Paradigm

Transport and Telecommunication, 2026, volume 27, no. 1, 93-105
Transport and Telecommunication Institute, Lauvas 2, Riga, LV-1019, Latvia
DOI 10.2478/ttj-2026-0008

STUDY OF THE BEHAVIOUR OF ALTERNATIVE FUEL INJECTION STRATIFICATION THROUGH NUMERICAL ANALYSIS

Gerson Irán Maturano-Dávalos[1], Adham Ahmed Awad Elsayed Elmenshawy[2], Cesar Eduardo Cea-Montufar[3], Luis Héctor Hernández-Gómez[4], Samuel Moveh[5], Emmanuel A. Merchán-Cruz[6]

[1,3]Unidad Profesional Interdisciplinaria de Ingeniería
Pachuca de Soto, Mexico, México Campus Hidalgo IPN
[1]Gmaturano1800@alumno.ipn.mx, [3]Ccea@ipn.mx
[2,5,6]Transport and Telecommunication Institute,
Riga, Latvia, LV-1019, Lauvas str. 2
[2]Elmenshawy@tsi.lv, [5]Moveh.S@tsi.lv, [6]Merchan.E@tsi.lv
[4]Instituto Politécnico Nacional, Unidad Profesional Zacatenco,
Sección de Estudios de Posgrado e Investigación, ESIME,
Ciudad de México 07738, Mexico, Edificio 5,
3er Piso, Lindavista, Gustavo A. Madero,
[4]lhhernandez@ipn.mx

International agreements and the urgency for the reduction of gases that contribute to climate change have sparked the search for alternative fuels of satisfactory qualities to fulfil the demand posed by internal combustion engines. In spite of this, it is indeed worth analysing their behaviour at every stage on one side or the other and to observe the contrasts depending on the design of the combustion chamber. For this reason, we focus on the injection of three fuels diesel, hydrogen and methane chosen because of their contrasting physical and chemical properties. Their behaviour is compared and its velocity is focused on. In order to perform it, a numerical simulation performed using ANSYS and selected mathematical models coherent with the physical phenomena dictated by the injection of fuel, turbulence, mass conservation, the atomization process. The values for the initial conditions of the study were drawn from literature and data from real experiments dependent on the velocity, pressure and temperature. Results showed that the behaviour was better suited the deeper combustion chamber, and that methane entered with a greater velocity corresponding to being more that %10 than even other fuels, thus favouring the filling of the combustion chamber and consequently assuring better combustion. Their behaviour reflects the way they should be treated and shows that temporary alternatives can ensure a clean transition for the automotive sector.

Keywords: Numerical analysis, alternative fuels, combustion engine, alternative energy sources

1. Introduction

On account of the growing necessity of reducing on a worldwide basis the fuel consumption and pollutant gas emission, the high-efficiency internal combustion engine has become a primary target of research activities. In order to comply with environmental regulations and satisfaction of performance, the most promising area of research is that of the optimization and understanding of mixing and combustion processes, through similar of stratification behaviour.

Gasoline Direct Injection (GDI) technology is one of the techniques used to improve the efficiency of internal combustion engines as mentioned (Chen *et al.*, 2025). As such, a detailed study of its in-cylinder behaviour in the combustion chamber is crucially important for determining its effects upon the engine performance.

Alternative fuels are another promising approach to replace fossil fuels as mentioned (Luo *et al.*, 2024). Of course, it is impossible to implement any fuel in practice without enough supporting data. Thus, it is very important to study the behaviour of alternative fuels according to which they could be operated in ideal conditions.

(Rotondi and Bella, 2006) presents an analysis of the mixture formation in a GDI engine. They use a modified KIVA III code to perform simulations of spray dynamics and combustion, the KIVA code having been improved into a tool called NCF 3D, with attention being paid to the primary and secondary

atomization phenomena. Both studies used Han & Reitz model, and Nagaoka & Kawamura model. Similarly (Banerjee and Kumar, 2016) do a numerical study to analyse atomization and formation of air-fuel mixture under GDI conditions. They compare a single hole injector and the length of penetration of the liquid and vaporised phase in terms of ambient pressure, temperature, and injected fuel temperature.

Likewise, (Costa *et al*., 2016) performed an experimental characterisation campaign that aims to assess in cylinder combustion progress. They monitored the cylinder pressure and have even combined this to also use UV chemiluminescence to monitor the evolution of OH radicals from ignition to the end of the combustion as permitted by the optical accessibility of the engine. The thermodynamic and validation of this computational fluid dynamics (CFD) model presented indicates that both methods have a good predictive capability, consistent with the experimental trends. The timing of the second injection relative to ignition is central in obtaining benefits in performance with this approach. Split injections in homogeneous stratified operation are presented as a means to increase combustion efficiency and leads to less pollutants being combusted during combustion duration.

A different take on the critical importance not just of in-cylinder events but of the fuel injection system is taken also from (Postrioti *et al.,* 2016) who enlarge their attention to the entire GDI injection system in their numerical 1-D and also experimental work concentrating on wave propagation phenomena and the dependence on system geometry: how the length of the pipe operating at high pressure, internal diameter etc. affect this.

(Wang *et al.,* 2017), investigate alternative fuel types using numerical simulation focusing on the pressure of combustion, radius of propagation of flame and consumption of fuel of the process of combustion. They probe and try to understand the mechanisms constraining combustion behaviour in different injection and ignition modes. Adopt a LES approach using open FOAM. For their numerical simulation had High pressure gaseous injection via gas sphere injection model then validated against experimental data. They find timing of injection and location of Ignition have the biggest effects in combustion efficiency in a direct injection natural gas engine.

In the case of (Lei *et al.*, 2019), the focus shifted to how the dynamic behaviour of fuel injections, especially pressure buildup, affects diesel spray penetration. Analyse the two – stage nature of spray tip penetration and its correlation with injection characteristics. They proposed modified equation for predicting spray tip penetration that accounts for dynamic injection pressure. It is based on the Wakuri equation, which assumes a constant injection pressure in a modified two – stage penetration model: stage I – dynamic pressure buildup and stage II – stable pressure phase. In stage I, it was observed that there was a rapid increase in spray tip penetration and speed. In stage II, there was a slower, steady increase. The turning point between stages correlates with the injector's pressure stabilization.

Research conducted by (Wu *et al*., 2023) established that the precision of in-cylinder charge motion and hydrogen-air mixing predictions is heavily dependent on the discretization scheme selected for the momentum transport equation. By performing parametric simulations with varying injection timings and pressures, the authors identified the fundamental mechanisms of fuel-air distribution. Their numerical framework employed a strong mesh density of four million cells and utilised the RNG k-ε model for turbulence closure.

On the other hand, (Ye *et al.,* 2023) explored how intake swirl and mixture stratification affect combustion behaviour in a dual–fuel marine engine. Natural gas and diesel improved combustion efficiency and reduced emissions. The optimization of these two parameters and provides and insight into engine design strategies for better performance in large marine engines operating under dual fuel conditions. The authors used CFD simulations to analyse combustion characteristics. Key aspects of their methodology include engine model, simulation tool, swirl ratio variation, stratification control, and validation. The results showed that higher swirl ratios improve air-fuel mixing, leading to more uniform combustion and reduced cycle-to-cycle variation. However, excessive swirl, can delay ignition and reduce peak pressure due to over-mixing and lower local equivalence ratios. Optimal combinations of swirl and stratification can reduce NO_x, emissions while maintaining high thermal efficiency.

In another study (Bestel *et al.,* 2024) evaluated various spray sub-models and developed a set of best practices for physical-numerical sub-models to enable accurate simulations of liquid spray behaviour in a DISI engine. Their work found that the parcel distribution, which dictates how droplets are distributed within the spray, significantly impacts the accuracy of model calibration. They also emphasize that correctly accounting for droplet collisions is crucial for capturing the spray morphology. The study evaluated the impact of the Kelvin-Helmholtz (KH) breakup model and the Rayleigh-Taylor (RT) breakup model constants.

In contrast, (Yang *et al.*, 2024) improved combustion efficiency and reduced emissions in large-bore methanol/diesel dual direct injection engines. They proposed and evaluated innovative diesel nozzle designs, specifically asymmetrical configurations. It enhances ignition and combustion performance.

Besides, asymmetrical nozzles with inconsistent hole diameters, and unilateral hole nozzle (diesel holes only on one side) have been proposed.

The results showed an increased ignition energy. A uniform and rapid methanol ignition reduced combustion duration by 4°CA, improved burn efficiency by 2%, increased engine output by 1.8%, reduced HC emissions by 97%, CO emissions dropped by 73%, and NO_x emissions remained stable.

The paper by (Musy *et al*., 2024) looked at how the different fuel delivery methods (DI and PFI), used for hydrogen fuel in an internal combustion engine (ICE), affect both performance and emissions. This particular study compared the two methods with other fuels such as COG and methane, using simulations based on CFD that were carried out at optimal spark timing and A/F ratio across a broad range of engine speeds (2000-5000 RPM). After defining the two variables necessary for conducting the simulations, injection speed (vi) and PWM, researchers believe that these two variables can make a difference in how well the engine performs and what is required to achieve success with each of the employed fuel types.

Finally, (Myong *et al.*, 2004) investigated droplet atomization and vaporization characteristics with multicomponent fuel using experimental and numerical simulation methods. The effect of fuel composition (specifically, the mass fraction of components with low and high boiling points) on spray behaviour was analysed. Subsequently, it was developed and ratified a numerical simulation model. A simplified two-phase region to predict vapor distribution in multi-component fuel sprays was considered. The model was proposed in KIVA code. A simple two-phase region model to approximate vapor-liquid equilibrium was used. A spatial and temporal vapor concentration for each fuel component was considered, and a separated fuel injected scheme was used to model multi-component droplets.

Spray cone and spray angles increase with higher mass fractions of low-boiling point components. Besides, spray tip penetration decreases in vaporizing sprays with low-boiling point fuel. In non-vaporizing, sprays, penetration is similar across blends and dominated by high boiling point components. The simple two–phase region model predicts vapor distribution. However, it diverges slightly from experimental data over time. Consequently, this work focuses on studying the behaviour of alternative fuels under real conditions. ANSYS Workbench has been used, specifically the Fluent module. In turn, different mathematical models and 3D CAD models were considered.

Numerical simulation using CFD offers an invaluable contribution to this challenge, as it allows the complex behaviour of these fuels including their atomization, vaporization, and stratified mixing, without incurring the excessive costs and time involved in experimental testing. In this way, numerical studies accelerate the development of cleaner and more efficient injection and combustion strategies, as shown by some studies such as (Nair *et al.,* 2024). The gap between theorical research and the practical application of new generation fuels is narrowed.

2. Methodology

The importance of analysing the behaviour of an alternative fuel inside the combustion chamber lies in selecting an appropriate methodology to ensure valid and well-supported conclusions. Therefore, the following steps were undertaken to carry out this research.

- Selection of appropriate mathematical models to simulate the behaviour of alternative fuels in ANSYS Fluent.
- Selecting the appropriate model to use for simulating turbulence within the fluid domain, which is the RNG k-ε model.
- Selecting the appropriate model to use for simulating atomization and pre-combustion within the fluid domain, which is the model species transport.
- Setting up the basic geometry and dimensions of the simulation domain, including incorporating models from previous studies that can help validate the behaviour and assess how the selected model may influence the fluid's response.
- Mesh generation using ANSYS Mesh to generate high-quality mesh, including the mesh independence study.
- CFD simulation using ANSYS Fluent to solve the 3D model with the boundary conditions and to discuss the resulting flow behaviour.

To conduct this research, three distinct geometric models were proposed, each with its corresponding 3D mesh. The mesh sensitivity analysis will be presented in subsequent sections. A workstation with an AMD® RYZEN® 7 7435HS at 3.20 GHz, 24 GB RAM and a 64-bit operating processor was used. Also, it has a NVIDIA GeForce RTX 4060 20 GB.

The simulations were conducted using ANSYS Workbench 2025 R1, specifically utilizing the ANSYS Fluent and CFD Post modules. Furthermore, SolidWorks 2024 was utilized for the development of the 3D geometries.

To resolve the problem, it was necessary to extract the geometric fluid domain from the proposed models. This region of interest was obtained using the "Volume extract" tool within ANSYS SpaceClaim. To ensure high mesh quality, a grid independence study was conducted, which will be detailed in subsequent sections. Furthermore, the implementation of both hexahedral and tetrahedral elements was evaluated to enhance the orthogonal quality of the domain.

To determine the appropriate mathematical model, an extensive literature review was conducted to select the model that most accurately represents the behaviour within the simulation environment.

Finally, the results were analysed using CFD-Post. The contour tool was utilized to visualize the fluid velocities, which were represented on 2D planes to facilitate a more comprehensive understanding of the fuel behaviour.

3. Mathematical formulation

3.1. Continuity and momentum equations

In this study, the continuity equation was solved for all simulated cases to ensure mass conservation during the injection process.

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = S_m. \quad (1)$$

The mass conservation, expressed in Equation (1), is applicable to both incompressible and compressible flow systems (Çengel and Cimbala, 2006). Where, S_m is the source function, representing how the continuous phase interacts with the dispersed phase and defines the mass contribution that affects the velocity v of the liquid phase. ρ is the liquid density and $\vec{v}$ its velocity. The source for mass in the continuous phase comes from two sources: added mass from the dispersed phase and user-defined sources added to the system.

3.2. Turbulence model

The turbulence model can be obtained using the "Reynolds number" which is calculated as illustrated in equation (2) (Potter and Wiggert, 2008).

$$Re = \frac{\rho V L}{\mu}. \quad (2)$$

It can be known with this mathematical equation if it is a turbulent flow or a laminar flow. Where ρ is density, V is the velocity of the fluid, L is the diameter of the pipe and μ is the viscosity of the fluid.

However, in all the studies analysed, it was found that a turbulent model was used due to fuel conditions.

The turbulent model commonly used in literature is the RNG k-ε model. It has more advantages than the others.

- The RNG model has an additional term in its ε equation that improves the certainty of rapidly strained flows.
- The effect of swirl on turbulence is included in the RNG model, enhancing accuracy for swirling flows.
- The RNG theory analyses turbulence with Prandtl numbers, while the standard k-ε model does so using constant values.

The RNG k-ε model is illustrated in (3) (Çengel and Cimbala, 2006):

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left(\alpha_\varepsilon \mu_{eff} \frac{\partial \varepsilon}{x_j}\right) + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} - R_\varepsilon + S_\varepsilon. \quad (3)$$

In equation (3), ρ is the density of the fluid, k and ε are turbulent Prandtl numbers, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients, G_b is the generation of turbulence kinetic energy due to buoyancy, α_ε is the inverse effective Prandtl number, S_ε and R_ε are user-defined source terms, u_i and μ_{eff} are viscosity and $C_{1\varepsilon}$, $C_{2\varepsilon}$ and $C_{3\varepsilon}$ are model constants.

These features make the RNG k-ε model more accurate and reliable for a wider class of flows than the standard k-ε model.

3.3. Model species transport

This model is necessary due to the chemical reactions that occur during the injection process. In addition, the chemical parameters of the substance can be modified to obtain the desired proportion.

$$\frac{\partial}{\partial t}(\rho Y_i) + \nabla \cdot (\rho \vec{v} Y_i) = -\nabla \cdot \vec{J}_i + R_i + S_i \quad (4)$$

In equation (4) (Çengel and Cimbala, 2006), ρ is the density, $\vec{v}$ velocity of the diffusing species, R_i is the net rate of production of species i by chemical reaction, S_i is the rate of creation by addition from the dispersed phase plus any user-defined sources, Y_i is the local mass fraction of each species, and $\vec{J}_i$ is the diffusion flux of species i. This equation is pre-loaded in ANSYS Workbench.

4. Validation

4.1. 3D simulation model

By employing numerical simulation to model spray formation and atomization, different 3D CAD geometries were selected for analysis. Having multiple CAD models allows for comparison and enables drawing conclusions with stronger supporting evidence.

Initially, a cylinder was analysed. Its radius and length are 30 mm, and 90 mm. It has holes with 0.1 mm radius holes (Fig. 1), the holes are located at the top of the model. This model has been used in several studies (Banerjee and Kumar, 2016) due to its simplicity and reliability.

In this model, a mesh sensitivity evaluation was conducted. Three meshes with different numbers of elements were proposed. Hexahedral elements were used in this model. The characteristics of the meshes are described in table 1. The medium mesh has the best quality. In addition, this evaluation showed that the convergence in results between the medium and fine meshes is 2%. Therefore, the medium mesh was chosen, because an efficient use of the computing resources can be made.

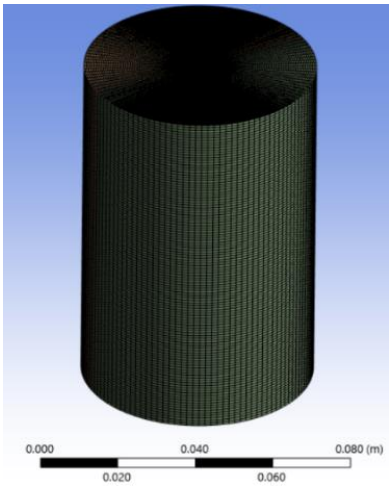
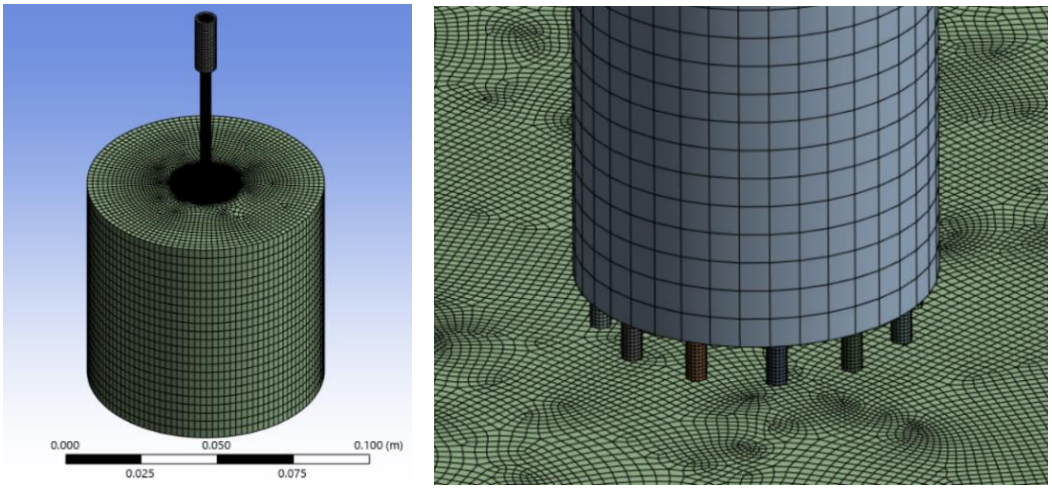


Figure 1. Mesh of the first model

Table 1. Mesh sensitivity analysis of the first model

Mesh	**Coarse**	**Medium**	**Fine**
Elements	914 397	923 853	1 204 797
Nodes	899 520	909 920	1 187 520
Mesh quality (orthogonal quality)	0.178	0.247	0.207

The second model corresponds to the internal flow generated by the coupling of a 3D combustion chamber geometry with a GDI injector. The injector considered was a Brück 1660003ac0a, selected due to its high market demand and its ability to operate over a wide range of fuel – pump pressures. The selected injector is coupled to a combustion chamber. Its radius and length are 78 mm and 77 mm, respectively (Fig.2).



(a) (b)

Figure 2. Mesh of the second model: (a) second full view model, (b) representation of the twelve holes in the injector of the second

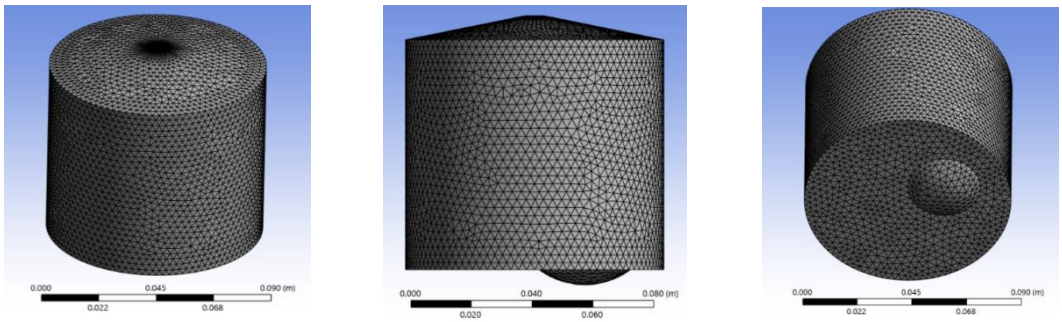
Figure 2 (b) shows that model two has twelve inlet holes with a radius of 0.1 mm. This allows for greater dispersion of the fluid in the combustion chamber. In addition, refinement was made in the area near the injector inlet, which would allow for more reliable results in this area. For the second model, hexahedral elements were considered.

As shown in Table 2, the coarse mesh is the best quality. However, the study of independence shows that the results of the medium and fine meshes are in a 0.4% range. The convergence of the medium, fine, and coarse meshes is in a 2.4% range. So, it was decided to make the analysis with the medium mesh.

Table 2. Mesh sensitivity analysis of the second model

Mesh	**Coarse**	**Medium**	**Fine**
Elements	2 515 704	2 697 992	3 068 907
Nodes	2 473 259	2 654 018	3 024 269
Mesh quality (orthogonal quality)	0.399	0.366	0.337

The third model was proposed by (Banerjee and Kumar, 2016). It is cylindrical geometry with a cone shape at the top. The geometry of a piston head is represented at the bottom, as shown in Figure 3 (c). This geometry provides a better understanding of fluid dispersion. The central cylinder has a radius of 85 mm and length of 76 mm. In addition to holes with a radius of 0.0825 mm.



(a) (b) (c)

Figure 3. Mesh of the third model: (a) third full view mode, (b) side view of the third model, (c) view of the bottom geometry

Table 3 shows that the medium mesh has the best quality with 0.219. The mesh sensitivity analysis also shows that there is a 0.8% difference in results between the medium and fine meshes, so the medium mesh was chosen for use.

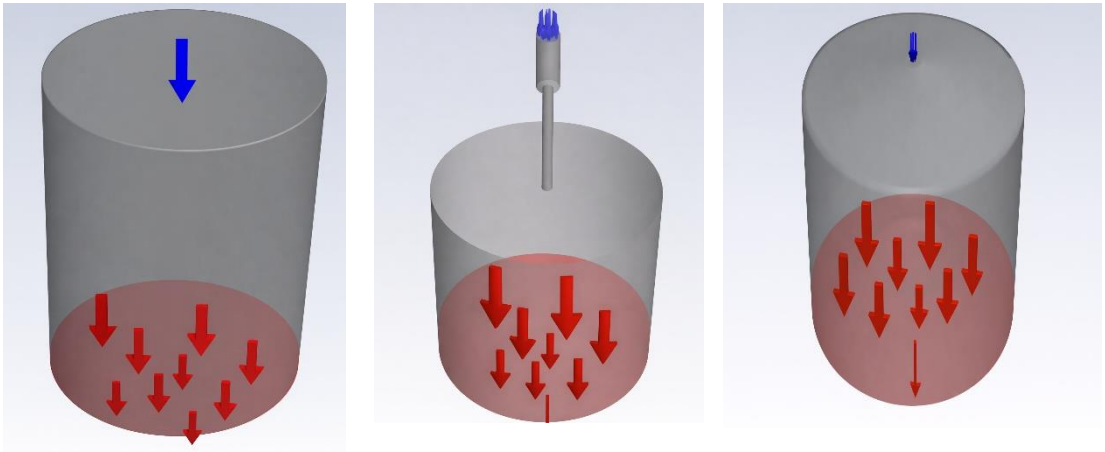
Table 3. Mesh sensitivity analysis of the third model

Mesh	Coarse	Medium	Fine
Elements	32 262	90 916	666 424
Nodes	176 061	511 117	3 863 877
Mesh quality (orthogonal quality)	0.206	0.219	0.202

It should be noted that in the third model, the tetrahedron method was used to increase the quality of the mesh. For this reason, the number of nodes far exceeds the number of elements. Previous models used the hexahedron method. However, it was necessary to use the body sizing method to improve the quality of the mesh.

4.2. Boundary conditions

Figure 4 shows the fluid flow direction, where blue arrows indicate the inlets and red arrows represent the outlets. Furthermore, the boundaries of the geometric model are depicted. The various locations of the inlet orifices are also observable; for instance, Figure 4 (a) show the single inlet orifice at the top, Figure 4 (c) illustrates the twelve-orifice configuration, and finally, Figure 4 (b) illustrates the upper region through which the fluid enters.



(a) (b) (c)

Figure 4. Inputs (blue arrows) and outputs (red arrows) of the three models

Table 4 presents the initial conditions for the simulated fuels. The selected operational variables included initial velocity, initial pressure, and initial temperature. The values were obtained from previous scientific studies (Liang, 2019), as well as from the average datasheet values of conventional fuel injection systems. The initial temperature is defined as the thermal state of the combustion chamber immediately prior to the onset of the compression and combustion processes.

Table 4. Initial conditions

Initial velocity	Initial pressure	Initial temperature	Fuels
107.9 m/s	5 MPa	400 K	Diesel, hydrogen, methane

The fuel properties presented in Table 5 highlight the vast differences in magnitude-specifically regarding density and viscosity among diesel, methane, and hydrogen. The impact of these physical and chemical properties on the simulation results will be detailed in the following sections of this paper.

Table 5. Fuel properties for simulation

Fuel properties	Diesel	Hydrogen	Methane
Density (kg/m^3)	900	0.083	0.657
Viscosity (kg /m s)	2.80X10^{-3}	8.35X10^{-6}	1.59X10^{-5}
Thermal conductivity (W/m K)	0.130	0.186	0.500

4.3. Simulation of the models

After defining all the simulation parameters and selecting the corresponding meshes, the three models were simulated with their initial conditions and the selected fuels (diesel, hydrogen, and methane).

The first model was simulated with the three different fuels and the initial conditions previously described. Figure 5 shows the behaviour of the three fuels simulated (diesel, hydrogen, and methane). A similar overall pattern can be observed among them. However, the spray, dispersion, and dispersion velocity varied. In Figure 5 (c), the jet reaches the highest velocity compared to the other. Additionally, its penetration into the combustion chamber is greater. On the other hand, the spray angle is similar in the three simulations. This can be explained by the fact that model 1 has a single orifice, which limits the lateral dispersion of the jet within the combustion chamber. However, it promotes vertical penetration, which may enhance the combustion process.

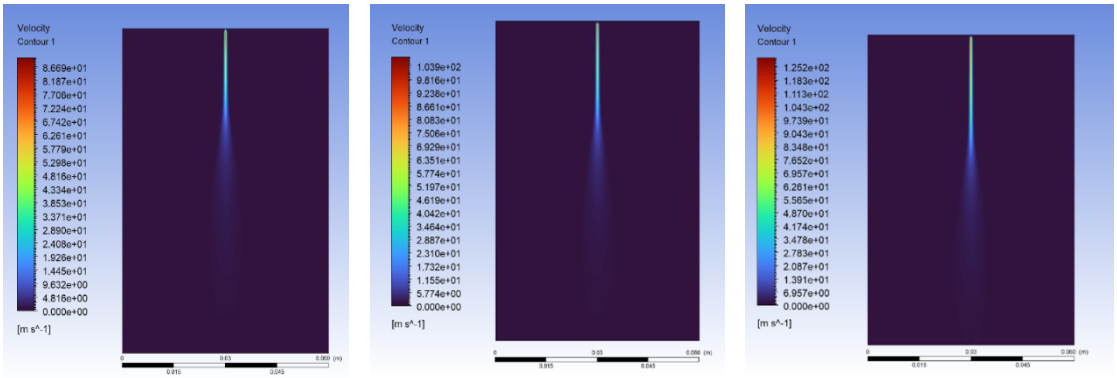


Figure 5. Velocity field of the ZY plane for model (a) diesel, (b) hydrogen, (c) methane

In the second model, an improved spray dispersion can be observed because of the injector's 12 orifices. Consequently, the spray angle is larger, and the liquid phase is smaller than the dispersed phase (Fig. 6). The three fuels exhibited similar behaviour. However, the velocity depends on the fuel. Liquid dispersion is more effective with 12 orifices. Therefore, the mass flow is distributed throughout the entire combustion chamber, allowing higher velocities to be achieved. In addition, both the velocity and the dispersion are influenced by the properties of the injection fuel, such as density and viscosity. Also, the maximum velocity was obtained in the methane due to its physicochemical properties. Consequently, improved dispersion within the combustion chamber was achieved.

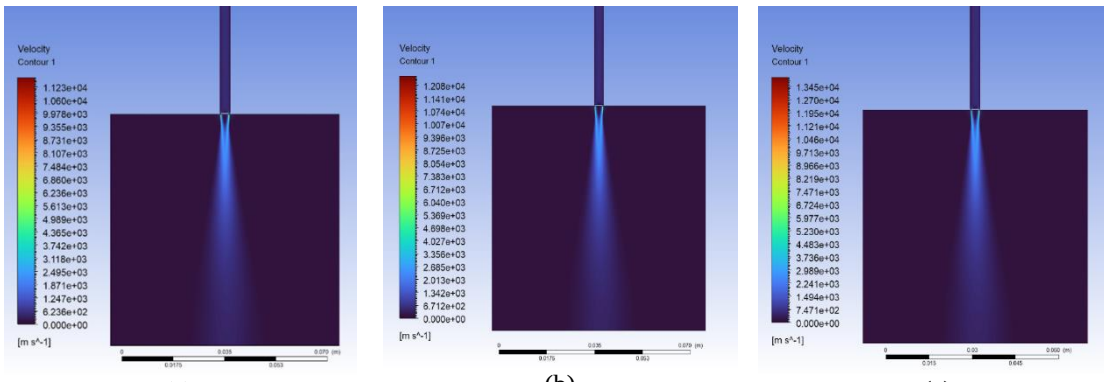


Figure 6. Velocity contours of the XY plane for model 2 (a) diesel, (b) hydrogen, (c) methane

In the last model, there is a good dispersion and penetration within the combustion chamber (Fig. 7). The dispersion of the fuel jet is observed from the orifice outlet. Model 3 features eight orifices, which ensure a balance of good dispersion while reducing the mass flow injected into the combustion chamber. However, upon closer examination, it is observed that the liquid dispersion is not uniform. It was also noted that the liquid phase is short due to the geometry of the combustion chamber. The dispersion phase predominates, contributing to a proper and efficient combustion process.

The maximum velocity develops when methane is simulated. The highest velocity is 110.8 m/s, compared to the lowest obtained with diesel at 101.1 m/s. The variation in velocities can be attributed to the chemical – physical properties of the selected fuels.

It was also observed that, across the different fuels, the penetration remained constant. The lower part of the combustion chamber was reached, and it is further noted that the geometry of the piston head contributes to dispersing part of the spray. However, since it is not located at the centre, the geometry only assists a portion of the spray.

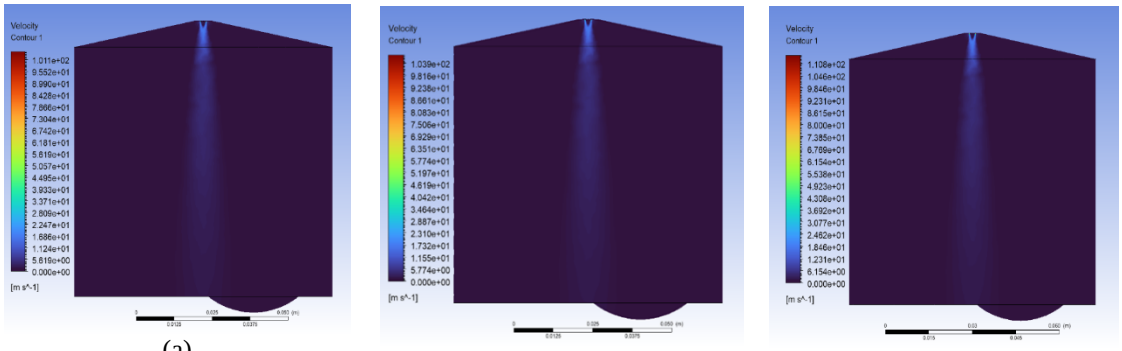


Figure 7. Velocity contours of the ZY plane for model 3: (a) diesel, (b) hydrogen, (c) methane

The upper part of model 3 has a conical shape and does not disturb the dispersion. However, it contributes to a greater dispersion due to its increased length.

5. Results

Figure 8 compares the velocities for each fuel in model 1. Similar behaviour is observed in the three models. However, there is a variation among the velocities themselves. As illustrated in Figure 8, methane reaches a velocity of approximately 130 m/s, compared to 105 m/s for hydrogen and 90 m/s for diesel.

Similarly, an almost linear behaviour is observed in Figure 8. This is due to the presence of only one orifice, which allows for a progressive response as the fuel is injected. However, as observed in Figure 5, the dispersion is inadequate.

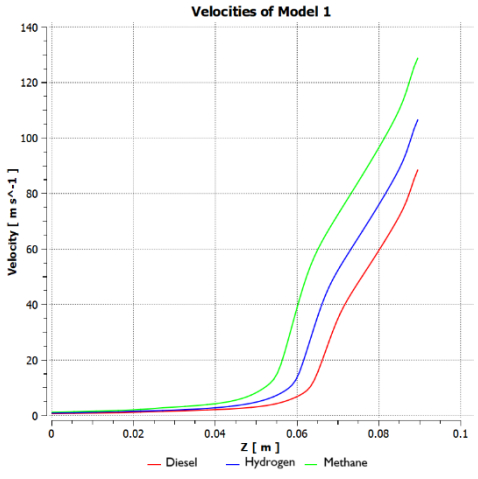


Figure 8. Behaviour of fuel velocities in model 1

In the second model, the velocities increase significantly despite the same initial conditions. The rise in velocity is attributed to the geometry of the combustion chamber as show in Figure 9. The three fuels continue to exhibit similar behaviour, while the velocity drop observed after 0.04 meters is due to jet impingement caused by the proposed geometry. The maximum velocity reached was 2,600 m/s with methane. This indicates that the combustion chamber fills more rapidly, allowing shorter injection times.

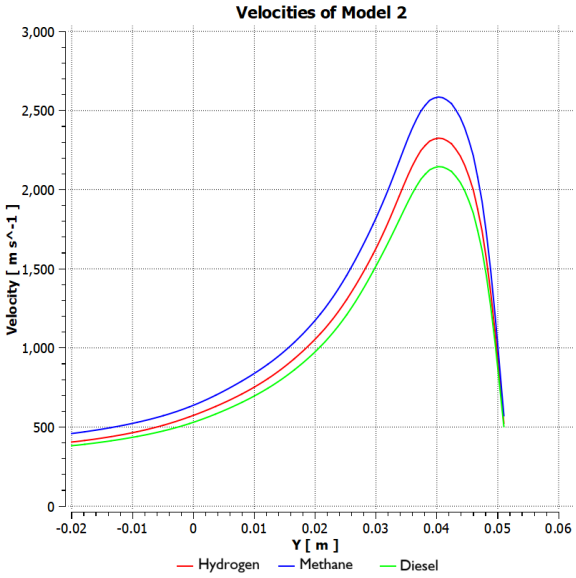


Figure 9. Behaviour of fuel velocities in model 2

In the last model, a less smooth fill behaviour with irregularities is observed. Moreover, the velocity differences of the fuels are smaller. As in the other models, methane is injected with a higher velocity of 17.5 m/s. Meanwhile, the hydrogen injection velocity was 16.5 m/s, and diesel was injected at 16 m/s. It is also observed that a sudden drop in velocity occurs after reaching its maximum value.

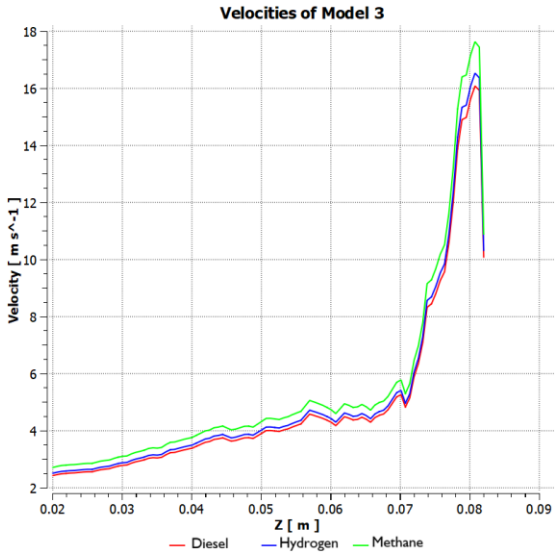


Figure 10. Behaviour of fuel velocities in model 3

The irregularity observed in Figure 10 is like Figure 7. These velocity variations may affect the dispersion within the space of the combustion chamber, and it would be necessary to apply additional pressure to adequately fill the chamber.

6. Discussion

The increase in spray penetration observed for methane can be attributed to its lower density and higher diffusivity compared to diesel. Penetration and dispersion are enhanced within the combustion chamber. The maximum methane velocity was 2,600 m/s, which was obtained from the second model. This can be attributed to the number of orifices in the second model (12 orifices). Furthermore, the size of the orifices and the combustion chamber geometry also influence this process. On the other hand, the lowest velocity across all

three models was observed with diesel fuel in model 3. As previously discussed, this model has a good dispersion angle. However, the velocity graph shows an irregular behaviour, which is interpreted as insufficient filling in the combustion chamber. The minimum velocity recorded in model 3 was 16 m/s.

Regarding the velocity contours, the first model has a linear behaviour, lacking dispersion capability. Although it showed adequate longitudinal penetration into the combustion chamber, and it did not achieve horizontal dispersion. In contrast, the second model demonstrated both characteristics: a wide dispersion angle and suitable penetration within the combustion chamber.

Other studies compare the injection behaviour (Banerjee and Kumar, 2016) with geometric deformation or changes to simulate the compression stroke in an internal combustion engine. However, this study also complements the comparison by including other models and fuels.

It can also be verified in the velocity graphs that the behaviour of each fuel is identical. In each model, the three fuels (diesel, methane, and hydrogen) exhibited similar behaviour. There is a congruence between the geometric model and the fluids. The difference in magnitudes is due to the fuel's specific chemical – physical properties.

On the other hand, it is important to demonstrate the simulations experimentally, as (Zhao *et al.*, 2023) did, in order to increase the reliability of the research. In addition to focusing research on reducing polluting gases (Payri *et al.*, 2024).

It is also necessary to consider the specific energy of each fuel and its reaction within the engine, as this will depend on the engine's requirements and the desired power and efficiency. Therefore, it is important to consider other factors when an alternative fuel is used.

7. Conclusions

The results of the simulations indicate that the design of the combustion chamber has a significant impact on how the injected fuel interacts with the combustion process. This is an important consideration when choosing which alternative fuels to use as the transition toward sustainable fuels continues.

The CFD simulations performed with ANSYS demonstrated various behaviours of the different fuels. For Example, Methane has a lower density than either Diesel or Hydrogen, which allows for greater dispersion velocity of the fuel in the combustion chamber resulting in improved combustion. Diesel has a higher density, which may limit dispersion velocity. Hydrogen also dispenses well, with a velocity of approximately 2300 m/s; however, this is less than Methane's maximum dispersion velocity but greater than Diesel's.

The CFD simulations were conducted by selecting appropriate mathematical models to accomplish each task. The Continuity equation was selected for representing fluid behaviour. A RNG k-ε-model was selected for predicting flow characteristics. The species transport model was selected for predicting fuel atomization based on other studies that showed satisfactory results. Acknowledging the importance of mesh independence in confirming the reliability of the CFD simulation, a mesh independence study was also conducted to understand how the selected parameters will affect the CFD simulation results. This provided insight into optimizing computation time and resources.

Therefore, it can be concluded from these results that Methane has a greater circulation velocity and thus, better dispersion within the combustion chamber compared to Diesel or Hydrogen. This finding was observed for all three models used, as explained in Table 6.

Table 6. Maximum velocities of the simulated fuels

Model/fuel	Methane	Hydrogen	Diesel
Model 1	130 m/s	115 m/s	90 m/s
Model 2	2600 m/s	2300 m/s	2100 m/s
Model 3	17.5 m/s	16.5 m/s	16 m/s

Methane developed the highest velocity across three models. It generated a greater dispersion within the combustion chamber. However, hydrogen also exhibited a velocity close to that of methane, differing by 11.5% in the first model and by 0.5% in the third model. Thus, it is another viable alternative.

This study is relevant by analysing and validating the models to determine, from the perspective of fluid behaviour, which fuel achieves a higher velocity and exhibits better performance. In addition, it determines which model develops a greater angle of dispersion and penetration.

Future work should include experimental validation and the analysis of transient injection conditions. It is also necessary to consider other factor such as compression and combustion to ensure that the fuel operates properly within the internal combustion engine. Likewise, aspects such as fuel storage and the appropriate selection of materials for the injection system must be considered.

Acknowledgements

The authors would like to thank COFAA-IPN and CITNOVA for providing the means to carry out the research internship in Latvia, as well as the Unidad Profesional Interdisciplinaria de Ingeniería Campus Hidalgo-IPN and the Transport and Telecommunication Institute, for the collaboration to carry out this study between Latvia and Mexico.

Declaration of Generative AI and AI-assisted technologies in the writing process:

The authors acknowledge the use of GenAI tools, namely OpenAI ChatGPT and Grammarly v1.2, used to ensure style consistency and grammatical correctness throughout the manuscript. All suggestions for edits provided were revised before integration to ensure accuracy and prevent bias. All intellectual content and data analysis were conducted solely by the authors.

References

1. Banerjee, R. and Kumar, S. (2016) Numerical investigation of stratified air/fuel preparation in a GDI engine. *Applied Thermal Engineering*, 104, 414–428. DOI:10.1016/j.applthermaleng.2016.05.050.
2. Bestel, D., Kim, J., Zhao, L., Zhang, A., Park, J.-W., Tagliante, F., Pickett, L., Ameen, M., & Torelli, R. (2024) A comprehensive numerical investigation on spray models for Direct-Injection Spark-Ignition engines. *Fuel*, 373, 132325. DOI:10.1016/j.fuel.2024.132325.
3. Çengel, Y.A. and Cimbala, J.M. (2006) *Fluid mechanics: fundamentals and applications*. Boston, Mass.: McGraw-Hill Higher Education (McGraw-Hill series in mechanical engineering).
4. Chen, W., He, W., Ma, Y., Yu, S., Zuo, Q., Kou, C., Ning, D., Gong, S., Wang, H., & Zhu, G. (2025) Numerical investigation of flow and combustion process in a hydrogen direction injection rotary engine. *Fuel*, 379, 133100. DOI:10.1016/j.fuel.2024.133100.
5. Costa, M., Sorge, U., Merola, S., Irimescu, A., La Villetta, M., & Rocco, V. (2016) Split injection in a homogeneous stratified gasoline direct injection engine for high combustion efficiency and low pollutants emission. *Energy*, 117, 405–415. DOI:10.1016/j.energy.2016.03.065.
6. Lei, Y., Liu, J., Qiu, T., Mi, J., Liu, X., Zhao, N., & Peng, G. (2019) Effect of injection dynamic behavior on fuel spray penetration of common-rail injector. *Energy*, 188, 116060. DOI: 10.1016/j.energy.2019.116060.
7. Liang, J. (2019) Study on emission characteristics of hydrogen direct-injection gasoline engine. *Journal of Physics: Conference Series*, 1213(5), 052101. DOI:10.1088/1742-6596/1213/5/052101.
8. Luo, Y., Wu, B., Li, Q., Tang, X., Yang, Z., Wu, C., & Wu, T. (2024) Experimental and simulation research on the lean combustion characteristics of direct-injection hydrogen engine. *International Journal of Hydrogen Energy*, 68, 398–409. DOI:10.1016/j.ijhydene.2024.04.184.
9. Musy, F., Ortiz, R., Ortiz, I., & Ortiz, A. (2024) Hydrogen-fuelled internal combustion engines: Direct Injection versus Port-Fuel Injection. *International Journal of Hydrogen Energy*, S0360319924028106. DOI:10.1016/j.ijhydene.2024.07.136.
10. Myong, K.-J., Arai, M., Tanaka, T., Senda, J., & Fujimoto, H. (2004) An experimental investigation and numerical analysis of multi-component fuel spray. *JSME International Journal Series B*, 47(2), pp. 200–206. DOI:10.1299/jsmeb.47.200.
11. Nair, A.G., P.S., T. and A.B., B. (2024) 'Numerical investigation of the effect of strut geometries on the flow characteristics in a model combustor. *Aircraft Engineering and Aerospace Technology*, 96(1), 96–112. DOI:10.1108/AEAT-03-2023-0074.
12. Payri, R., Novella, R., Nasser, K. I., & Bori-Fabra, O. (2024) Spray characterization of direct hydrogen injection as a green fuel with lower emissions. *Energies*, 17(10), 2405. DOI:10.3390/en17102405.
13. Postrioti, L., Cavicchi, A., Paolino, D., Guido, C., Parotto, M., & Di Gioia, R. (2016) An experimental and numerical analysis of pressure pulsation effects of a Gasoline Direct Injection system. *Fuel*, 173, 8–28. DOI:10.1016/j.fuel.2016.01.012.
14. Potter, M.C. and Wiggert, D.C. (2008) *Schaum's outline of fluid mechanics*. First edition. New York, N.Y: McGraw-Hill Education LLC.
15. Rotondi, R. and Bella, G. (2006) Gasoline direct injection spray simulation. *International Journal of Thermal Sciences*, 45(2), 168–179. DOI:10.1016/j.ijthermalsci.2005.06.001.
16. Wang, T., Zhang, X., Zhang, J., & Hou, X. (2017) Numerical analysis of the influence of the fuel injection timing and ignition position in a direct-injection natural gas engine. *Energy Conversion and Management*, 149, 748–759. DOI:10.1016/j.enconman.2017.03.004.

17. Wu, B., Torelli, R. and Pei, Y. (2023) Numerical modeling of hydrogen mixing in a direct-injection engine fueled with gaseous hydrogen. *Fuel*, 341, 127725. DOI:10.1016/j.fuel.2023.127725.
18. Yang, Y., Long, W., Dong, P., Qian, Y., Cao, J., & Dong, D. (2024) Performance of large-bore methanol/diesel dual direct injection engine applying asymmetrical diesel nozzle strategies. *Applied Thermal Engineering*, 244, 122674. DOI:10.1016/j.applthermaleng.2024.122674.
19. Ye, Y., Liu, H., Li, J., Liu, T., Dong, J., Liu, B., Wu, C., Yue, Z., & Yao, M. (2023) A numerical investigation on the effects of intake swirl and mixture stratification on combustion characteristics in a natural-gas/diesel dual-fuel marine engine. *Journal of Thermal Science*, 32(1), 414–426. DOI:10.1007/s11630-022-1733-3.
20. Zhao, Z., Huang, Y., Yu, X., Sun, P., Li, M., Shi, W., Guo, Z., & Wang, T. (2023) Effect of HHO addition on combustion and emission in SI engine with butanol direct injection and gasoline port injection. *Case Studies in Thermal Engineering*, 42, 102746. DOI:10.1016/j.csite.2023.102746.