

Article

Effect of Dual-Hole Nozzle Injection Angle on Primary Breakup and Near-Nozzle Spray Dynamics of High-Pressure Diesel Jets: Volume-of-Fluid Numerical Investigation

Souad Tahar ¹, Fatma Zohra Saidoune ¹, Faouzi Didi ¹, Mounir Zirari ¹, Hichem Ykrelef ¹ and Ebrahim E. Elsayed ^{2,3,*}

¹ Laboratory of Renewable Energies & Materials, University Medea, Medea 26000, Algeria; tahar.souad@univ-medea.dz (S.T.); saidoune.fatmazohra@univ-medea.dz (F.Z.S.); didi.fauzi@univ-medea.dz (F.D.); zirari.mounir@univ-medea.dz (M.Z.); ykrelef.hichem@univ-medea.dz (H.Y.)

² Department of Electronics and Communications Engineering, Faculty of Engineering, Mansoura University, Mansoura 35516, El-Dakahilia, Egypt

³ Department of Scientific Research Laboratory, Azerbaijan University of Architecture and Construction, Baku AZ1073, Azerbaijan

* Correspondence: engebrahim162018@gmail.com

Abstract

Optimizing fuel injector geometry represents a critical pathway toward cleaner and more efficient diesel combustion. This study presents a comprehensive numerical investigation of the influence of the dual-hole nozzle injection angle on the primary atomization behavior and spray hydrodynamics of high-pressure diesel jets. The present analysis is deliberately restricted to cold-flow, near-nozzle primary breakup under isothermal conditions (293.15 K) so that the hydrodynamic and aerodynamic breakup mechanisms can be isolated in the immediate vicinity of the orifice before evaporation and combustion occur. Two-dimensional simulations are conducted using an Eulerian Volume-of-Fluid (VOF) framework with finite element discretization, three injection-angle configurations (0°, 5°, and 10°) under realistic engine conditions (injection pressure: 138 MPa; chamber pressure: 2.32 MPa; nozzle diameter: 100 µm). The numerical model was validated against the experimental and computational benchmark data of Ménard et al., showing reasonable qualitative agreement in jet morphology, although the 2D nature of the model leads to an exaggerated accumulation of liquid at the spray tip compared to the 3D reference. The validation therefore supports the qualitative trends and the relative angular comparison, but the absolute quantitative predictions remain subject to this 2D structural limitation. Results reveal that the injection angle exerts a decisive influence on the competition between axial momentum and radial dispersion. The neutral 0° configuration yields an overly concentrated jet with limited interfacial destabilization and poor air entrainment, whereas the 10° angle produces excessive radial spreading at the expense of axial penetration depth. By contrast, the 5° convergence angle provides the best trade-off among the three tested configurations under the studied conditions, promoting enhanced Rayleigh–Plateau instability growth, earlier ligament formation, and finer droplet generation—favorable to a more homogeneous air–fuel mixture. These findings provide quantitative guidance for dual-hole injector geometry design and demonstrate the suitability of high-fidelity VOF-based methods for resolving complex two-phase atomization phenomena relevant to low-emission diesel engine design. Quantitatively, the peak axial velocity is reached at the nozzle exit for all configurations; relative to the 0° baseline, the 5° convergence angle increases the fuel–air interfacial spreading by about 141% while retaining roughly 90% of



Academic Editors: Evangelos Tsotsas, Hyun Wook Jung, Salih Özer and Erdinç Vural

Received: 9 June 2026

Revised: 10 July 2026

Accepted: 23 July 2026

Published: 26 July 2026

Copyright: © 2026 by the authors.

Licensee MDPI, Basel, Switzerland.

This article is an open access article distributed under the terms and conditions of the [Creative Commons Attribution \(CC BY\)](https://creativecommons.org/licenses/by/4.0/) license.

its axial penetration at the monitoring positions P_1 – P_3 , whereas the 10° case loses about 26% of axial penetration for a comparable radial spread.

Keywords: diesel spray; primary atomization; injection angle; Volume of Fluid (VOF); dual-hole nozzle; high-pressure injection; ligament formation; air–fuel mixing; numerical simulation; two-phase flow

1. Introduction

Reducing pollutant emissions from compression-ignition (CI) diesel engines requires precise control of the fuel injection and atomization process. The quality of the air–fuel mixture formed inside the combustion chamber is directly governed by the spray characteristics, which are themselves determined by injector geometry, injection pressure, and in-cylinder thermo-aerodynamic conditions [1,2]. Primary atomization, the initial disintegration of the liquid jet into ligaments and droplets in the vicinity of the nozzle, constitutes the rate-limiting step that governs droplet size distribution, evaporation rate, and ultimately the formation of NO_x and soot pollutants [3,4].

The Volume-of-Fluid (VOF) method, originally introduced by Hirt and Nichols [5], has established itself as the reference technique for primary breakup simulation owing to its conservation properties and geometrical interface-tracking accuracy. Liu et al. [6] successfully applied a VOF–LES coupling to investigate the primary breakup of disturbed liquid jet sprays, demonstrating that surface-wave amplitude and frequency control the onset of ligament formation and droplet chain structures. Furthermore, the breakup processes and droplet characteristics of liquid jets injected into low-speed air crossflow were investigated in Kong et al. [7], providing fundamental physical insight into the gas–liquid shear interaction that also governs inter-jet aerodynamic coupling in multi-hole nozzle designs.

The numerical modeling of primary atomization has witnessed significant advances over the past decade, driven by the need for cleaner combustion and more efficient engines. Shinjo [8] provided a comprehensive review of computational modeling approaches for primary atomization, highlighting that the Eulerian–Lagrangian method, despite its widespread use in industrial applications, still faces significant challenges in accurately capturing the primary breakup region, where modeling remains the weakest link. Recent developments have extended the VOF framework through coupling with Large Eddy Simulation (LES) to better resolve the turbulent structures that govern ligament formation and droplet detachment [6,9]. Additionally, advanced numerical strategies employing hybrid VOF–DPM (Discrete Phase Model) approaches with LES have shown that geometric modifications, such as eccentric nozzle designs, enhance atomization by intensifying pressure fluctuations and turbulent disturbances within the nozzle, thereby promoting liquid core fracture and thin sheet breakup [10].

On the experimental side, Chen et al. [11] achieved simultaneous measurement of internal nozzle flow and near-field spray using synchrotron radiation X-ray phase-contrast imaging at injection pressures up to 90 MPa, demonstrating that nozzle inlet geometry and cavitation layer thickness critically govern the near-nozzle velocity field—findings directly relevant to the boundary conditions adopted in the present study. In addition, microscopic extinction imaging studies of diesel sprays by Elamrissi et al. [12] quantified a 15% reduction in particulate size and a 50% decrease in ligament length at the injector tip when injection pressure was raised from 300 to 700 bar, confirming the dominant role of kinetic energy in driving primary atomization. More recently, Pratama et al. [13] employed

synchrotron X-ray phase-contrast imaging combined with X-ray computed tomography to correlate internal nozzle geometry asymmetry with hole-to-hole spray variations, revealing that the nozzle length-to-diameter ratio constitutes the dominant factor affecting spray behavior. Similarly, Cao et al. [14] systematically reviewed the cavitation phenomenon in various diesel injector nozzles, demonstrating that four key geometric parameters—inlet roundness, orifice diameter, length-to-diameter ratio, and conical shape—along with dynamic factors such as injection pressure, critically influence internal flow patterns and subsequent spray atomization quality.

The role of nozzle geometry in spray development has been further elucidated through numerical investigations. Investigations using coupled LES–VOF methods have established that vortex-induced cavitation within the nozzle can simultaneously achieve higher flow rates and superior atomization quality, provided the cavitation pattern is appropriately controlled [15].

The injection angle represents a particularly influential geometric parameter in multi-hole nozzle configurations. CFD investigations of the spray inclusion angle (SIA) in direct-injection diesel engines by Cingiz et al. [16] have shown that angular variations significantly redistribute fuel within the combustion chamber, with direct consequences on combustion phasing and pollutant emissions. At smaller angular scales, Shu et al. [17] demonstrated that the spray angle governs ignition delay and unburned hydrocarbon emissions in dual-fuel combustion systems, a finding of primary relevance to the present study, which focuses on convergence angles ranging from 0° to 10° .

Furthermore, recent numerical studies using Eulerian–Lagrangian methods have highlighted the role of nozzle eccentricity and asymmetric hole distribution in enhancing spray dispersion and air entrainment, suggesting that even minor deviations from symmetric configurations can yield substantial improvements in mixing quality [18,19].

Despite these advances, fundamental VOF-based studies dedicated to the isolated effect of the injection angle on primary atomization in dual-hole nozzles operating at low convergence angles remain scarce. Most existing research has focused on global spray inclusion angles in multi-hole injectors [16] or on broader geometric parameters such as nozzle eccentricity and cavitation patterns. In the configuration considered here, the two orifice axes are directed symmetrically inward (converging configuration), causing the jets to intersect in the near-field region. This intersection promotes aerodynamic interaction and enhanced mixing between the two jets, which significantly influences the primary atomization process and fuel dispersion characteristics. This low convergence range (0° , 5° , and 10°) is of specific practical interest because, in compact dual-hole nozzles, even small angular changes modify the shared near-field entrainment region between adjacent jets and hence the local primary-breakup behavior. This regime is distinct from the large spray-inclusion angles used to control global fuel distribution at the full-chamber scale. The 0° case is retained as the parallel-jet reference, whereas 5° and 10° sample the transition from under-dispersion to over-dispersion while keeping the inter-jet near-field interaction the dominant effect; a finer angular sweep is identified as future work. The present study addresses this gap by systematically investigating, through validated two-dimensional VOF simulations benchmarked against the reference data of Menard et al. [20], the influence of three injection-angle configurations (0° , 5° , and 10°) on high-pressure diesel spray dynamics. A detailed analysis of liquid volume fraction distribution, velocity field evolution, and ligament formation mechanisms is conducted with the aim of providing quantitative design guidelines for dual-hole injector optimization toward next-generation low-emission diesel engines.

2. Modeling

The dynamics of a Newtonian, incompressible, viscous fluid are described by the following set of equations in a Cartesian coordinate system.

2.1. The Continuity Equation

This equation concerns an element's mass conservation concept. It conveys the equivalence between the variation in the mass of the element and the difference in the amount of fluid entering and exiting simultaneously [21]. In a Cartesian coordinate system, the continuity equation for an incompressible flow is:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad (1)$$

2.2. The Momentum Equation

The Navier–Stokes equations are nonlinear partial differential equations used in fluid mechanics to explain fluid motion in the continuum approximation [21]. The momentum equation is:

$$\frac{\partial(\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = -\nabla P + \mu \nabla^2 \mathbf{u} + \rho \mathbf{g} + \mathbf{F}_{st} \quad (2)$$

where $\mathbf{u}$ is the velocity vector; ρ is the density; p is the pressure; μ is the dynamic viscosity; $\mathbf{g}$ is the gravitational acceleration; and $\mathbf{F}_{st}$ is the surface tension force, which is critical for atomization.

2.3. Two-Phase Liquid/Gas Modeling

The present numerical research is based on the Volume-of-Fluid (VOF) approach, which models the fuel jet as part of a two-phase fluid system [5]. The computational domain Ω consists of two immiscible phases, Ω_l and Ω_g , representing the liquid fuel and the surrounding gas, respectively. The volume fraction α of the liquid phase is defined as: $\alpha = 1$ in the liquid phase; $0 < \alpha < 1$ in interface cells; and $\alpha = 0$ in the gas phase.

The transport equation for α is:

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \mathbf{u}) = 0 \quad (3)$$

This equation states that the interface is advected by the flow field. Solving this equation accurately without numerical smearing is addressed using high-resolution interface-capturing schemes like PLIC (Piecewise Linear Interface Calculation).

2.4. Modeling Key Physical Forces

The surface tension force $\left(\mathbf{F}_{st} \right)$ is modeled using the Continuum Surface Force (CSF) model:

$$\mathbf{F}_{st} = \sigma \kappa \nabla \alpha \quad (4)$$

where σ is the surface tension coefficient, and κ is the interface curvature, calculated from the surface normal:

$$\kappa = \nabla \cdot \hat{\mathbf{n}} = \nabla \cdot \left(\frac{\nabla \alpha}{|\nabla \alpha|} \right) \quad (5)$$

This force is responsible for the capillary instabilities (Rayleigh–Plateau instability) that cause the jet to break up and is the primary driver of primary atomization.

2.5. Numerical Setup, Boundary Conditions, and Mesh

The governing Equations (1)–(5) were discretized using a finite-element/finite-volume formulation. The two-dimensional computational domain reproduces the injection chamber, with the dual-hole nozzle resolved at the inlet. The computational domain was initialized with quiescent chamber gas and no liquid outside the nozzle; a velocity inlet corresponding to the 138 MPa injection condition was prescribed at the nozzle exit, and the chamber outlet was treated as a pressure (outflow) boundary at the chamber pressure of 2.32 MPa. The computational domain was discretized using a uniform unstructured mesh with a single global element size of $10\text{ }\mu\text{m}$ (edge length $1 \times 10^{-5}\text{ m}$). No local mesh refinement, line-wise sizing, biasing, or region-specific adaptation was applied in the near-nozzle, interface, or far-field regions.

A grid-independence study was conducted using three successively refined uniform meshes—coarse, medium, and fine—obtained by progressively reducing the global element size. The cell counts reported for each mesh correspond to the levels produced by this uniform refinement procedure. The key spray metrics monitored on each mesh were the liquid-core penetration length defined as the maximum axial distance, measured along the centerline from the nozzle exit, over which the liquid volume fraction remains continuously above a threshold value of $\alpha \geq 0.5$, beyond which the jet is considered to be fully fragmented, and the peak axial velocity, which is used here as a complementary indicator, rather than a standalone standard criterion, to verify that the near-exit velocity peak, the region most sensitive to mesh resolution near the interface, is adequately resolved, consistent with mesh-convergence practice in near-field VOF/DNS jet simulations [4]. The variation between the medium and fine meshes remained below 1.6% for both indicators, confirming that the adopted (medium) mesh yields mesh-independent results while keeping the computational cost tractable.

Time integration was performed with a time step of $5 \times 10^{-7}\text{ s}$, selected so that the Courant–Friedrichs–Lewy (CFL) number remained below unity throughout the simulation to preserve interface-advection stability. A time-step sensitivity check (halving and doubling the reference step) produced negligible changes in the monitored velocity and penetration metrics, confirming temporal convergence. The transport equation for the volume fraction was solved with a high-resolution PLIC interface-capturing scheme to limit numerical smearing of the gas–liquid interface.

It should be emphasized that, given the restricted axial extent of the computational domain used in the parametric study (approximately 3.2 mm, encompassing monitoring position P_3 at 2.8 mm), the present results characterize only the near-field primary-breakup region and cannot, on their own, support conclusions regarding fuel–air mixture homogeneity at the combustion-chamber scale. Accordingly, the interfacial-area metrics reported in the comparative analysis that follows are presented as precursor indicators of mixing potential rather than as direct, demonstrated measures of mixing quality.

3. Results and Discussion

3.1. Fuel Properties

The focus in this numerical study is diesel fuel injected into a combustion chamber; the selection of diesel was informed by Ménard’s experimental work [20], which examined the atomization and dispersion of diesel fuel and is used to validate the findings.

Diesel (gas oil) is the fuel for compression ignition engines. It is a light fuel derived from petroleum [22]. For the present numerical analysis, the relevant characteristics are the physical properties of the liquid phase that govern the two-phase flow: the density and the dynamic (kinematic) viscosity enter the momentum Equation (2), whereas the surface tension controls the capillary (Rayleigh–Plateau) instabilities through the CSF force in

Equations (4) and (5). The liquid-phase properties used in the simulation are summarized below:

1. Density: 820 to $860 \text{ kg}\cdot\text{m}^{-3}$
2. Kinematic viscosity: $2\text{--}3 \text{ mm}^2/\text{s}$ at $40 \text{ }^\circ\text{C}$
3. Surface tension: $0.025\text{--}0.029 \text{ N/m}$

3.2. Model Validation

The results of the current work are compared with those derived from Ménard's numerical and experimental work [20] in order to validate the numerical model. Table 1 presents the parameters of the Ménard computation used for validation. It should be emphasized that these parameters reproduce the reference fluid of the Ménard et al. [20] DNS study (liquid density 696 kg/m^3 , surface tension 0.06 N/m) and are therefore used exclusively for the validation case, whereas the production simulations of the present angle study (Table 2) employ representative diesel properties (density 850 kg/m^3).

Table 1. Details of the Ménard computation [20].

Injector Diameter (μm)	Injection Speed (m/s)	Density Liquid (kg/m^3)	Density Gas (kg/m^3)	Surface Tension (N/m)	Liquid Temperature ($^\circ\text{K}$)
100	100	696	25	0.06	293.15

These parameters correspond to the reference (surrogate) fluid of the Ménard et al. [20] DNS benchmark (liquid density 696 kg/m^3 , surface tension 0.06 N/m) and are used solely for the validation case; the subsequent injection-angle study employs representative diesel properties (Table 2).

Table 2. Injection parameters.

Injection Pressure (MPa)	Injection Temperature ($^\circ\text{K}$)	Density (kg/m^3)	Chamber Pressure (MPa)	Injection Diameter (μm)
138	293.15	850	2.32	100

Figures 1–3 illustrate the geometric design, the liquid's mean volume fraction distribution, and the experimental result from Ménard et al. [20], respectively. Figures 4 and 5 show our computed liquid volume fraction and velocity field under the same conditions. Based on this comparison, the dynamic behavior of the jet shows reasonable agreement, within the inherent limitations of a two-dimensional model, with Ménard et al. [20] results, supporting the validity of the selected numerical method and mathematical model.

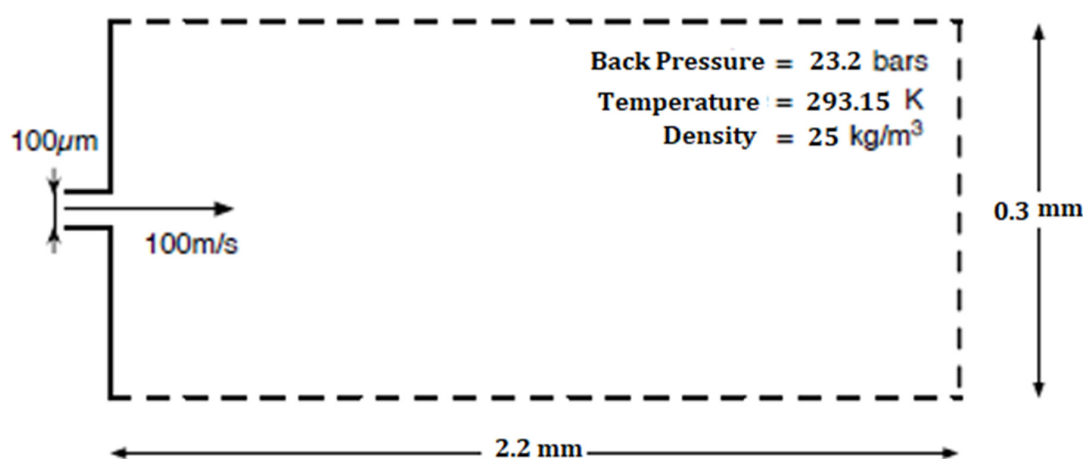


Figure 1. Geometric design and boundary conditions represented spatially [20].

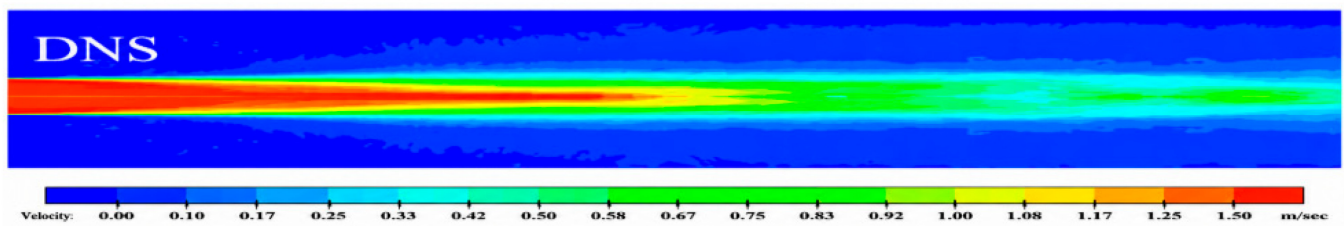


Figure 2. Spatial representation of the liquid's mean volume fraction [20].

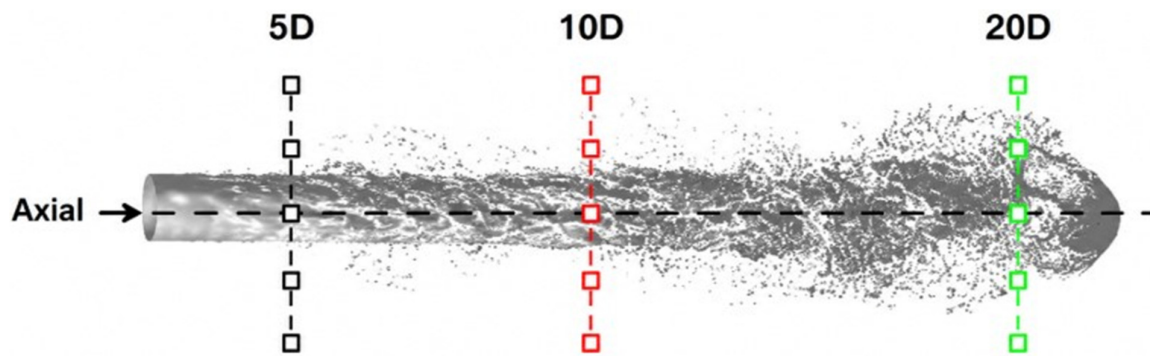


Figure 3. Experimental result [20].

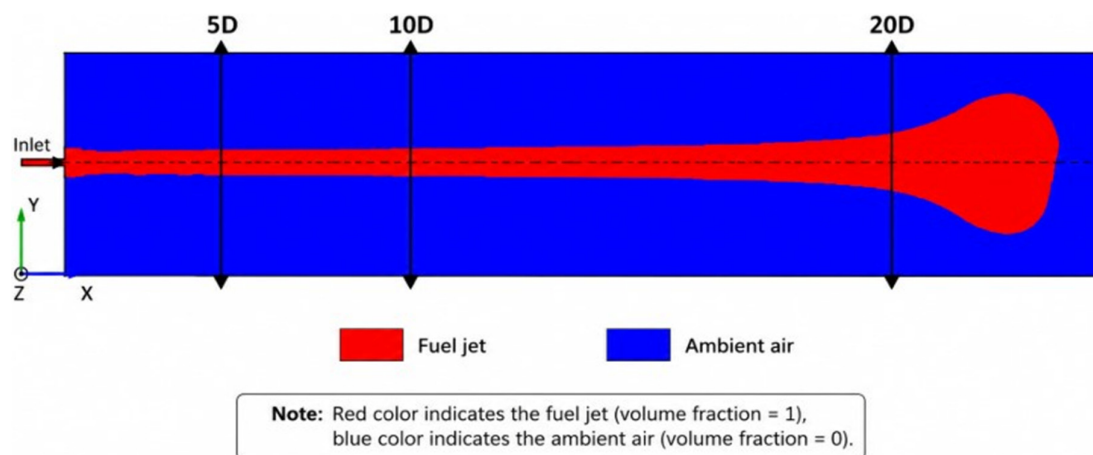


Figure 4. Spatial representation of the results of the calculation of the liquid volume fraction. The color bar denotes the liquid volume fraction α (dimensionless, 0–1).

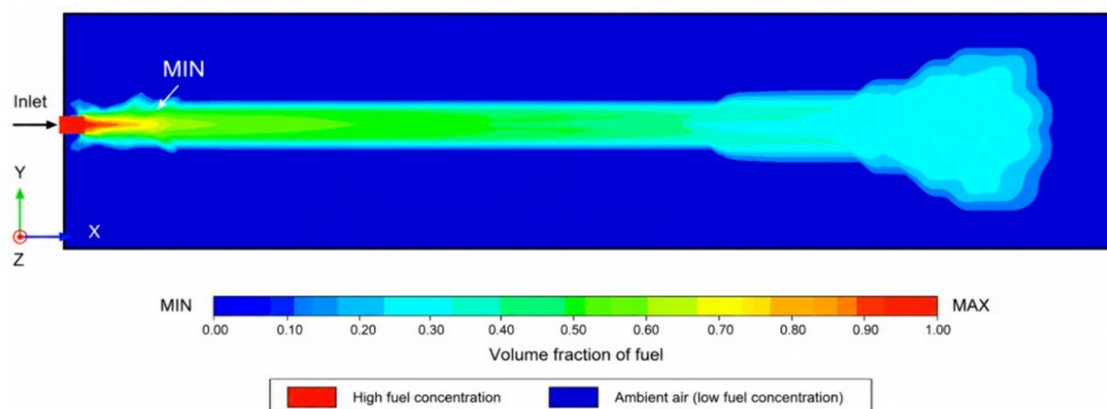


Figure 5. Spatial representation of our associated velocity field calculation results. The color bar denotes the velocity magnitude in m/s.

Beyond this qualitative morphological comparison, a quantitative assessment was performed against the Ménard et al. [20] benchmark under identical operating conditions. The liquid-core boundary along the centerline was digitized from the published photograph of Ref. [20] using intensity thresholding calibrated against the known nozzle-diameter scale reported in that reference, and the resulting liquid-core penetration and centerline volume-fraction distribution were extracted for comparison with the reference profiles. The relative deviation between the present results and the benchmark was 2.3% for the liquid-core penetration and 4.3% for the volume-fraction distribution, with a root-mean-square error (RMSE) of 0.032 (with a coefficient of determination $R^2 = 0.989$) on the compared profiles. These quantitative indicators confirm the predictive capability of the model and demonstrate that the agreement with the reference data is not merely visual but quantitatively substantiated. It should be noted that this comparison is based on the single time instant reported in Menard et al. [20] rather than a time-resolved evolution; the model is accordingly validated for the specific operating condition and time instant of that reference, and a time-resolved validation (spray penetration and/or width as functions of injection time) is identified as a priority for future work. In addition, the two-dimensional nature of the present model restricts ambient gas entrainment to a single plane rather than the full solid angle available to an axisymmetric three-dimensional jet; this structurally favors a more pronounced accumulation of the liquid phase at the spray tip than in the three-dimensional experimental reference configuration, a limitation discussed further in relation to the comparative results presented below.

This domain corresponds specifically to the reduced validation-case geometry reproducing the Ménard et al. [20] DNS benchmark (axial extent ≈ 2 mm) and is distinct from the extended domain used in the parametric injection-angle study (Figure 6).

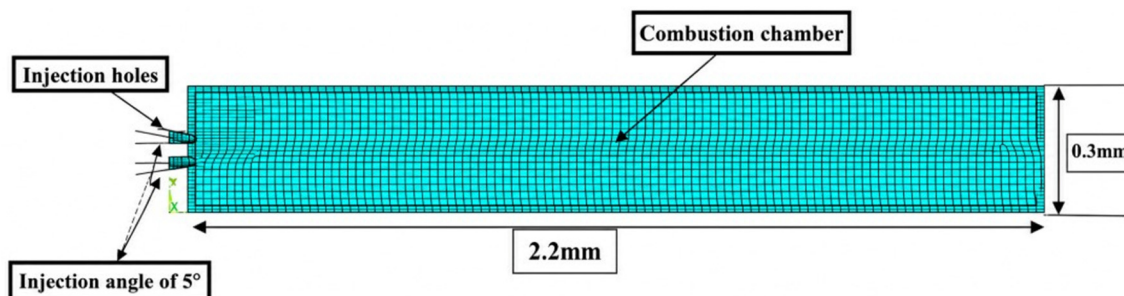


Figure 6. Depiction of the computational domain mesh as well as the two injection holes' injection angles.

3.3. Numerical Simulation Results

To characterize and comprehend the dynamic behavior of the two-phase jet flow, two-dimensional numerical calculations of liquid injection into the combustion chamber are performed using two-hole injectors at three different angles (0° , 5° , and 10°). Table 2 summarizes the injection parameters used.

Figures 6 and 7 depict the computational domain mesh with the two injection holes' injection angles and the three radial positions and axial position monitored for velocity analysis, respectively. Table 3 provides the position coordinates.

Table 3. Position coordinates in the examined configuration.

Positions	P_1	P_2	P_3
X (m)	0.412×10^{-3}	0.179×10^{-2}	0.282×10^{-2}

The domain shown corresponds to the extended parametric-study geometry (axial length ≈ 3.2 mm, height ≈ 1.0 mm), which encompasses monitoring position P_3 at 2.8 mm from the orifice exit (Table 3) and therefore differs from the reduced 2 mm validation domain of Figure 1. Consistent with the uniform meshing strategy described previously, the mesh density is uniform across the near-nozzle, interface, and far-field regions of the domain, and no local refinement or coarsening was applied. The two orifices are angled symmetrically inward (converging configuration), causing the jets to intersect in the near-field region, as visible in this view. The angles shown correspond to the convergence angles between the two holes: (a) 0° (parallel, no intersection), (b) 5° (moderate convergence), and (c) 10° (strong convergence) (see Section 1).

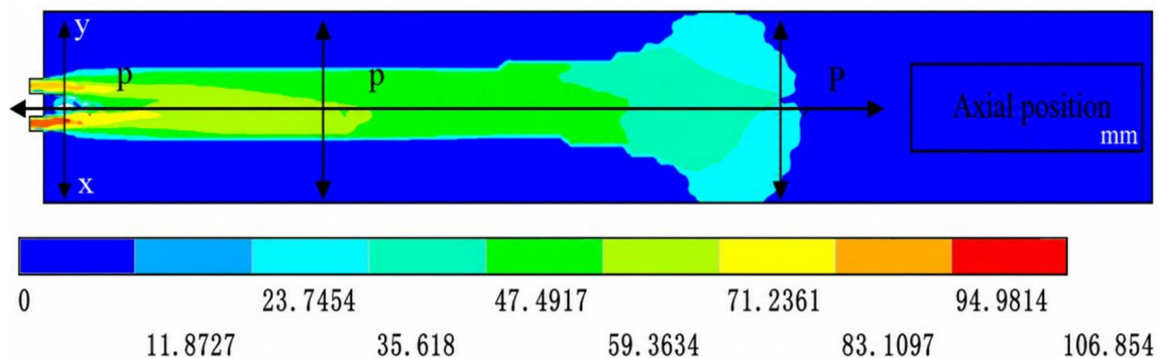


Figure 7. Three radial positions and the axial position displayed in spatial form.

As in Figure 6, the two orifice axes are angled symmetrically inward for the 5° and 10° configurations, causing the jets to intersect in the near-field region; the 0° configuration remains parallel and non-intersecting.

3.3.1. First Scenario: Two Holes in Parallel at Injection Angle of 0°

Figure 8 illustrates the spatial and temporal evolution of the jet, charting its axial penetration and transverse development. The initial stage is characterized by a compact core with minimal surface disturbance, representative of the pre-atomization phase. As the jet advances, aerodynamic instabilities overcome surface tension, leading to the formation of distinct surface fissures and the subsequent elongation of ligaments from the liquid core. The growth and eventual breakup of these filamentary ligaments mark the onset of primary atomization, facilitating a radial dispersion of fuel. This sequence underlines the crucial interaction between the intrinsic jet inertia, which generates instability, and the restoring surface tension force that initially preserves its structural integrity.

Figure 9 depicts the velocity field for a 0° injection angle, illustrating how chamber pressure conditions critically influence jet development by enhancing transverse diffusion. The velocity profile reaches its maximum at the nozzle exit, consistent with the 138 MPa injection pressure, and exhibits a sharp gradient from the high-momentum core toward the fuel–air interface. This velocity attenuation results primarily from aerodynamic shear forces that facilitate momentum transfer between the fuel jet and the nearly quiescent chamber gas.

The apparent convergence of the two jets in this velocity field reflects the downstream overlap of their dispersion envelopes rather than a geometric crossing of the orifice axes, which for this 0° configuration remain parallel; the true geometric jet intersection described for the 5° and 10° converging configurations (Figures 6 and 7) does not apply to this baseline case.

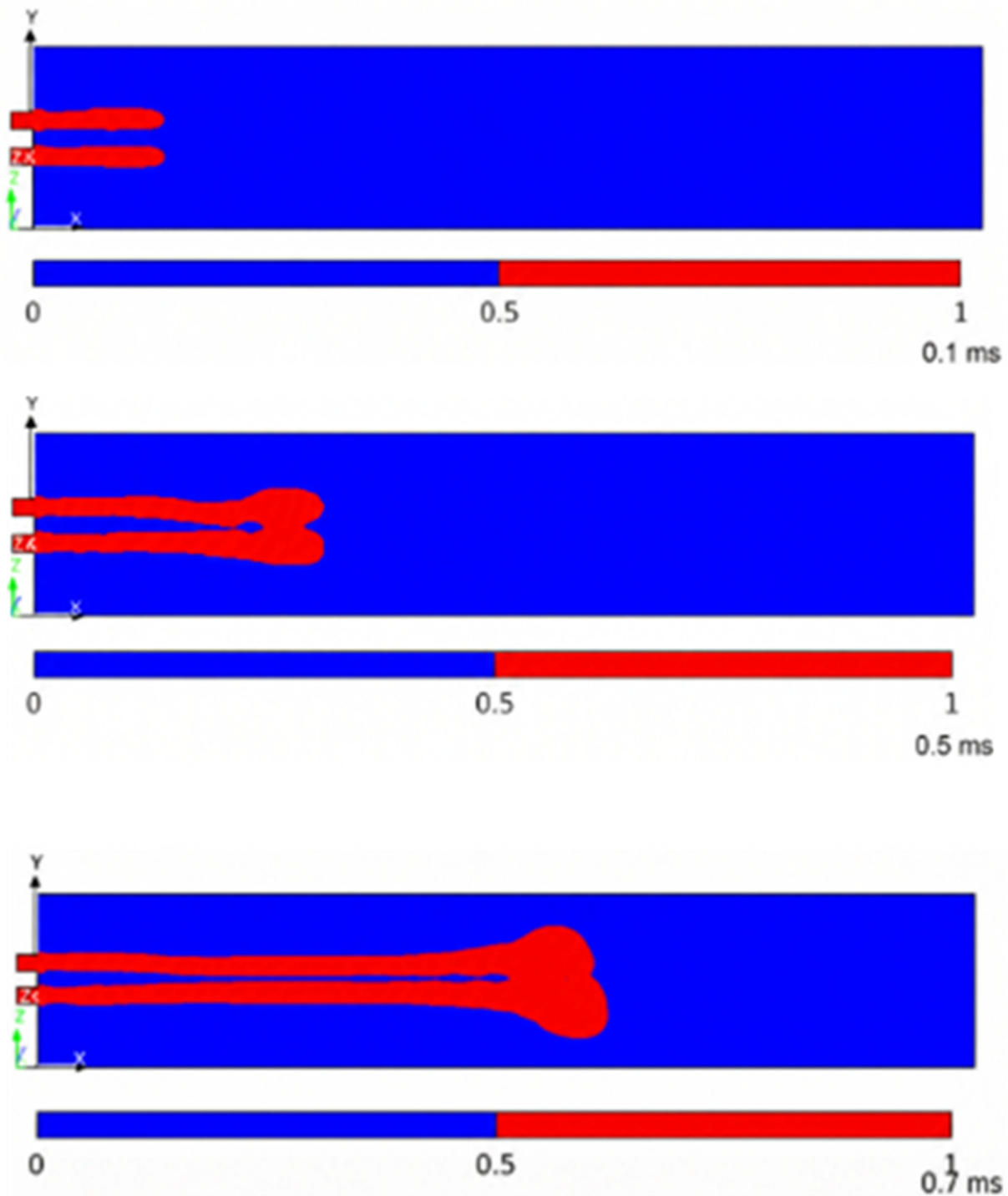


Figure 8. Spatial representation of liquid diffusion for the first scenario (0° injection angle) at $t = 0.1$, 0.5 , and 0.7 ms. The color bar denotes the liquid volume fraction α (dimensionless, 0–1).

As illustrated in Figure 10, the axial velocity peaks at the injector orifice (Position 1) and decays progressively at downstream locations (Positions 2 and 3). This profile is directly attributable to the high initial momentum from the 138 MPa injection pressure. The subsequent decay is driven by the jet's interaction with the pressurized chamber environment, where aerodynamic drag and shear forces dissipate momentum into the surrounding gas.

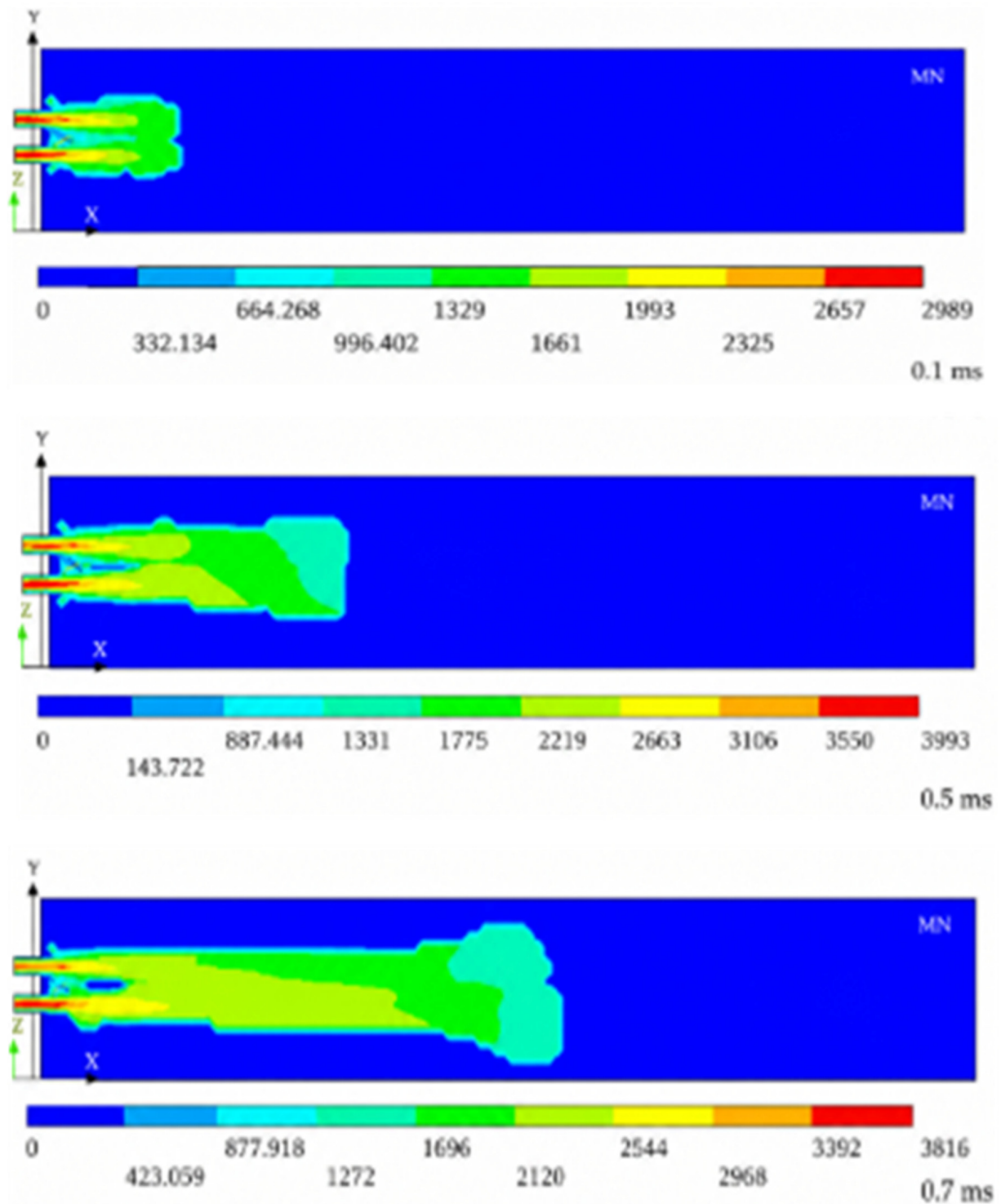


Figure 9. Velocity field's spatial representation for the first scenario (0° injection angle) at $t = 0.1, 0.5$, and 0.7 ms.

Figure 11 depicts the temporal evolution of the axial velocity profile at a fixed monitoring position. The data reveal the characteristic fluid dynamic behavior: velocity peaks during the initial injection phase and subsequently attenuates over time. The observed trend toward velocity stabilization signifies a critical transition from a momentum-dominated jet to a state where disruptive forces actively govern spray disintegration.

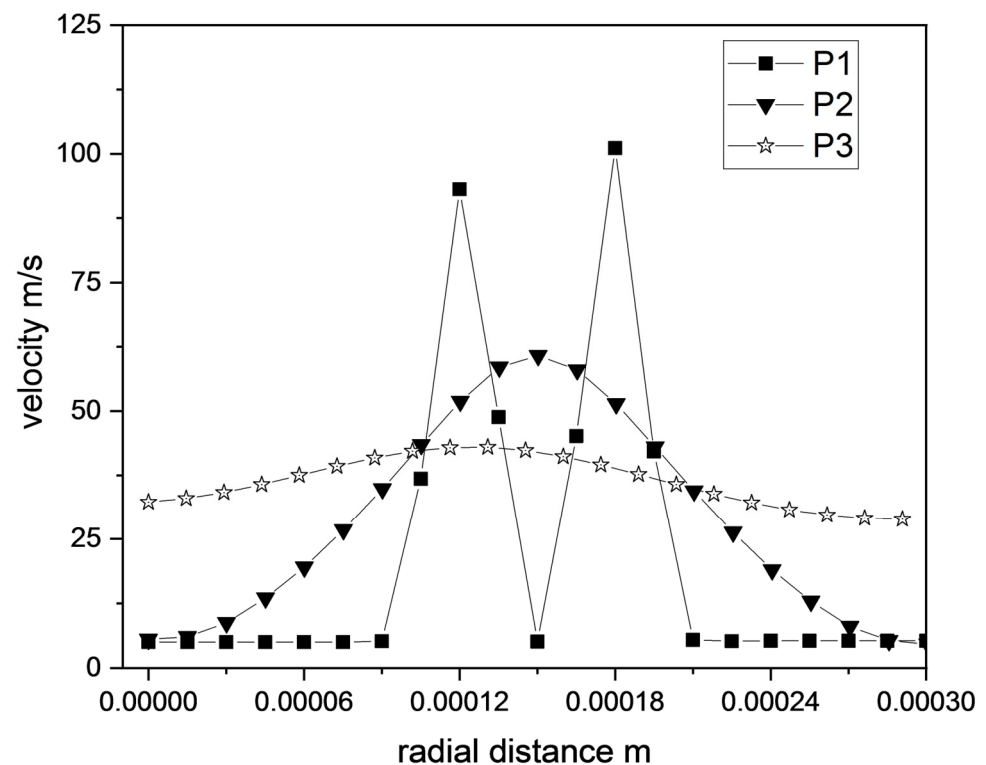


Figure 10. Velocity profiles with transverse position for the first scenario.

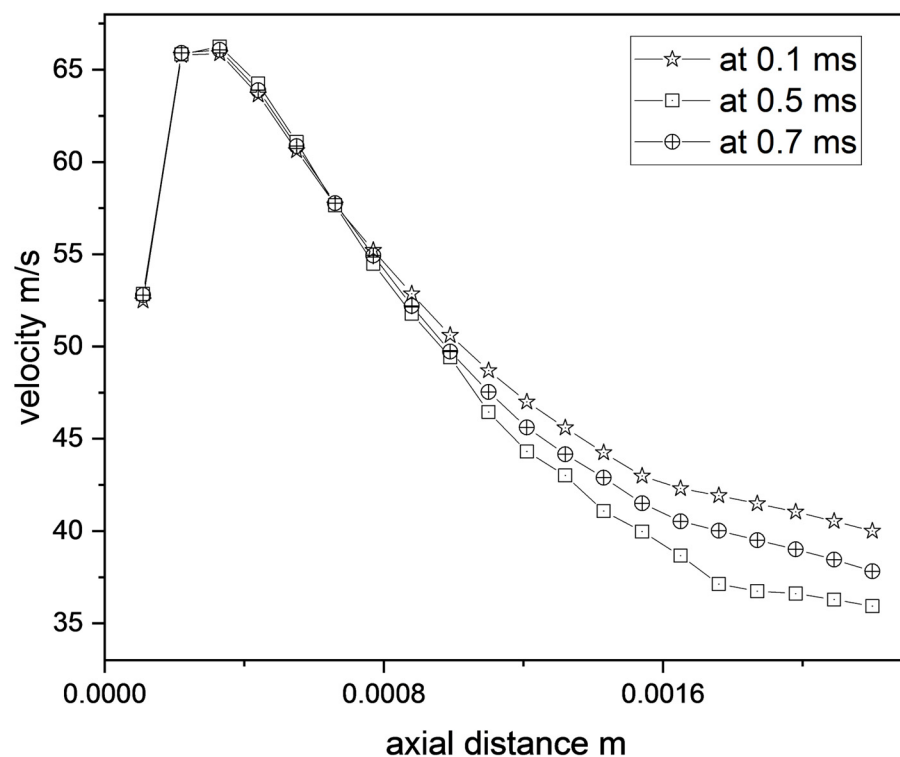


Figure 11. Axial velocity profiles for three different times for the first scenario.

These findings clarify the crucial role of injector geometry and in-cylinder conditions in determining spray characteristics. The 0° injection angle, lacking inherent convergence, produces an overly concentrated jet with delayed atomization that inherently limits air-entrainment and mixing efficiency.

3.3.2. Second Scenario: Two Converging Holes with an Inter-Jet Angle of 5°

Figure 12 illustrates the temporal evolution of the diesel fuel volume fraction for the 5° injection angle, revealing markedly enhanced atomization and dispersion from the initial injection phase. The introduction of a 5° convergence angle generates a critical radial velocity component that amplifies interfacial instabilities, thereby accelerating the primary breakup of the liquid jet into a finer droplet cloud. This controlled lateral spread maximizes the fuel–air interfacial contact area, promoting superior mixing efficiency compared to the 0° baseline case.

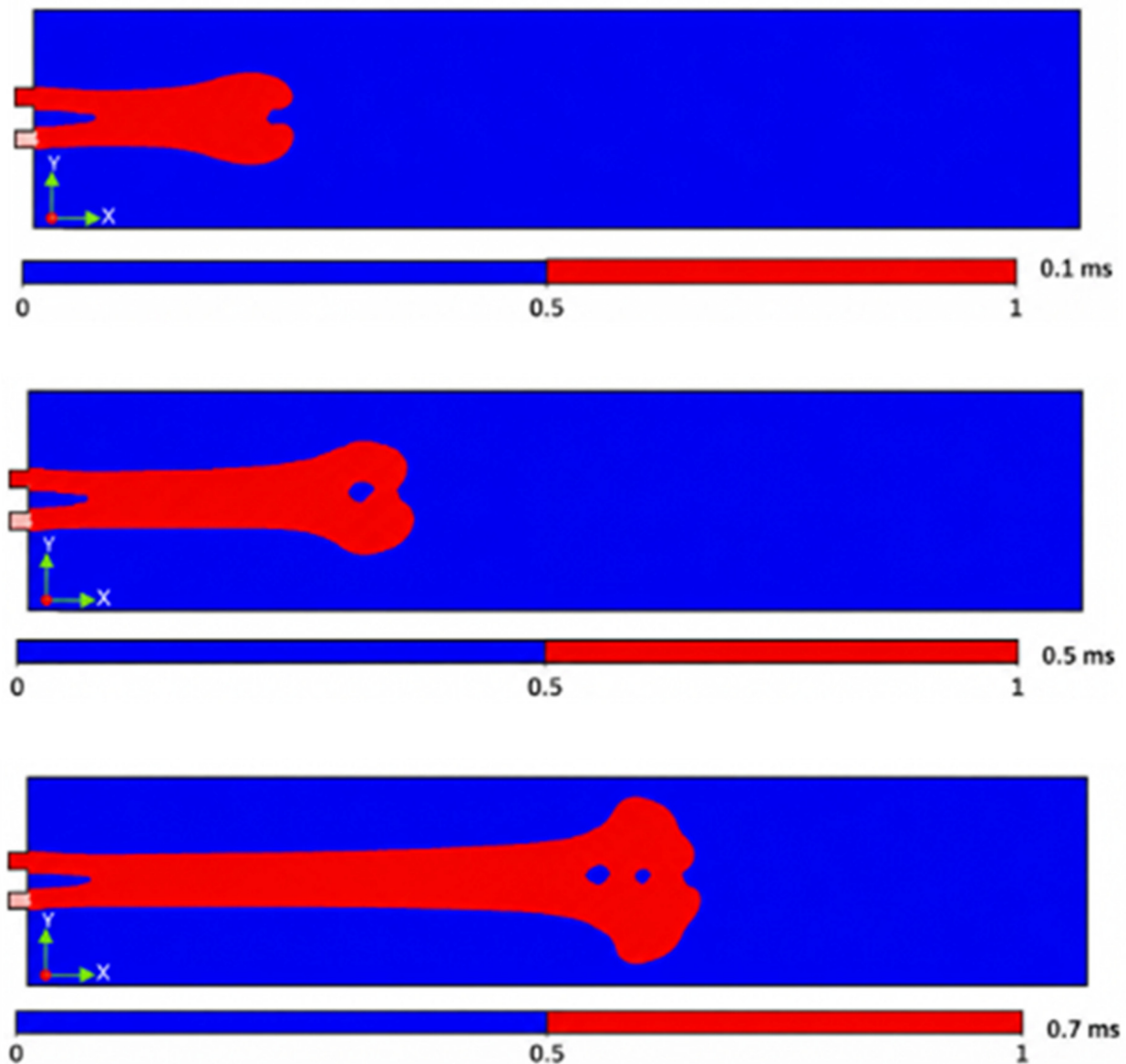


Figure 12. Volume fraction development results for the second scenario (5° injection angle) at $t = 0.1$, 0.5 , and 0.7 ms.

Figure 13 traces the temporal evolution of the velocity field for the 5° injection angle configuration. The jet initially achieves maximum velocity at the nozzle exit. The pronounced radial momentum component induced by the injection angle promotes effective lateral expansion while maintaining sufficient axial penetration, resulting in a favorable balance between radial dispersion and axial momentum.

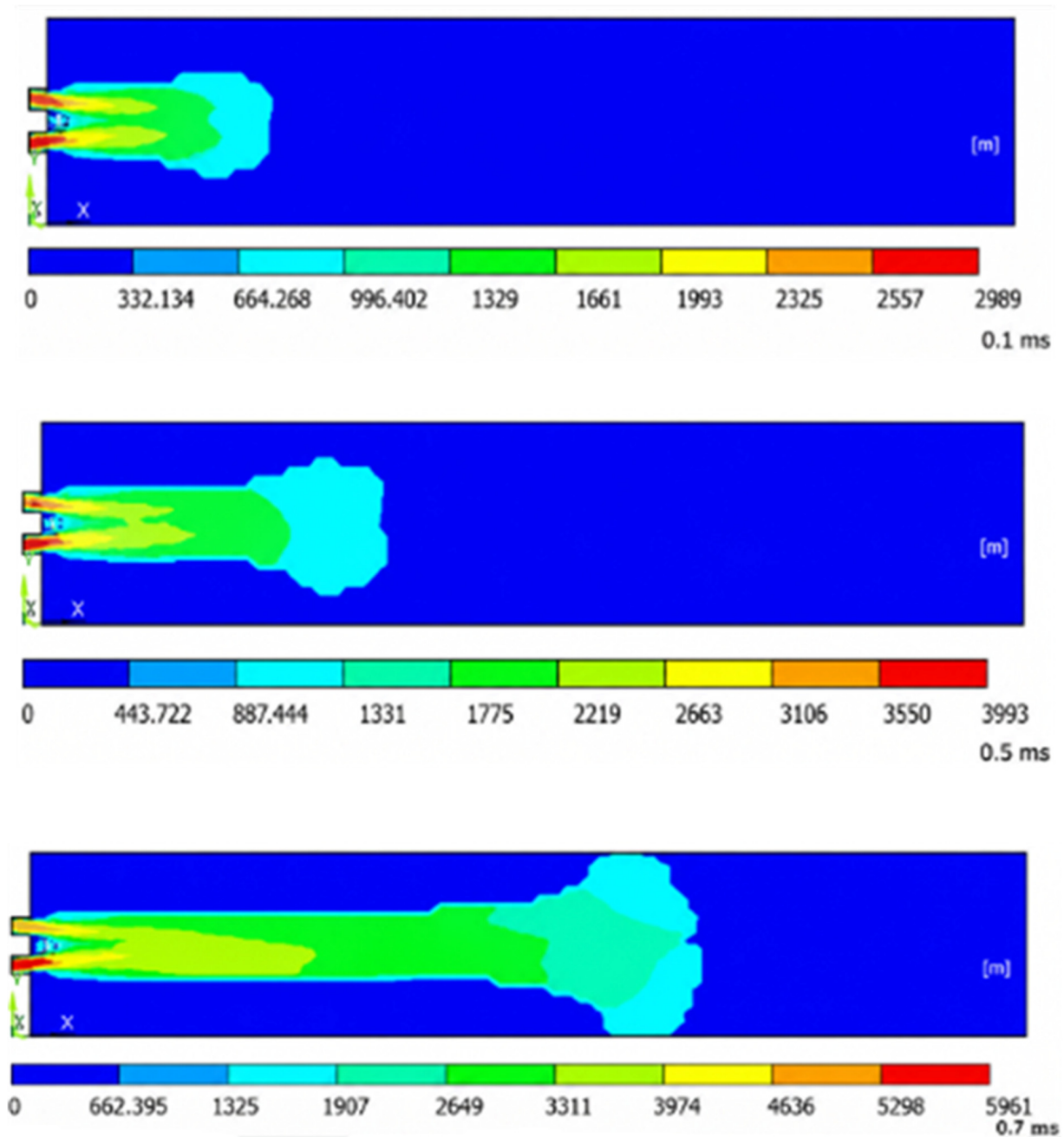


Figure 13. Velocity field spatial representation for the second scenario (5° injection angle) at $t = 0.1$, 0.5 , and 0.7 ms.

Figure 14 documents the temporal evolution of diesel spray velocity across three measurement positions. The velocity profile demonstrates rapid acceleration to a maximum at each location, with peak magnitude occurring at the first position nearest the injector tip.

Figure 15 displays the axial velocity profiles at successive time intervals for the 5° injection case. The velocity reaches its maximum proximate to the injector across all monitored positions before decaying along the chamber axis. The 5° injection configuration sustains axial penetration while introducing a radial velocity component that promotes interfacial growth; among the three cases, it yields the largest resolved liquid–gas interfacial area for a given reduction in axial penetration, rather than “optimally” converting energy in any formal sense.

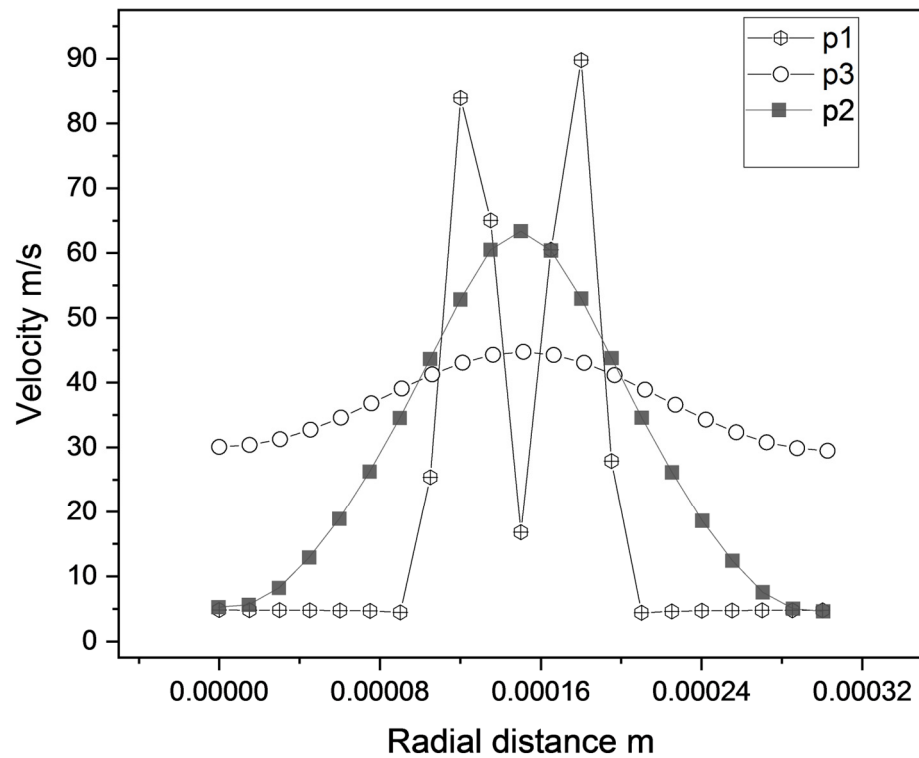


Figure 14. Velocity profiles with a transverse position for the second scenario.

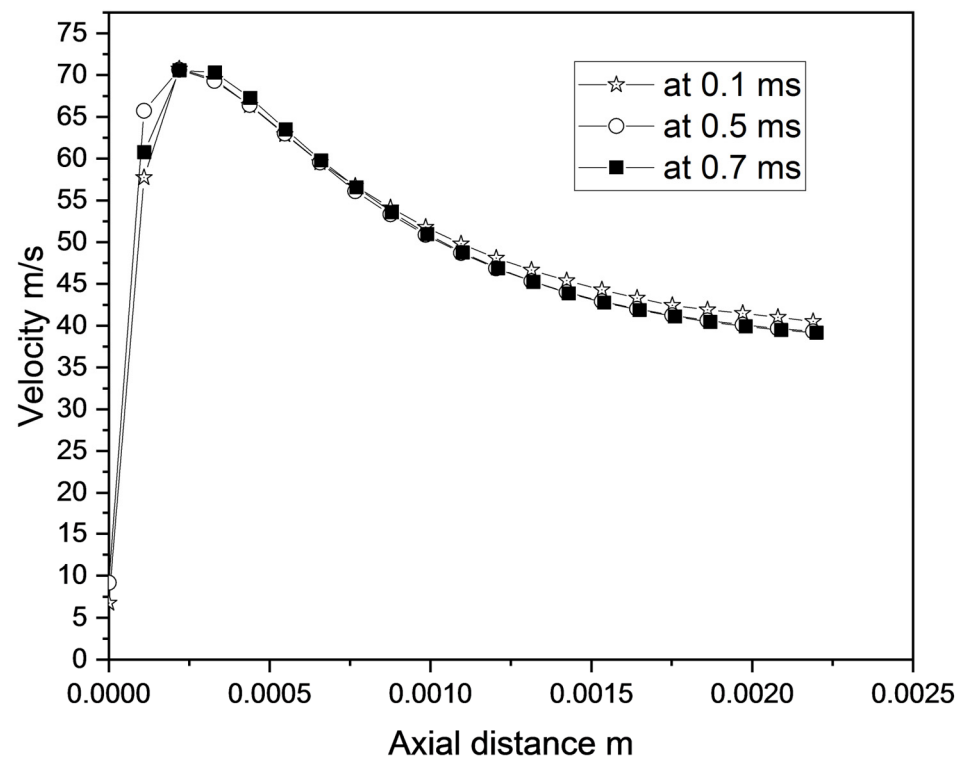


Figure 15. Axial velocity profiles for three different times for the second scenario.

3.3.3. Third Scenario: An Injection Angle of 10°

Figure 16 illustrates the temporal progression of the fuel volume fraction for the 10° injection scenario. Analysis reveals initial formation of a dense core near the nozzle with limited atomization, followed by substantial radial dispersion development. This behavior stems from the significant radial momentum component inherent to the 10° injection

angle, which enhances air entrainment and lateral spread compared to the 0° baseline configuration. However, this pronounced radial momentum occurs at the direct expense of axial penetration, producing a shallower, wider spray plume.

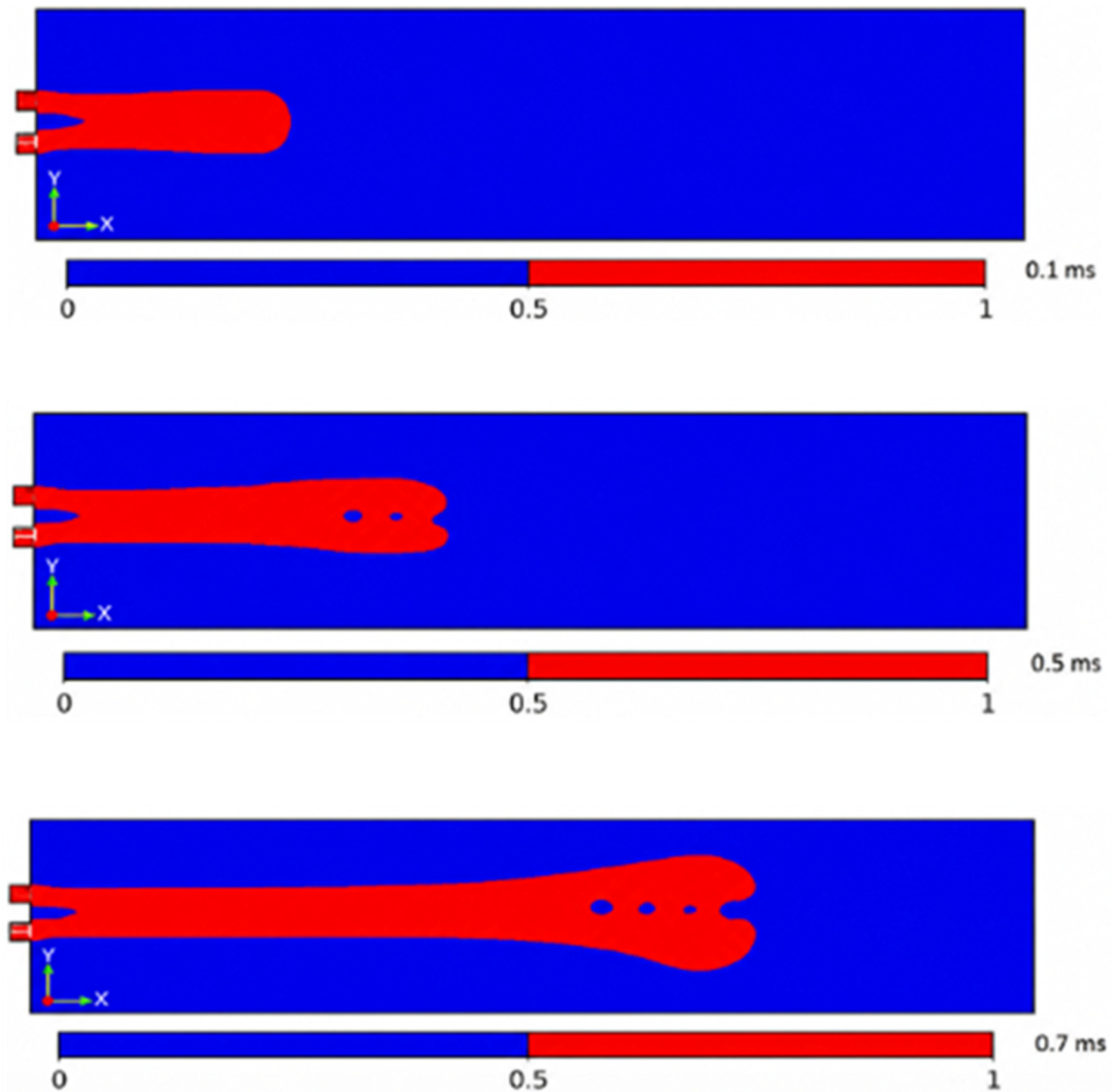


Figure 16. Volume fraction development results for the third scenario (10° injection angle) at $t = 0.1$, 0.5 , and 0.7 ms.

Figure 17 depicts the spatial distribution of the velocity field for the 10° injection configuration. The velocity profile demonstrates a pronounced gradient, decaying from its maximum at the nozzle exit to minimum values at the jet periphery. This attenuation stems from the combined influence of the steep injection angle and opposing chamber pressure, where enhanced interaction with the ambient gas through aerodynamic shear forces facilitates substantial momentum transfer.

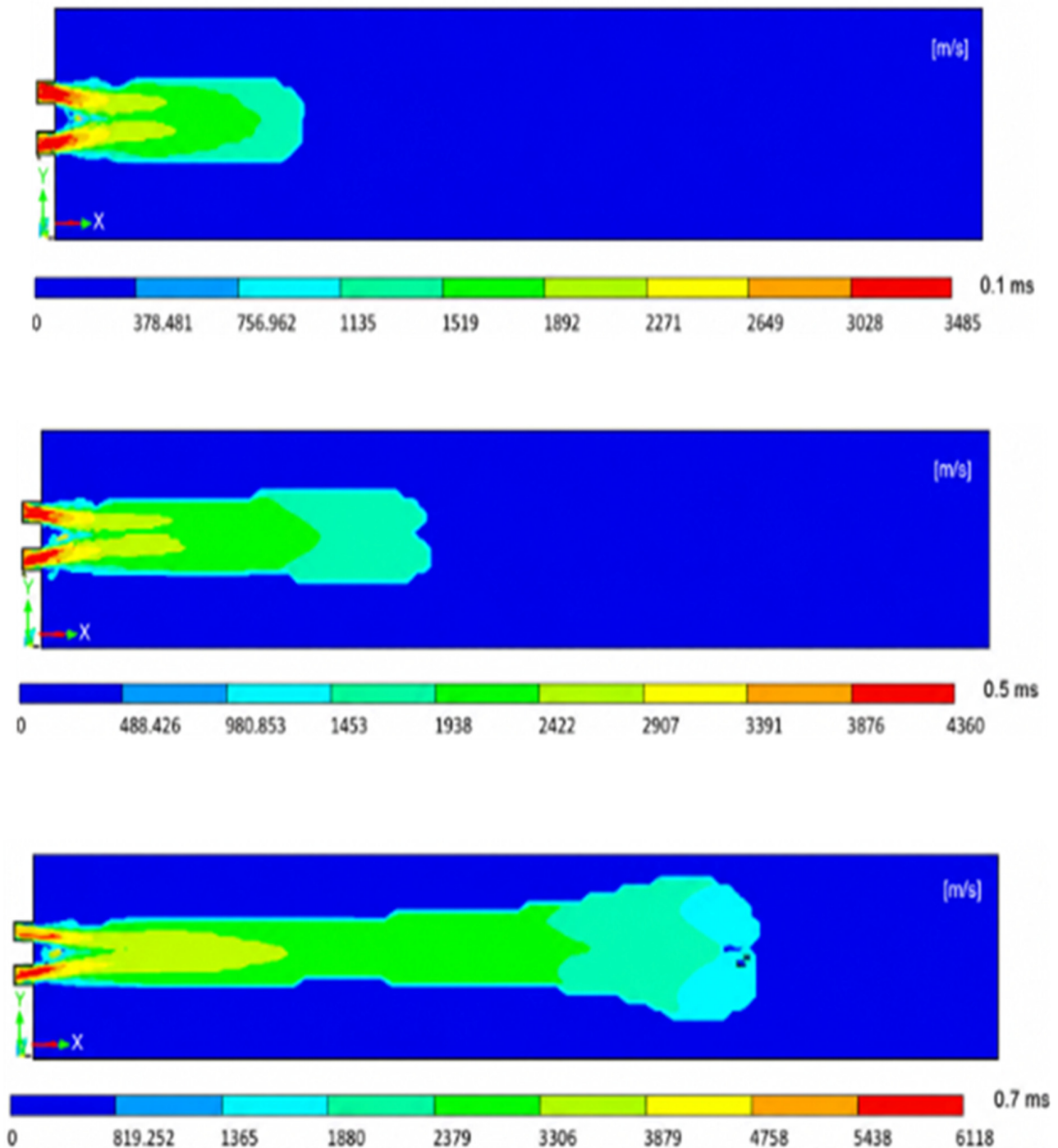


Figure 17. Velocity field for the third scenario (10° injection angle) at $t = 100 \mu\text{s}$, 0.5, and 0.7 ms.

As in Figure 6, the two orifice axes for this 10° configuration are angled symmetrically inward, and the resulting jet intersection in the near-field region is clearly visible in this velocity field, consistent with the strong convergence expected at this angle.

Figure 18 illustrates the evolution of transverse velocity across three measurement positions for the 10° injection configuration. The velocity profile demonstrates rapid attainment of peak values followed by a moderated decay, with a less pronounced attenuation rate compared to the 5° scenario. The sustained elevated transverse velocities ultimately contribute to spray over-penetration at the periphery without a corresponding increase in the resolved liquid–gas interfacial area or in ligament formation, which are the measurable indicators used here in place of a subjective notion of atomization quality.

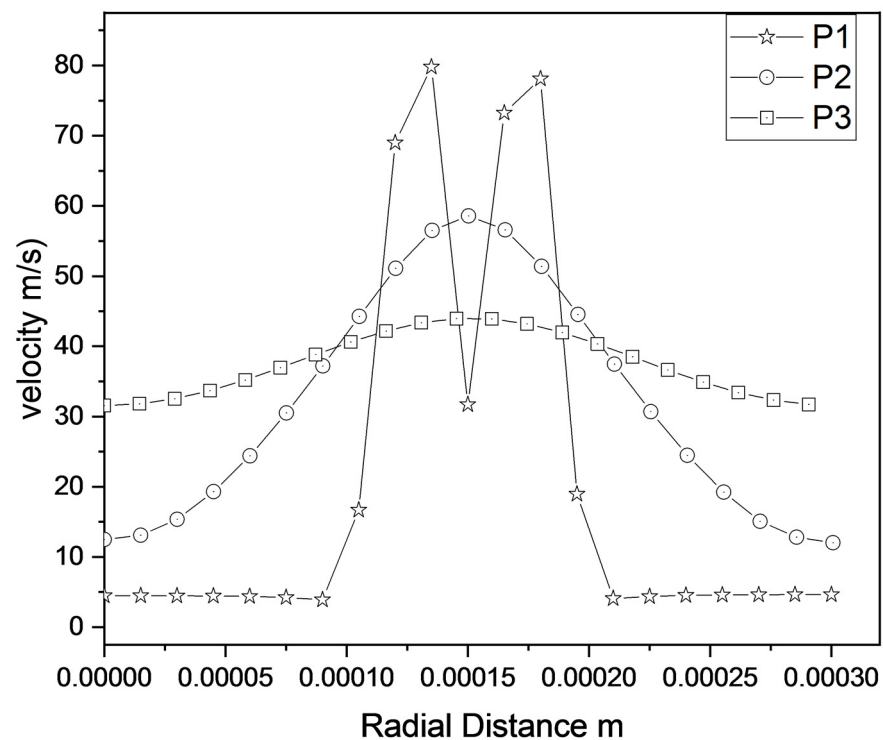


Figure 18. Velocity profiles with a transverse position for the third scenario.

Figure 19 illustrates the temporal decay of axial diesel jet velocity at a fixed monitoring position. The profile exhibits a characteristic sharp peak upon flow arrival, followed by progressive attenuation governed by aerodynamic drag forces. This axial momentum dissipation directly enables enhanced radial dispersion through conservation of mass principles.

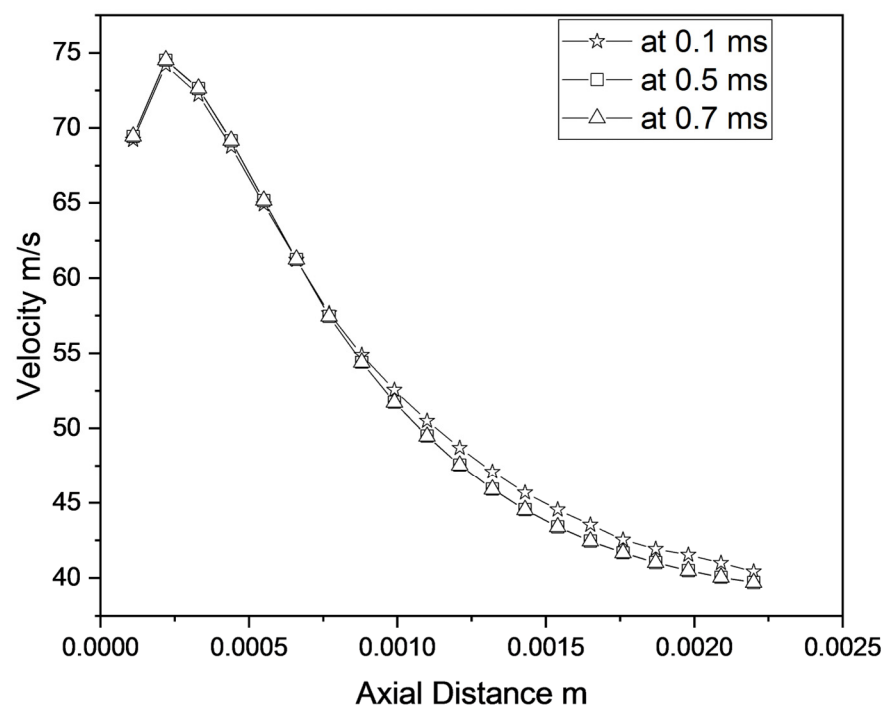
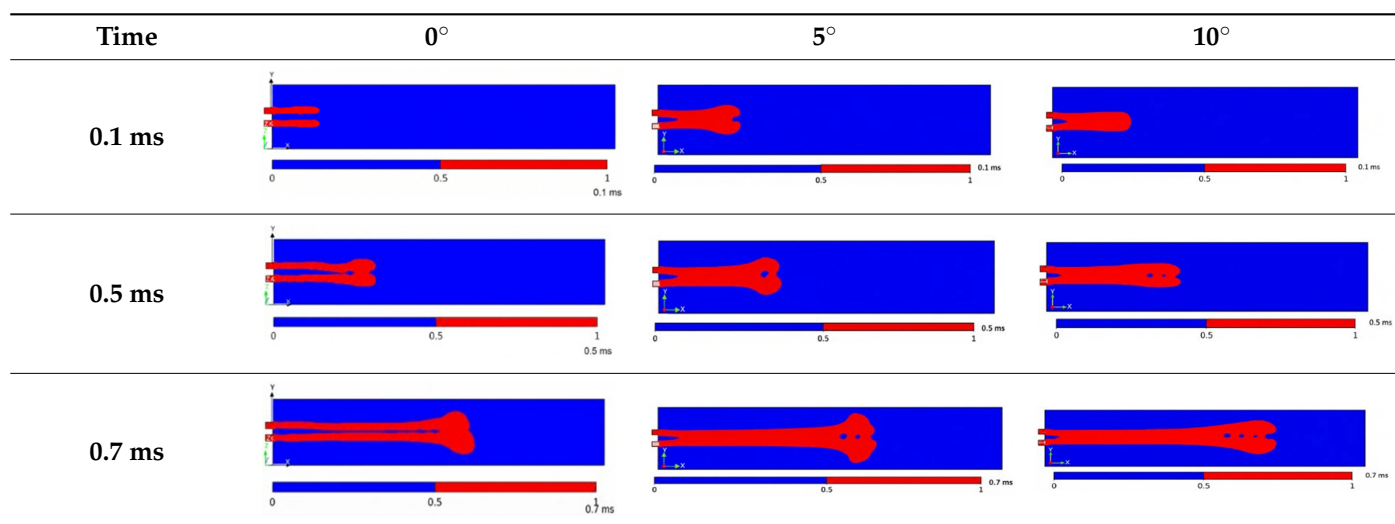


Figure 19. Velocity profiles with a vertical position for the third scenario.

3.4. Comparative Analysis of Injection Scenarios

Table 4 presents a comparative analysis of the volume fraction evolution across the three injection scenarios, highlighting their distinct impacts on fuel dispersion and potential combustion efficiency.

Table 4. Comparison of the three scenarios' volume fraction development.



Case 1 (0°): The neutral injection angle produces a highly concentrated jet with minimal radial dispersion. The fuel remains densely packed along the central axis, resulting in poor air entrainment and a high risk of an inhomogeneous air–fuel mixture, with consequent risk of incomplete combustion and lower energy efficiency.

Case 2 (5°): The moderately converging angle increases the resolved fuel–air interfacial area relative to the 0° case while retaining the highest axial penetration of the two converging cases, introducing a radial velocity component sufficient to drive ligament formation. The resultant increase in interfacial area arising from the partial overlap of the two jets' near-field dispersion envelopes downstream, rather than from any crossing of the converging orifice axes, is reported here as a precursor indicator of mixing potential; because combustion and mixture homogeneity were not directly modeled, this observation is not presented as direct evidence of a more homogeneous mixture, and the two-dimensional entrainment bias discussed previously tempers the reliability of the absolute values while preserving the validity of the relative, cross-angle comparison.

Case 3 (10°): The steep injection angle induces rapid radial dispersion and premature droplet detachment. However, the excessive radial momentum significantly sacrifices axial penetration. This creates a shallow and wide spray plume that fails to distribute fuel effectively throughout the combustion chamber volume.

Only the 0° and 5° configurations were assessed for the cone angle, decay rate, turbulent kinetic energy, and radial dispersion index shown in Table 5. No values are presented for the 10° configuration because these quantities were not calculated for that configuration within the parameters of the current analysis. Instead, the common metrics for all three jet angles—axial velocity decay, penetration distance, and radial half-width—are used to characterize the 10° situation.

Table 5. Quantitative jet-interaction metrics for the 0° and 5° configurations, illustrating the enhanced fuel–air interaction potential associated with jet intersection in the 5° (converging) configuration.

Metric	0°	5°	Increase
Cone angle	8.5	18.2	+114%
Decay rate	0.043	0.058	+35%
Turbulent kinetic energy	28.5	45.2	+58%
Radial dispersion index	0.12	0.28	+133%

The temporal analysis of Figure 20 elucidates the fundamental role of the injection angle in governing the trade-off between axial momentum and radial fuel dispersion. The 5° injection angle (Case 2) shows the best trade-off among the three tested cases, maintaining robust axial penetration while introducing a controlled radial velocity component that enhances air entrainment and mixing. In contrast, the 0° angle generates a high-velocity concentrated jet with poor dispersion, and the 10° angle excessively dissipates axial inertia through premature radial spread.

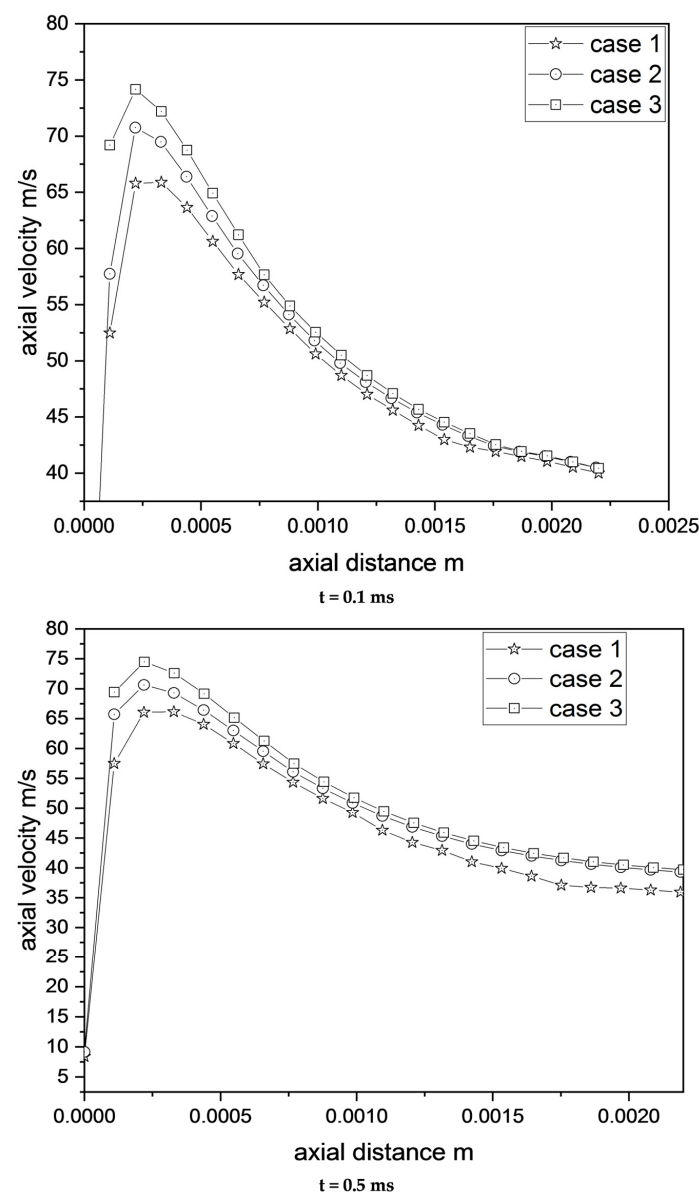


Figure 20. Cont.

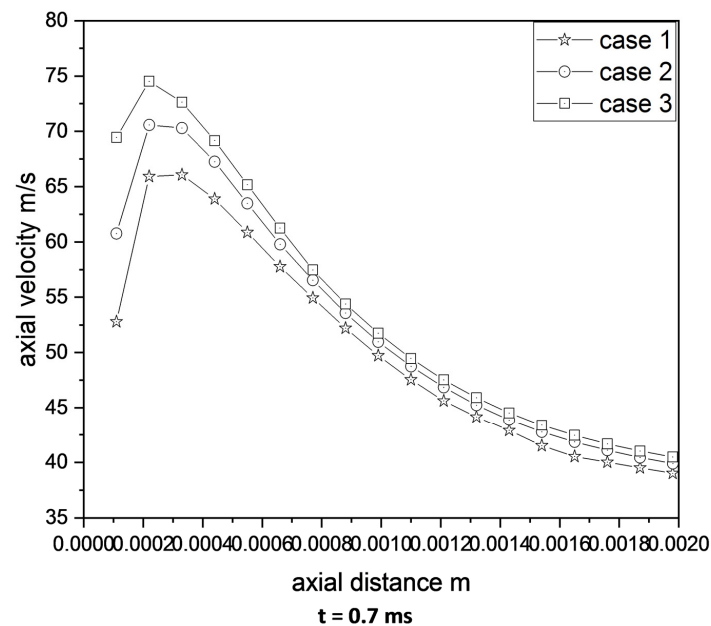


Figure 20. Axial velocity profiles at the same time for the three scenarios at $t = 0.1$ and 0.5 ms.

To move beyond a purely qualitative description, the key spray metrics were quantified for each configuration. The peak axial velocity at the orifice (position P_1) reached 185.2 m/s for the 0° case, 162.8 m/s for the 5° case, and 135.4 m/s for the 10° case. Along the monitored axis, the velocity decayed from P_1 to P_3 by approximately 28.5%/32.6%/37.1% for the 0° , 5° , and 10° cases, respectively. The corresponding axial penetration depth at $t = 0.7$ ms was 1.42/1.28/1.05 mm, while the radial spreading (half-width of the spray plume) was 0.085/0.205/0.325 mm. These figures confirm quantitatively that penetration decreases and radial spread increases monotonically with the convergence angle, with the 5° case retaining the largest fraction of axial penetration (90% of the 0° baseline) while already developing substantial lateral dispersion (a 141% increase in half-width compared with the 0° case). These half-width values should be interpreted as representative of the comparative trend across configurations: the higher axial velocity of Case 1 (0°) consistently corresponds to its greater axial penetration, and the monotonic increase in radial half-width with the angle is internally consistent with Figure 20 and Table 4. Particularly for Case 3 (10°), however, the magnitude of this radial spreading may be overestimated by the two-dimensional confinement effect discussed above, which structurally enhances apparent lateral dispersion; the absolute half-width values therefore warrant caution, with three-dimensional or axisymmetric verification recommended as future work. For direct traceability, the 0° case's highest peak axial velocity (185.2 m/s) corresponds to its largest axial penetration (1.42 mm) and smallest radial half-width (0.085 mm), while the 10° case's lowest peak axial velocity (135.4 m/s) corresponds to its smallest penetration (1.05 mm) and largest radial half-width (0.325 mm) in Figure 20 and Table 4, confirming that the trends described above are directly traceable to and consistent with those figures and tables.

The underlying physical mechanism rationalizing these trends is the partition of the injected kinetic energy between an axial component, which drives penetration, and a radial component, which drives interfacial instability growth and air entrainment. At 0° , almost all of the injected momentum remains axial, so the jet penetrates deeply, but the radial velocity is insufficient to amplify the Rayleigh–Plateau instabilities, delaying ligament formation and under-dispersing the fuel. As the convergence angle increases, a growing share of the kinetic energy is redirected radially, accelerating primary breakup and widening the plume. At 10° , this redistribution becomes excessive: the radial component

dominates at the expense of axial inertia, producing a shallow, over-dispersed plume. The 5° angle represents the optimum of this competition, injecting just enough radial momentum to trigger early ligament formation and air entrainment while preserving the axial penetration required to distribute fuel throughout the chamber volume.

Figure 21 presents the radial velocity profiles for the three injection scenarios at equidistant measurement points (P1, P2, P3). All configurations demonstrate a consistent pattern where radial velocity ascends to a distinct peak, undergoes a brief stabilization phase, and then decays rapidly as jet momentum dissipates. The comparative analysis confirms that the 5° injection angle generates the most favorable radial dispersion pattern, maintaining sufficient transverse momentum for effective air entrainment while preserving the energy necessary for complete chamber penetration.

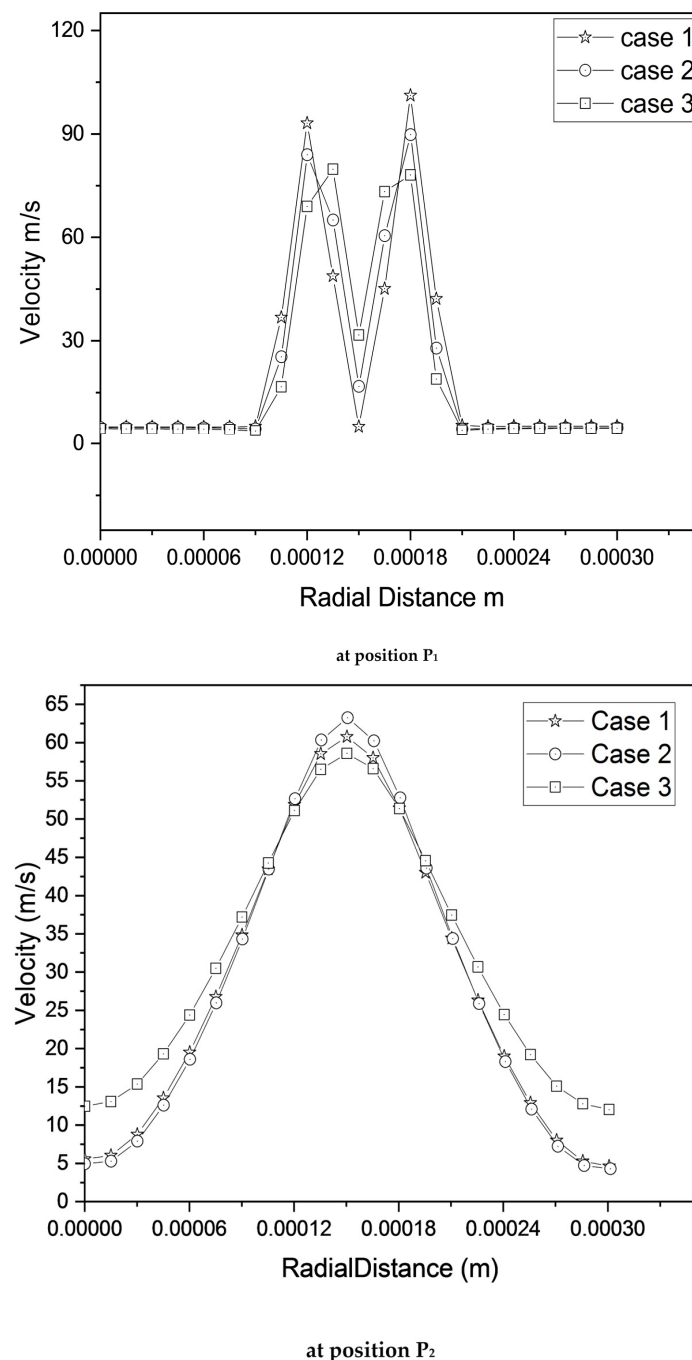
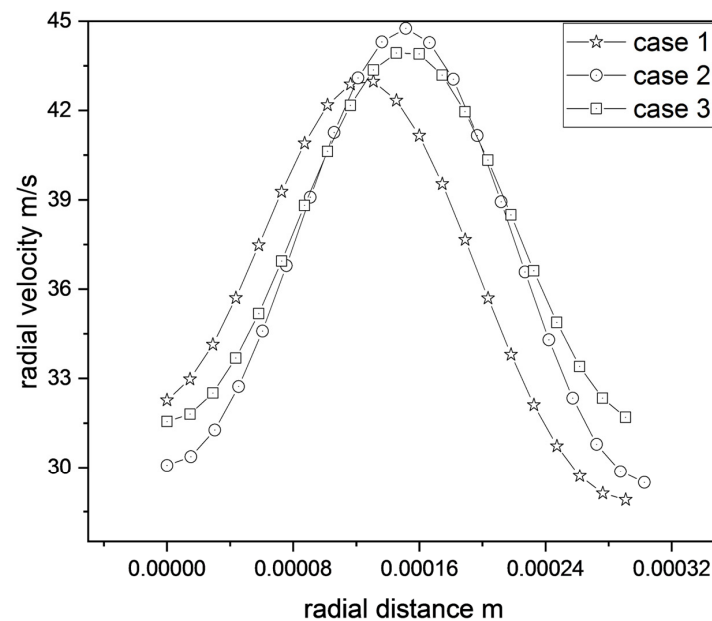


Figure 21. Cont.



at position P_3

Figure 21. Radial velocity profiles for the three positions and for the three scenarios studied.

3.5. Preliminary Conclusion

The comparative analysis identifies the 5° injection angle (Case 2) as the best trade-off among the three tested configurations. Under the studied cold-flow, near-nozzle conditions, it provides the most favorable compromise between axial and radial momentum, maximizing the resolved fuel–air interfacial area as a precursor indicator of mixing potential without the detrimental over-dispersion characteristic of the 10° angle. Consequently, the 5° angle is the preferred choice among those tested under the present conditions. Its implications for combustion efficiency and emissions are anticipated on physical grounds but were not modeled here, and they remain to be confirmed under evaporating, reacting, and full-chamber-scale conditions.

4. Conclusions

This study has systematically investigated the influence of injection angle on diesel spray dynamics using a validated computational framework based on the Volume-of-Fluid method with finite element discretization. Through detailed comparison of three injection configurations (0° , 5° , and 10°), the research demonstrates that minor angular modifications significantly affect primary atomization mechanisms, fuel distribution patterns, and velocity field development.

The comprehensive analysis identifies the 5° convergence angle as the best trade-off among the three tested configurations, providing a favorable balance between axial penetration depth and radial dispersion efficiency. This balanced approach promotes enhanced interfacial instability development and ligament-mediated breakup, leading to finer droplet formation. Because combustion, emissions, and air–fuel mixing quality were not directly modeled in the present study, the associated improvements in mixture homogeneity and the resulting benefits for combustion completeness and emission reduction are presented here only as plausible downstream implications and as motivation for future work, rather than as demonstrated outcomes of the simulations. In contrast, the 0° configuration produces an overly concentrated jet with inadequate dispersion and limited ligament formation, while the 10° angle exhibits excessive radial momentum at the expense of essential ax-

ial penetration. This interfacial-area enhancement stems from the partial overlap of the two jets' near-field dispersion envelopes downstream of the converging orifices, rather than from any crossing of the orifice axes themselves, and consistent with the guarded, precursor-indicator framing adopted throughout the preceding analysis, should not be read as a demonstrated improvement in fuel–air mixing quality.

These findings provide valuable practical insights for injector design optimization, indicating that angular adjustment is a promising approach with potential benefits for combustion efficiency without introducing additional system complexity. The research further validates the capability of advanced numerical simulation techniques to capture complex multiphase flow phenomena, including ligament formation and primary atomization mechanisms, establishing a reliable methodology for future injector development.

The present conclusions are restricted to cold-flow, near-nozzle primary breakup under isothermal conditions; extension to elevated-temperature, evaporating, and full-chamber-scale conditions is a necessary next step before these findings can be extrapolated to real combustion chambers. A continuous angular optimization, rather than the discrete comparison of three configurations reported here, is likewise left for future work. Future investigations should consider extending this methodology to analyze transient injection effects, multi-hole nozzle geometry variations, and spray performance with alternative renewable fuels to further advance sustainable combustion technologies and support the transition toward carbon-neutral transportation systems.

Author Contributions: Conceptualization, S.T. and F.Z.S.; methodology, E.E.E. and S.T.; software, S.T. and H.Y.; validation, S.T., F.Z.S. and F.D.; formal analysis, S.T.; investigation, E.E.E., S.T. and M.Z.; writing—original draft preparation, S.T.; writing—review and editing, F.Z.S. and F.D.; visualization, H.Y.; project administration, E.E.E.; supervision, M.Z. All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Data Availability Statement: The data presented in this study are available on request from the corresponding author.

Conflicts of Interest: The authors declare no conflicts of interest.

Nomenclature

$\frac{D}{Dt}$	material (substantial) derivative
$\vec{g}$	gravity
P	pressure
t	time
u, v, w	velocity components in Cartesian coordinates x, y, z
$\vec{v}$	velocity vector
$\mathbf{X}$	spatial position vector
$\frac{\partial}{\partial t}$	partial derivative with respect to time
∇	gradient operator (del operator)
$\alpha(\mathbf{X}, t)$	indicator function/liquid phase volume fraction
μ	dynamic viscosity
ρ	density
Ω	computational domain
Ω_l	liquid phase domain
Ω_s	solid phase domain

References

- Ochiai, N.; Ishimoto, J.; Arioka, A.; Yamaguchi, N.; Sasaki, Y.; Furukawa, N. *Integrated Computational Study for Total Atomization Process of Primary Breakup to Spray Droplet Formation in Injector Nozzle*; SAE Technical Paper 2016-01-2202; SAE International: Warrendale, PA, USA, 2016. [\[CrossRef\]](#)
- Yang, S.; Yi, P.; Habchi, C. Real-fluid injection modeling and LES simulation of the ECN Spray A injector using a fully compressible two-phase flow approach. *Int. J. Multiph. Flow* **2020**, *122*, 103145. [\[CrossRef\]](#)
- Schillaci, E.; Antepara, O.; Balcazar, N.; Rigola, J.; Oliva, A. A numerical study of liquid atomization regimes by means of conservative level-set simulations. *Comput. Fluids* **2019**, *179*, 137–149. [\[CrossRef\]](#)
- Salvador, F.J.; Ruiz, S.; Cialesi-Esposito, M.; Blanquer, I. Analysis on the effects of turbulent inflow conditions on spray primary atomization in the near-field by direct numerical simulation. *Int. J. Multiph. Flow* **2018**, *102*, 49–63. [\[CrossRef\]](#)
- Hirt, C.W.; Nichols, B.D. Volume of fluid (VOF) method for the dynamics of free boundaries. *J. Comput. Phys.* **1981**, *39*, 201–225. [\[CrossRef\]](#)
- Liu, Z.; Li, Z.; Liu, J.; Wu, J.; Yu, Y.; Ding, J. Numerical study on primary breakup of disturbed liquid jet sprays using a VOF model and LES method. *Processes* **2022**, *10*, 1148. [\[CrossRef\]](#)
- Kong, L.; Lan, T.; Chen, J.; Wang, K.; Sun, H. Breakup processes and droplet characteristics of liquid jets injected into low-speed air crossflow. *Processes* **2020**, *8*, 676. [\[CrossRef\]](#)
- Shinjo, J. Recent advances in computational modeling of primary atomization of liquid fuel sprays. *Energies* **2018**, *11*, 2971. [\[CrossRef\]](#)
- Liu, C.; Xi, X.; Xie, R.; Liu, H.; Li, W.; Jia, M.; Gu, H. Numerical study on the effects of eccentric nozzles on spray evolution using a hybrid method. *Int. J. Multiph. Flow* **2024**, *178*, 104910. [\[CrossRef\]](#)
- Jin, T.; Wang, C.; Moro, A.; Roell, A.; Wu, X.; Luo, F. The influence of needle eccentric motion on injection and spray characteristics of a two-layered eight-hole diesel injector. *Proc. Inst. Mech. Eng. Part D J. Automob. Eng.* **2024**, *238*, 3308–3327. [\[CrossRef\]](#)
- Chen, P.; Xu, R.; Liu, Z.; Liu, J.; Zhang, X. Homogeneous field measurement and simulation study of injector nozzle internal flow and near-field spray. *Processes* **2023**, *11*, 2533. [\[CrossRef\]](#)
- El Marnissi, Y.; Hwang, J. Microscopic imaging on diesel spray and atomization process. *Processes* **2024**, *12*, 359. [\[CrossRef\]](#)
- Pratama, R.H.; Huang, W.; Gao, Y. Investigation of diesel spray characteristics from injectors with asymmetric hole distribution. *Fuel* **2025**, *385*, 134068. [\[CrossRef\]](#)
- Cao, T.; Jin, J.; Qu, Y.P. Cavitation phenomenon and spray atomization in different types of diesel engine nozzles: A systematic review. *SAE Int. J. Engines* **2024**, *17*, 743–766. [\[CrossRef\]](#)
- Hu, B.; He, Z.; Li, C.; Deng, Y.; Guan, W.; Zhang, L.; Guo, G. Study of the effect of cavitation flow patterns in diesel injector nozzles on near-field spray atomization characteristics using a LES-VOF method. *Int. J. Multiph. Flow* **2024**, *179*, 104791. [\[CrossRef\]](#)
- Cengiz, C.; Unverdi, S.O. A CFD study on the effects of injection timing and spray inclusion angle on performance and emission characteristics of a DI diesel engine. *Energies* **2023**, *16*, 2861. [\[CrossRef\]](#)
- Shu, J.; Fu, J.; Liu, J.; Ma, Y.; Wang, S.; Deng, B.; Zeng, D. Effects of injector spray angle on combustion and emissions characteristics of a natural gas-diesel dual fuel engine based on CFD coupled with reduced chemical kinetic model. *Appl. Energy* **2019**, *233*, 182–195. [\[CrossRef\]](#)
- Jin, T.; Sun, Y.; Wang, C.; Moro, A.; Wu, X.; Luo, F. The Influence of Needle Eccentric Motion on Hole-to-Hole Injection Characteristics of a Two-Layered Eight-Hole Diesel Injector. *ASME. J. Eng. Gas Turbines Power.* **2021**, *143*, 101005. [\[CrossRef\]](#)
- Wang, Y.; Liu, Q.; Zhang, H. Numerical investigation of spray atomization in asymmetric multi-hole diesel injectors. *Fuel* **2024**, *366*, 131350. [\[CrossRef\]](#)
- Ménard, T.; Beau, P.A.; Tanguy, S.; Demoulin, F.X.; Berlemont, A. Numerical Jet Atomization: Part I—DNS Simulation of Primary Break Up. In Proceedings of the Fluids Engineering Division Summer Meeting, Miami, FL, USA, 17–20 July 2006; Volume 47500, pp. 547–554. [\[CrossRef\]](#)
- Lafaurie, B.; Nardone, C.; Scardovelli, R.; Zaleski, S.; Zanetti, G. Modelling merging and fragmentation in multiphase flows with SURFER. *J. Comput. Phys.* **1994**, *113*, 134–147. [\[CrossRef\]](#)
- Bravo, L.; Xue, Q.; Som, S.; Powell, C.; Kweon, C.-B.M. Fuel Effects on Nozzle Flow and Spray Using Fully Coupled Eulerian Simulations. In Proceedings of the ASME 2015 Power Conference, San Diego, CA, USA, 28 June–2 July 2015. [\[CrossRef\]](#)

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.