression, partially counteracting the compression work. In A–C, the ignition is more retarded, so the effective compression stroke is increased, raising net output work. Among the shapes, combustion chamber A shows the largest efficiency gain, whereas B and C are intermediate.

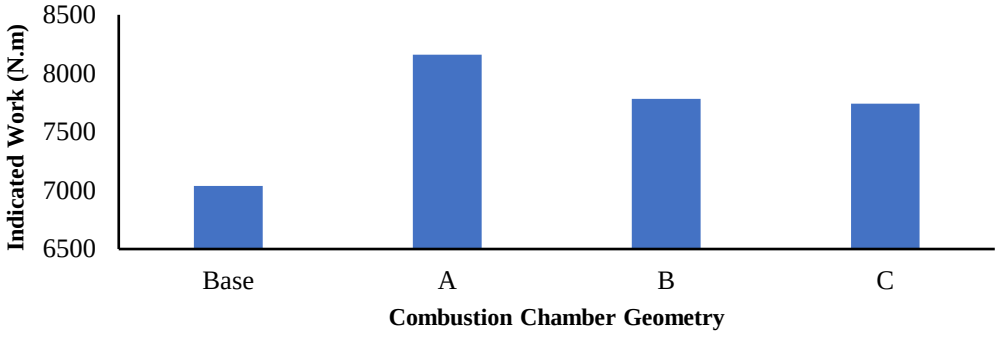

Figure 9. Indicated work output vs. crank angle for the investigated combustion chambers

Finally, pollutant emissions indicators are reported in ***Figure 10*** through ***Figure 12***. ***Figure 10*** plots NOx concentration vs CAD. Chamber A produces the highest NOx of all cases, despite its faster mixing. This is because its maximum temperature is also highest, accelerating thermal NOx chemistry during the burn. Therefore, potential mitigation strategies such as EGR and post-combustion treatments may be needed for the case of Chamber A. At ignition start, the available oxygen fraction is the same in all cases (***Figure 11***), so the difference in NOx is driven by temperature. The baseline and Chamber C have lower peak temperatures, so they emit less NOx. For soot (***Figure 12***), the trend is reversed: Chamber A has the lowest soot and the baseline has the highest. The well-mixed burn in A consumes nearly all fuel, leaving less soot precursors. In the baseline chamber, the local equivalence ratio is higher in the core, and some fuel cannot find oxygen during the rapid burn, so soot formation is higher. These trends agree with established behavior, which indicates that increased turbulence tends to reduce soot while raising NOx [1 ,11].

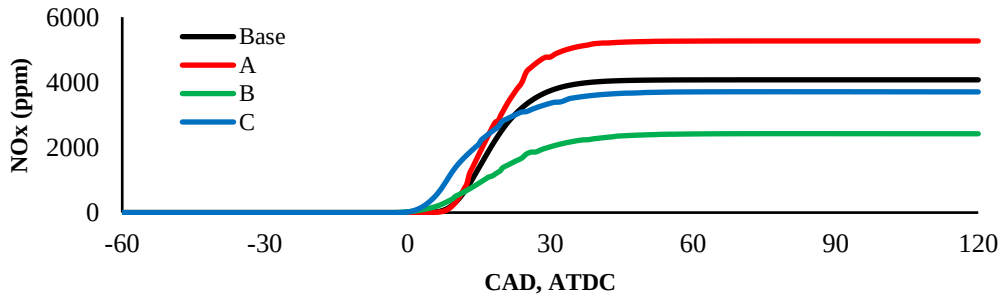

Figure 10. NOx emissions vs. crank angle for the investigated combustion chambers

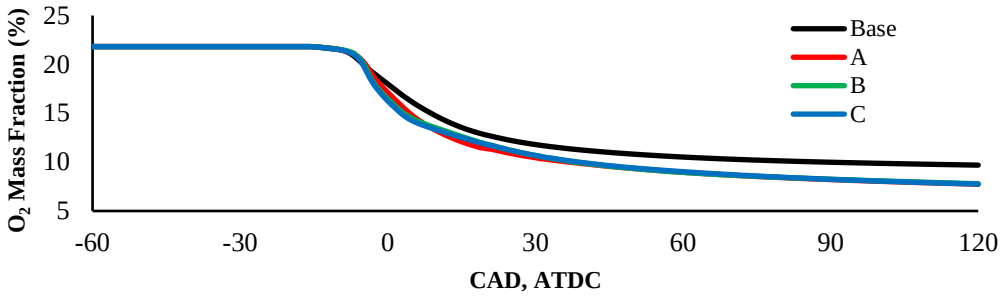

Figure 11. In-cylinder oxygen mass fraction vs. crank angle for the investigated combustion chambers

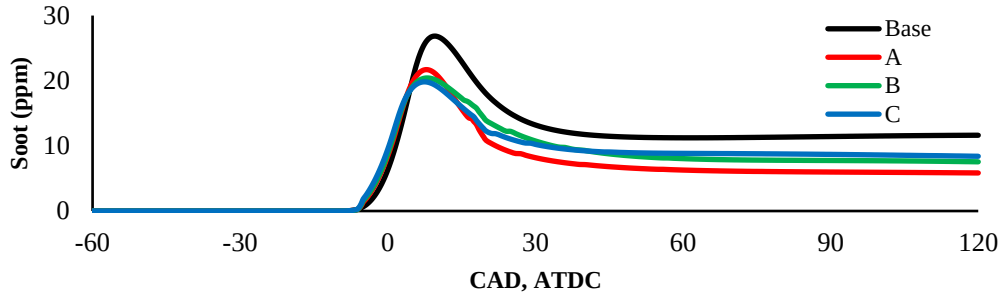

Figure 12. Soot emissions vs. crank angle for the investigated combustion chambers

The modified combustion bowls (A–C) enhance turbulence and mixing (higher TKE), which spreads combustion and raises indicated efficiency. Chamber A (the most extreme re-entrant design) achieved the cleanest burn (lowest soot) but at the cost of higher NOx due to its higher burn temperatures. Chambers B and C show intermediate behavior. Specifically, combustion chamber B, simultaneously improves power output by 10%, NOx emissions by 41 %, and soot emissions by 33%. These findings are in line with prior studies that attribute pollutant trade-offs to geometry-driven temperature and mixing differences. Crucially, these results all follow from geometry changes alone, since injection timing and intake conditions were held fixed.

4. Conclusion

This study has systematically unveiled the profound influence of piston-bowl geometry on in-cylinder flow structures, combustion dynamics, and pollutant formation in a diesel engine. By holding injection timing, intake conditions, and all other operating parameters constant, we isolated geometry as the sole variable and demonstrated that even modest bowl modifications can markedly reshape in-cylinder flows. Using validated 3D CFD, the study compared four chamber shapes under identical operating conditions. All modified bowls (A, B, C) achieved higher pre-ignition TKE, delayed combustion onset, and broadened heat-release profiles resulting in smoother pressure traces and lower peak pressure-rise rates. Therefore, they delivered higher indicated work and cycle efficiency than the baseline due to reduced negative compression work Notably, Chamber A's deep re-entrant profile delivered the cleanest burn, reducing soot emissions at the expense of an increase in NOx, thereby epitomizing the classic trade-off inherent in diesel combustion. Shallower bowls (B and C) furnished intermediate benefits, cutting soot while limiting NOx growth. Across all geometries, indicated work improved, highlighting geometry's

promise as a lever for performance enhancement without altering injection hardware. The results underscore that combustion chamber design can be tuned to improve mixing and efficiency. In practice, a re-entrant or omega-shaped bowl may allow simultaneous gains in power and soot reduction, but may require additional NOx control.

Looking forward, integrating slight injection-timing shifts or targeted EGR with these optimized bowl shapes could further mitigate NOx formation while harnessing the soot-reduction gains documented here. Additionally, extending this methodology to alternative fuels, such as biodiesel blends or hydrogen-enriched sprays, may unlock new routes for ultra-low emissions combustion. Finally, future studies should examine the long-term durability impacts of re-entrant geometries under real operating conditions, ensuring that the laboratory gains translate into field-ready solutions. By charting these next steps, our findings lay the groundwork for piston designers to craft next-generation diesel engines that balance efficiency, life, power, and environmental concerns.

Funding sources

This research received no specific grant from any funding agency in the public, academic, commercial, or not-for-profit sectors.

References

[1] Pham, V. C., Kim, J. K., Lee, W.-J., Choe, S.-J., Le, V. V., & Choi, J.-H. (2022). Effects of Piston Bowl Geometry on Combustion and Emissions of a Four-Stroke Heavy-Duty Diesel Marine Engine. *Applied Sciences, 12*(24), 13012. https://doi.org/10.3390/app122413012

[2] Torregrosa, A. J., Broatch, A., Martín, J., & Monelletta, L. (2007). Combustion noise level assessment in direct injection Diesel engines by means of in-cylinder pressure components. *Measurement Science and Technology*, *18*(7), 2131. https://doi.org/10.1088/0957-0233/18/7/045

[3] Reitz, R. D., & Rutland, C. J. (1995). Development and testing of diesel engine CFD models. *Progress in Energy and Combustion Science, 21*(2), 173–196. https://doi.org/10.1016/0360-1285(95)00003-Z

[4] Benajes, J., Pastor, J. V., García, A., & Monsalve-Serrano, J. (2015). The potential of RCCI concept to meet EURO VI NOx limitation and ultra-low soot emissions in a heavy-duty engine over the whole engine map. *Fuel*, *159*, 952-961. https://doi.org/10.1016/j.fuel.2015.07.064

[5] Raj, A. R. G. S., Mallikarjuna, J. M., & Ganesan, V. (2013). Energy efficient piston configuration for effective air motion – A CFD study. *Applied Energy, 102*, 347–354. https://doi.org/10.1016/j.apenergy.2012.07.022

[6] Abdelrazek, M.K., Abdelaal, M.M., & El-Nahas, A.M. (2022). Piston bowl shape and biodiesel fuel effects on combustion and emission of diesel engines. *Journal of Engineering and Applied Science, 69*(Article 103). https://doi.org/10.1186/s44147-022-00158-5

[7] Li, J., Yang, W. M., An, H., Maghbouli, A., & Chou, S. K. (2014). Effects of piston bowl geometry on combustion and emission characteristics of biodiesel-fueled diesel engines. *Fuel, 120*, 66–73. https://doi.org/10.1016/j.fuel.2013.12.005

[8] Dent, J. C., & Derham, J. A. (1974). Air motion in a four-stroke direct injection diesel engine. *Proceedings of the Institution of Mechanical Engineers, 188*(1), 269-280. https://doi.org/10.1243/PIME_PROC_1974_188_030_02

[9] Miles, P. C., RempelEwert, B. H., & Reitz, R. D. (2003, September). Squish-swirl and injection-swirl interaction in direct injection diesel engines. In *6th International Conference on Engines for Automobile, Proceedings, ICE2003, Capri-Naples, Italy*.

[10] Gonzalez, D.M.A., Lian, Z.W., & Reitz, R.D. (1992). Modeling diesel engine spray vaporization and combustion. *SAE Technical Paper 920579*. https://doi.org/10.4271/920579

[11] Saito, T., Daisho, Y., Uchida, N., & Ikeya, N. (1986). Effects of Combustion Chamber Geometry on Diesel Combustion. *SAE Technical Paper 861186*. https://doi.org/10.4271/861186.

[12] Prasad, B. V. V. S. U., Sharma, C. S., Anand, T. N. C., & Ravikrishna, R. V. (2011). High swirl-inducing piston bowls in small diesel engines for emission reduction. *Applied Energy*, *88*(7), 2355–2367. https://doi.org/10.1016/j.apenergy.2010.12.068

[13] Abdul Gafoor, C. P., & Gupta, R. (2015). Numerical investigation of piston bowl geometry and swirl ratio on emission from diesel engines. *Energy Conversion and Management*, *101*, 541–551. https://doi.org/10.1016/j.enconman.2015.06.007

[14] Yan, B., Wang, H., Zheng, Z., Qin, Y., & Yao, M. (2018). The effect of combustion chamber geometry on in-cylinder flow and combustion process in a stoichiometric operation natural gas engine with EGR. *Applied Thermal Engineering*, *129*, p. 199–211. https://doi.org/10.1016/j.applthermaleng.2017.09.067

[15] Abraham, J., & Magi, V. (1998). A model for multicomponent droplet vaporization in sprays. *SAE Technical Paper 980511*. https://doi.org/10.4271/980511

[16] Taghavifar, H., Khalilarya, S., & Jafarmadar, S. (2014). Engine structure modifications effect on the flow behavior, combustion, and performance characteristics of DI diesel engine. *Energy Conversion and Management*, *85*, 20–32. https://doi.org/10.1016/j.enconman.2014.05.076

[17] Kidoguchi, Y., Yang, C., & Miwa, K. (1999). Effect of High Squish Combustion Chamber on Simultaneous Reduction of NOx and Particulate from a Direct-Injection Diesel Engine. *SAE Technical Paper 1999-01-1502*. https://doi.org/10.4271/1999-01-1502

[18] 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 & Sustainable Energy Reviews*, *152*, 111683. https://doi.org/10.1016/j.rser.2021.111683

[19] Mikalsen, R., & Roskilly, A. P. (2009). A computational study of free-piston diesel engine combustion. *Applied Energy*, *86*(7-8), 1136–1143. https://doi.org/10.1016/j.apenergy.2008.08.004

[20] Rakopoulos, C. D., Kosmadakis, G. M., & Pariotis E.G. (2010). Investigation of piston bowl geometry and speed effects in a motored HSDI diesel engine using CFD against a quasi-dimensional model. *Energy Conversion and Management*, *51*(3), 470–484. https://doi.org/10.1016/j.enconman.2009.10.010

[21] Khan, S., Panua, R., & Bose, P. K. (2018). Combined effects of piston bowl geometry and spray pattern on mixing, combustion and emissions of a diesel engine: A numerical approach. *Fuel*, *225*, 203–217. https://doi.org/10.1016/j.fuel.2018.03.139

[22] Xu, L., Bai, X.-S., Li, Y., Treacy, M., Li, C., Tunestål, P., Tunér, M., & Lu, X. (2020). Effect of piston bowl geometry and compression ratio on in-cylinder combustion and engine performance in a gasoline direct-injection compression ignition engine under different injection conditions. *Applied Energy*, *280*, 115920. https://doi.org/10.1016/j.apenergy.2020.115920

[23] Shojae, K., Mahdavian, M., & Baghshani, E. (2022). Investigation of piston bowl geometry, injection timing, and EGR mass fraction to improve the performance of ISM 370 diesel engine. *Fuel*, *324*, 124422. https://doi.org/10.1016/j.fuel.2022.124422

[24] Hao, C., Zhang, Z., Wang, Z., Bai, H., Li, Y., Li, Y., & Lu, Z. (2022). Investigation of spray angle and combustion chamber geometry to improve combustion performance at full load on a heavy-duty diesel engine using genetic algorithm. *Energy Conversion and Management*, *267*, 115862. https://doi.org/10.1016/j.enconman.2022.115862

[25] Hanjalić, K., Popovac, M., & Hadžiabdić, M. (2004). A robust near-wall elliptic-relaxation eddy-viscosity turbulence model for CFD. *International journal of heat and fluid flow*, 25(6), 1047-1051. https://doi.org/10.1016/j.ijheatfluidflow.2004.07.005

[26] Popovac, M., & Hanjalic, K. (2007). Compound wall treatment for RANS computation of complex turbulent flows and heat transfer. *Flow, turbulence and combustion*, 78(2), 177-202. https://doi.org/10.1007/s10494-006-9067-x

[27] Costa, M., Catapano, F., Sementa, P., Sorge, U., & Vaglieco, B. M. (2016). Mixture preparation and combustion in a GDI engine under stoichiometric or lean charge: an experimental and numerical study on an optically accessible engine. *Applied Energy*, 180, 86-103. https://doi.org/10.1016/j.apenergy.2016.07.089

[28] Jafarmadar, S., Taghavifar, H., Taghavifar, H., & Navid, A. (2016). Numerical assessment of flow dynamics for various DI diesel engine designs considering swirl number and uniformity index. *Energy Conversion and Management*, 110, 347-355. https://doi.org/10.1016/j.enconman.2015.12.035

[29] Liu, K., Yang, J., Jiang, W., Li, Y., Wang, Y., Feng, R., Chen, X., & Ma, K. (2016). Effect of asynchronous valve timing on combustion characteristic and performance of a high speed SI marine

engine with five valves. *Energy Conversion and Management*, 123, 185-199. https://doi.org/10.1016/j.enconman.2016.06.042

[30] Liu, Y., Zhang, F., Zhao, Z., Dong, Y., Ma, F., & Zhang, S. (2016). Study on the synthetic scavenging model validation method of opposed-piston two-stroke diesel engine. *Applied Thermal Engineering*, 104, 184-192. https://doi.org/10.1016/j.applthermaleng.2016.03.094

[31] Prabhu, S. S., Nayak, N. S., Kapilan, N., & Hindasageri, V. (2017). An experimental and numerical study on effects of exhaust gas temperature and flow rate on deposit formation in Urea-Selective Catalytic Reduction (SCR) system of modern automobiles. *Applied Thermal Engineering*, 111, 1211-1231. https://doi.org/10.1016/j.applthermaleng.2016.09.134

[32] Taghavifar, H., Jafarmadar, S., Taghavifar, H., & Navid, A. (2016). Application of DoE evaluation to introduce the optimum injection strategy-chamber geometry of diesel engine using surrogate epsilon-SVR. *Applied thermal engineering*, 106, 56-66. https://doi.org/10.1016/j.applthermaleng.2016.05.194

[33] Ji, C., Liu, X., Wang, S., Gao, B., & Yang, J. (2014). A laminar burning velocity correlation for combustion simulation of hydrogen-enriched ethanol engines. *Fuel*, 133, 139-142. https://doi.org/10.1016/j.fuel.2014.05.013

[34] Shi, X., Li, G., & Zhou, L. (2007). DI diesel engine combustion modeling based on ECFM-3Z model. *SAE Technical Paper 2007-01-4138*. https://doi.org/10.4271/2007-01-4138

[35] Mobasheri, R. (2015). Analysis the ECFM-3Z combustion model for simulating the combustion process and emission characteristics in a HSDI diesel engine. *International journal of spray and combustion dynamics*, 7(4), 353-371. https://doi.org/10.1260/1756-8277.7.4.353

[36] Mittal, G., Subhash, M., & Gwalwanshi, M. (2021). Effect of initial turbulence on combustion with ECFM-3Z model in a CI engine. *Materials Today: Proceedings*, 46, 11007-11010. https://doi.org/10.1016/j.matpr.2021.02.097

[37] Krnac, D., Manickam, B., Holand, P., Pathak, U., Scharl, V., & Sattelmayer, T. (2024). Comparison of Tabulated and Complex Chemistry Approaches for Ammonia–Diesel Dual-Fuel Combustion Simulation. *SAE International Journal of Engines*, 17(7), 945-960. https://doi.org/10.4271/03-17-07-0055

[38] Priesching, P., Ramusch, G., Ruetz, J., & Tatschl, R. (2007). 3D-CFD modeling of conventional and alternative diesel combustion and pollutant formation-a validation study. *SAE Technical Paper 2007-01-1907*. https://doi.org/10.4271/2007-01-1907

[39] Viscione, D., Ravaglioli, V., Mariani, V., Silvagni, G., & Bianchi, G. M. (2024). CFD Methodology to Capture the Combustion Behavior of a Conventional Diesel Engine Retrofitted to Operate in Gasoline Compression Ignition Mode. *Energies*, *17*(16), 4061. https://doi.org/10.3390/en17164061