



Sprey Nozullerde Konik Nozul Tasarımının Aerodinamik Etkilerinin Hesaplamalı Akışa Dayalı Analizi

Fuat TAN^{1*} , Alp Eren DEDE² 

^{1,2}Makine Mühendisliği Bölümü, Mühendislik Fakültesi, Balıkesir Üniversitesi, Balıkesir, Türkiye.
¹fuattan@balikesir.edu.tr, ²alperendede46@gmail.com

Geliş Tarihi: 04.03.2025
Kabul Tarihi: 16.06.2025

Düzeltilme Tarihi: 10.06.2025

doi: <https://doi.org/10.62520/fujece.1651099>
Araştırma Makalesi

Alıntı: F. Tan ve A. E. Dede, "Sprey nozullerde konik nozul tasarımının aerodinamik etkilerinin hesaplamalı akışa dayalı analizi", Fırat Üni. Deny. ve Hes. Müh. Derg., vol. 4, no 3, pp. 474-489, Ekim 2025.

Öz

Bu çalışmada, spreyleme nozülü ucundaki püskürtme davranışı hesaplamalı akışkanlar dinamiği yöntemi ile parametrik analiz yapılarak incelenmiştir. Bu amaçla, Ansys Fluent ticari yazılımı kullanılarak deneysel tasarım yöntemi olan Yanıt Yüzey Yönteminin (RSM) oluşturduğu parametrik tablo ile analizler yürütülmüştür. Giriş değişkenleri olarak nozul çıkış açısı, nozul çıkış çapı ve nozul çıkış hızı seçilmiştir. Buna bağlı olarak, kontrol hacmindeki püskürtme davranışını gösteren akış hızı ve basınç dağılımı ile türbülans kinetik enerjisi ve Eddy viskozitesi incelenerek karşılaştırılmıştır. Giriş değişkenleri için nozul çıkış açısı 145°-165°, çıkış çapı 0,6-0,8 mm ve nozul çıkış hızı ise 10-16 m/s olarak sırasıyla minimum ve maksimum değer aralığında alınmıştır. Çalışma sonunda maksimum akış hızı olan 1187.34 m/s değerine; nozul çıkış açısı 155°, nozul çıkış çapı 0.6 mm ve çıkış hızı 16 m/s değerindeyken ulaşılmıştır. Maksimum basınç değerine; 145° nozul açısı ve 0.6 mm nozul çapı ile 614029 Pa değerinde ulaşılrken, maksimum türbülans kinetik enerjisi olan 24260 J değerine ise 145° nozul çıkış açısı ve 13 m/s nozul çıkış hızı ile ulaşılmıştır. Eddy viskozitesi için de maksimum değer olan 0.02339 Pa·s değerine, 155°lik nozul çıkış açısı ve 16 m/s nozul çıkış hızı değerinde ulaşılmıştır. Analiz sonucu elde edilen sayısal bulgular ile bir nozulun nasıl tasarlanması gerektiği konusunda öneriler yapılmıştır. RSM yöntemi ile elde edilen bulgulara göre, nozul açısı azaltıldığında akış hızının arttığı buna karşın türbülans davranışının güçlendiği görülmüştür. Bu nedenle özellikle bir nozul tasarımı ile yapılacak çalışmalarda, bu konunun sayısal bir yöntemle ele alınması ve sonuçlardan elde edilen bazı önerilerle sunulması gerektiği görülmüştür.

Anahtar kelimeler: Aerodinamik, CFD, Nozul tasarımı, RSM, Püskürtme

*Yazılan yazar



İntihal Kontrol: Evet – Turnitin

Şikayet: fujece@firat.edu.tr

Telif Hakkı ve Lisans: Dergide yayın yapan yazarlar, CC BY-NC 4.0 kapsamında lisanslanan çalışmalarının telif hakkını saklı tutar.



Computational Flow Analysis of the Aerodynamic Effects of Conical Nozzle Design in Spray Nozzles

Fuat TAN^{1*} , Alp Eren DEDE² ,

^{1,2}Department of Mechanical Engineering, Faculty of Engineering, Balikesir University, Balikesir, Türkiye.

¹fuattan@balikesir.edu.tr, ²alperendede46@gmail.com

Received: 04.03.2025

Revision: 10.06.2025

doi: <https://doi.org/10.62520/fujece.1651099>

Accepted: 16.06.2025

Research Article

Citation: F. Tan and A. E. Dede, "Computational flow analysis of the aerodynamic effects of conical nozzle design in spray nozzles", *Firat Univ. Jour. of Exper. and Comp. Eng.*, vol. 4, no 3, pp. 474-489, October 2025.

Abstract

In this study, spray behavior in the nozzle was analyzed using the computational fluid dynamics (CFD) method through parametric simulations. Accordingly, parametric analyses were conducted using the Response Surface Methodology (RSM) in the commercial software ANSYS Fluent. The input variables were the nozzle outlet angle, the outlet diameter, and the nozzle outlet velocity. The flow velocity and pressure distribution in the control volume, as well as turbulence kinetic energy and eddy viscosity, were compared. For the input variables of the nozzle output angle the minimum value of 145°-165°, the width of the output orifice 0.6-0.8 mm and the outlet velocity of the nozzle 10-16 m/s were sequentially taken as the minimum and maximum values. The maximum flow velocity (1187.34 m/s) occurred at 155°, 0.6 mm, and 16 m/s. The highest pressure (614029 Pa) was recorded at 145° and 0.6 mm, while turbulence kinetic energy peaked at 24260 J at 145° and 13 m/s. The maximum eddy viscosity (0.02339 Pa·s) was found at 155° and 16 m/s. These results provide practical recommendations for optimizing nozzle design using CFD and RSM. The analyses revealed that reducing the nozzle angle increases flow velocity but simultaneously intensifies turbulence, highlighting the trade-off between efficiency and stability. Overall, the study emphasizes the importance of computational methods in nozzle design and offers clear guidelines for achieving improved spray performance.

Keywords: Aerodynamic, CFD, Nozzle design, RSM, Spraying

*Corresponding author

Plagiarism Checks: Yes – Turnitin

Complaints: fujece@firat.edu.tr

Copyright & License: Authors publishing with the journal retain the copyright to their work licensed under the CC BY-NC 4.0

1. Introduction

Nozzle spray systems are widely used in various industrial applications, from fuel injection to spray coating. Spray performance is a key factor affecting efficiency and energy consumption. Therefore, optimizing spray systems is essential for enhancing performance and reducing costs. The impact of nozzle geometry on spray performance has been explored in numerous studies. Parameters such as nozzle angle, outlet diameter and outlet velocity are particularly influential factors directly affecting spray quality. The fuel injection process, a key application area for nozzle spray systems, plays an essential role in combustion and pollutant formation in diesel engines. Therefore, accurate management of this process is necessary to achieve reliable engine performance and emission predictions.

Diesel spray modeling remains a challenging task due to complex phenomena such as primary atomization [1] and injector cavitation [2], which are not yet fully understood. The Discrete Droplet Method (DDM) has been widely used for over 30 years in diesel spray modeling for practical design applications [3]. The Discrete Droplet Method (DDM) emphasizes droplet dynamics and makes significant efforts to accurately represent breakup and collision processes. However, in the high pressure environments of diesel engines, turbulent mixing may be the dominant factor. For example, experimental results by Siebers [4-6] indicated that mass and energy transfer at droplet surfaces and during atomization were not limiting steps for evaporation in direct-injection (DI) diesel sprays.

To ensure the stability of diesel sprays, grid sizes generally need to be larger than the injector diameter, which can result in insufficient resolution of flow structures [7-8]. In the Eulerian model, the scope of the atomization process is calculated using an interface surface density equation, eliminating the need to assume a specific shape for liquid particles. This model provides a natural approach to incorporate the flow effects of injector geometry into spray calculations [9].

Another application area for nozzles, water sprays, is widely used in fire control and suppression. Numerous experimental programs have been developed to study the dynamics of these sprays and their effectiveness in various fire scenarios, such as passing through a fire flame or interacting with a ceiling vent [10-11]. The main complexity in water spray simulations is the breakup of the liquid into small droplets, which limits the ability to predict sprinkler atomization accurately [12].

The limitations in modeling flow patterns within spray dryers have led to the use of more fundamental approaches, specifically Computational Fluid Dynamics (CFD) packages, to better represent these patterns. This trend has been enabled by the development of powerful workstations at a more accessible cost. Crowe was the first researcher to apply this approach to spray dryers [13]. The CFD program predicted that the most likely areas for wall deposits are the circular zone on the ceiling, where a small backflow vortex gathers medium-sized droplets and a region below the atomizer where larger particles tend to accumulate. These predictions align with practical experience and highlight the industrial applications of CFD programs. Oakley et al. provided a strong example of the predictive capability of these techniques [14].

Nasal drug delivery offers new opportunities for administering systemic medications traditionally given orally or intravenously. Several Computational Fluid Dynamics (CFD) studies have been reported on the deposition regions of nasal spray drug delivery [15-16]. However, the initial droplet conditions used in these studies do not realistically reflect the behavior of atomized spray droplets from a nasal spray drug delivery device. Enhancing current CFD simulations for more accurate virtual drug delivery predictions could be achieved by implementing more realistic initial droplet conditions, crucial for downstream behavior.

Recent studies on drug delivery through the oral cavity using metered-dose inhalers (MDI) and dry powder inhalers (DPI) have used experimental measurements to determine droplet size distributions post-breakup, applying these as initial conditions for spray nozzle simulations [17-18]. Zhang et al. investigated the effects of nozzle hole diameter and injection pressure on the spray angle and velocity distribution of jet fuel injected from pressure-swirl nozzles. They found that above the critical pressure of 1.5 MPa, the spray angle became independent of pressure and nozzles with a length to diameter ratio of 2 provided wider spray angles [19]. Önen et al. observed that as the nozzle exit and throat diameters increased, the impact area on the target

material also increased [20]. It was determined that momentum flow varied depending on the nozzle geometry and the difference increased with higher injection pressure [21]. In another study, Du et al. conducted Computational Fluid Dynamics (CFD) simulations using the Discrete Phase Model (DPM). They revealed that nozzle spray shape (circular, elliptical Type A and B) had a greater effect on cavitation and spray characteristics than fuel type in hybrid biofuel and diesel blends. Elliptical nozzles provided a wider spray, while circular nozzles exhibited deeper penetration [22]. Wang et al. found that in high-pressure fuel injection systems, nozzle geometry (chamber volume, hole curvature and inclination angle) significantly influenced spray penetration and Sauter mean diameter [23].

Recent research has drawn attention to the considerable role that the nozzle design plays in the characteristics of the spray. An example of this is a research paper [24] that brought the comparison between the twin fluid pressure nozzle and the blade designed nozzle to the table and the results showed that the latter delivered better atomization and the sprays were more evenly distributed due to increased turbulence generation. Another study aims to examine the effects of aerodynamic modifications of pneumatic nozzles on droplet size distribution, revealing that it is possible to achieve higher efficiency in atomization processes through specific geometric changes [25]. Moreover, one research assessed the behavior of the flow through the nozzle in conjunction with the characteristics of spray, which showed that having bigger diameters of the nozzle leads to longer reach and better spray targeting, both of which are crucial for precise applications [26].

The purpose of this study is to examine the effects of nozzle geometry on spray performance through both numerical simulations and statistical modeling. Numerical simulations were conducted using ANSYS Fluent to evaluate factors influencing spray performance. Additionally, using the Response Surface Methodology (RSM), a design of experiments approach, the effects of input variables on response variables were modeled and optimized.

2. Material and Method

In this study, numerical simulations of nozzle spray systems were conducted using ANSYS Fluent software. CFD analyses were performed to examine parameters such as flow velocity, pressure and turbulence kinetic energy.

The realizable k-ε turbulence model is used for high-Reynolds-number flows and simulates the turbulent gas phase induced by droplet momentum [27]. While this model is extensively used, it often overestimates turbulent viscosity and may not be suitable for low pressure applications where induced flow by spray droplets in stagnant air is present. Fogliati et al. demonstrated good agreement with experimental results by applying the realizable k-ε turbulence model in simulations of paint sprays at low injection pressures [28]. Therefore, the realizable k-ε turbulence model is used in this study. The governing equations for the fluid phase are as follows:

$$\frac{\partial}{\partial x_j} (\rho_g u_j^g) = 0 \tag{1}$$

$$\rho_g \frac{\partial u_i^g}{\partial t} + \rho_g u_j^g \frac{\partial u_i^g}{\partial x_j} = -\frac{\partial p_g}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu_g \frac{\partial u_i^g}{\partial x_j} \right] + \frac{1}{\rho_g} M_p \tag{2}$$

In these equations, the external force refers to the drag force caused by the interaction with the droplet phase. The symbols ρ_g and μ_g represent the gas phase density and effective viscosity, respectively. M_p is the momentum change, and u is the flow velocity [29].

The model assumes a linear stability analysis for a two-dimensional, viscous, incompressible liquid layer of thickness ($2h$) moving at a relative velocity (U) within an inviscid, incompressible gas medium. An infinitesimal disturbance spectrum is defined as follows [30]:

$$\eta = \eta_0 e^{ik_w x + \omega t} \tag{3}$$

The total velocity (U) is determined in relation to the injection pressure:

$$U = k_v \sqrt{\frac{2\Delta P}{\rho_l}} \quad (4)$$

The mass flow rate is given by the following equation:

$$\dot{m} = \pi \rho U \cos \theta h (dn - h) \quad (5)$$

2.1. Model

The three dimensional model used for aerodynamic simulations was created using the Design Modeler CAD program. In the model, the total dimensions ($L1 + L2 + L3$) is fixed at 20 mm. Therefore, when the nozzle exit angle (α) changes, the dimensions $L1$ and $L2$ will also vary. The dimensions $D1$ and $D2$ are independent of α changes. The range values for the other dimensions are provided in Table 3.

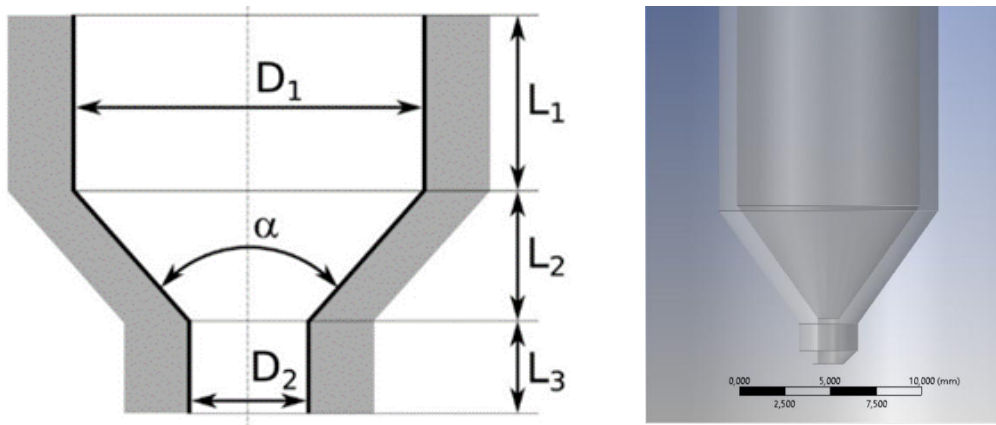


Figure 1. 2D and 3D nozzle model dimensions

Table 1. Model dimension values

Dimension	Unit	Value
α	degree	145-165
D1	mm	5
D2	mm	0.6-1
$L1 + L2 + L3$	mm	20

2.2. Mesh

After defining the nozzle's design parameters, the system must be divided into numerous small parts, called "mesh" in the numerical analysis software for accurate analysis. As the number of mesh elements increases, the system is divided into smaller parts, yielding more precise results in numerical analysis when quality ratios are met. The mesh structure of the model is shown in Figure 2.

In order to ensure that the CFD results are not affected by the size of the mesh and to ascertain the numerical precision, a mesh independence study was conducted. The three distinct types of the meshes used and vital output parameters such as the velocity at the outlet and the kinetic energy of turbulence were compared. It was discovered that the relative changes between medium and fine mesh outputs are less than 2% and this denotes mesh independent behavior. Thus, the mesh of 38,447 elements and 7,760 nodes was preferred because it was a compromise between computational efficiency and solution

accuracy. The element size set to 4.95 mm and a tetrahedral element structure was used. Accurate data in the analysis is directly linked to the mesh structure and its properties, so the meshing process must be conducted carefully and precisely.

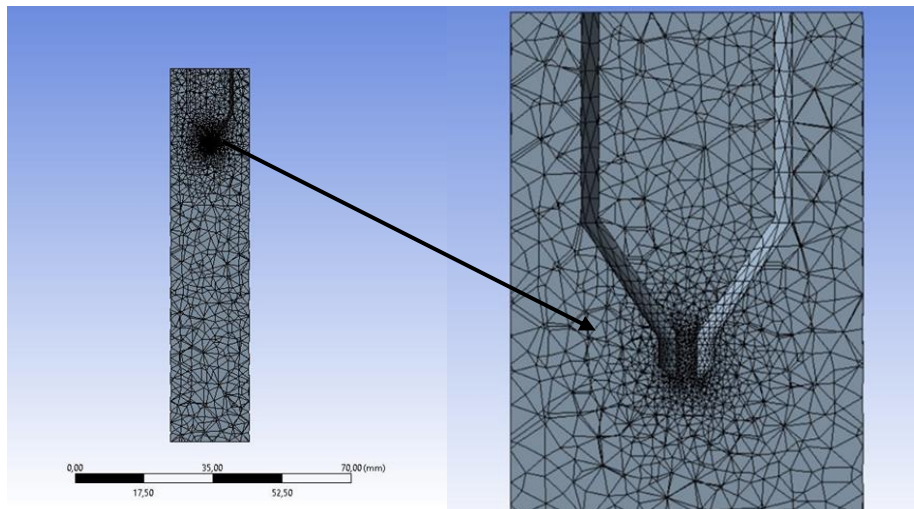


Figure 2. Mesh structure of the model

2.3. Solution

The analyses were conducted on an i7 core processor using ANSYS Fluent 2020 R2. Numerical simulations were performed on the nozzle spraying model with defined dimensions. A pressure-based solver was used and the analysis was steady-state. The turbulence model applied was the standard k- ϵ model. For boundary conditions, a velocity inlet was defined at the nozzle entry and a pressure outlet for the control volume exit. A velocity inlet was defined at the nozzle entrance with a velocity of 10, 13 and 16 m/s along with a pressure outlet which was set to 0 Pa at the domain exit. The turbulence at the inlet was determined by an intensity of 5% and a hydraulic diameter based on the inlet geometry. A no-slip condition was applied to all the walls and a wall model was used to treat the near the wall area. Air was used as the flow and aluminum was the nozzle material. Aluminum properties are presented in Table 2. The coupled method was chosen for pressure velocity coupling, with a convergence criterion of 0.001. Due to the turbulent nature of the flow, Hybrid initialization was applied.

Table 2. Physical properties of Aluminum used in the CFD Model

Physical Property	Aluminum
Density (Kg/m ³)	2719
Specific Heat Capacity (J/kgK)	871
Thermal Conductivity (W/mK)	202.4
Thickness (m)	0.00125

To confirm the correctness of the regression equations derived from the RSM analysis, residual plots and diagnostic tests were conducted. The probability plots of the normal distribution of residuals supported the claim that the errors had a normal distribution, whereas the plots of the residuals against the predicted values revealed no evident patterns that would indicate homoscedasticity. Moreover, all studentized residuals remained within the ± 2 range, which corroborated the view that there were no outliers.

2.4. Experimental design method

The CFD analyses were conducted to examine and mathematically model parameters such as flow velocity, pressure distribution, turbulence kinetic energy and Eddy viscosity. For this purpose, the Response Surface Methodology (RSM) was applied using the Design Expert software, with a total of 12 analyses performed to model the relationships among input variables. These analyses were recorded in the experimental design results table (Table 3), followed by regression analysis (Table 4). Table 3 shows the results of the RSM based experimental design. The nozzle angle, outlet diameter and inlet velocity were combined with the corresponding CFD outputs: outlet velocity, pressure, turbulence kinetic energy and eddy viscosity. The outputs can help understand the different aspects of the flow. The regression results, with a reliability close to 99%, demonstrate the accuracy of the derived mathematical equations. Furthermore, residual plots and ANOVA output supported the idea that, these models were statistically significant and appropriate for the prediction of flow characteristics. In this research, the impacts of input variables on the spraying characteristics were represented by means of Response Surface Methodology (RSM). Due to its regression-oriented framework, RSM allows not only the assessment of main effects but also of two-factor interactions. Hence, the influence of variables was examined without the need to perform an additional sensitivity analysis. While variance-based techniques suggested in the literature are highly efficient in investigating higher-order interactions, the RSM method was accepted as appropriate given the range of parameters and the nature of the model of this study. Although the current design successfully captured the main effects and two factor interactions, future studies could assist in improving predictive performance through a selection of more center points or higher order interactions. Within the study, the Box-Behnken method in the program was used to present the effects of nozzle exit diameter, nozzle angle and nozzle outlet speed on velocity, pressure, turbulence kinetic energy and Eddy viscosity results on the midplane (plane 1) in the form of 2D contour plots. In the experimental design process, each variable was tested at three levels. The coefficients of regression for all response variables are summarized in Table 4. Here, A refers to the nozzle angle, B to the outlet's diameter and C to the inlet's velocity. An interaction effect such as BC is also included. These models illustrate both the single and joint impacts of design parameters and thus they contribute to the RSM method's prediction capability.

Table 3. Experimental design results table

Angle (degree)	Outlet diameter (mm)	Nozzle velocity (m/s)	Outlet velocity (m/s)	Pressure (Pa)	Turbulence Kinetic Energy (m ² /s ²)	Eddy Viscosity (Pa.s)
145	0.6	13	991.565	614029	24260	0.0195643
165	0.6	13	954.84	558683	12348.4	0.0160239
145	1	13	355.44	76540	1947.28	0.0148878
165	1	13	340.354	69755	1452.07	0.0135086
145	0.8	10	427.741	113965	3923.34	0.0121054
165	0.8	10	411.601	103394	2181.52	0.013572
145	0.8	16	683.825	288328	8389.56	0.0167124
165	0.8	16	658.646	262539	6004.8	0.0180066
155	0.6	10	745.733	347017	10790.2	0.0172133
155	1	10	267.097	43353	1061.79	0.0114878
155	0.6	16	1187.34	877943	22780.3	0.0233976
155	1	16	427.45	109672	2478.35	0.0162995

Table 4. Mathematical model

	Intercept	A	B	C	BC
Velocity Plane 1	0.0434507		0.0108803	-0.00510677	
Pressure Plane 1	0.00254068		0.00123055	-0.00058395	-0.000288207
Tke Plane 1	0.0160458	0.00169901	0.00850103	-0.00339448	-0.00190008
Eddy Plane 1	8.0026		0.5942	-0.633352	

3. Results

In this study, the main goal of computational analyses is quantitative evaluation of characteristic flow parameters, which are essential to the description of the spray behavior. In this regard, emphasis has been given to variables such as exit velocity, pressure distribution, turbulence kinetic energy, and eddy viscosity on contour maps. (The overall results of these parameters on plane 1 are presented in Figures 7–10). Since the scope of the study is focused on the understanding of aerodynamic influence through these basic parameters, the qualitative flow visualizations such as streamlines, vorticity fields, or velocity vectors have not been included. However, the presented data and analysis results provide a sufficient level of representation for the assessment of three-dimensional flow characteristics.

3.1. Velocity

It was observed that flow velocity distribution varies based on the nozzle's geometric dimensions and exit velocity. The highest flow velocity, 1187.3 m/s, was measured at a 155° nozzle angle, 0.6 mm exit width and 16 m/s inlet velocity, as seen in Figure 3. Increased flow velocity with smaller nozzle diameters is expected. At these parameter values, pressure also reached a high level of 877943 Pa and turbulence kinetic energy reached its peak at 22780.3 J.

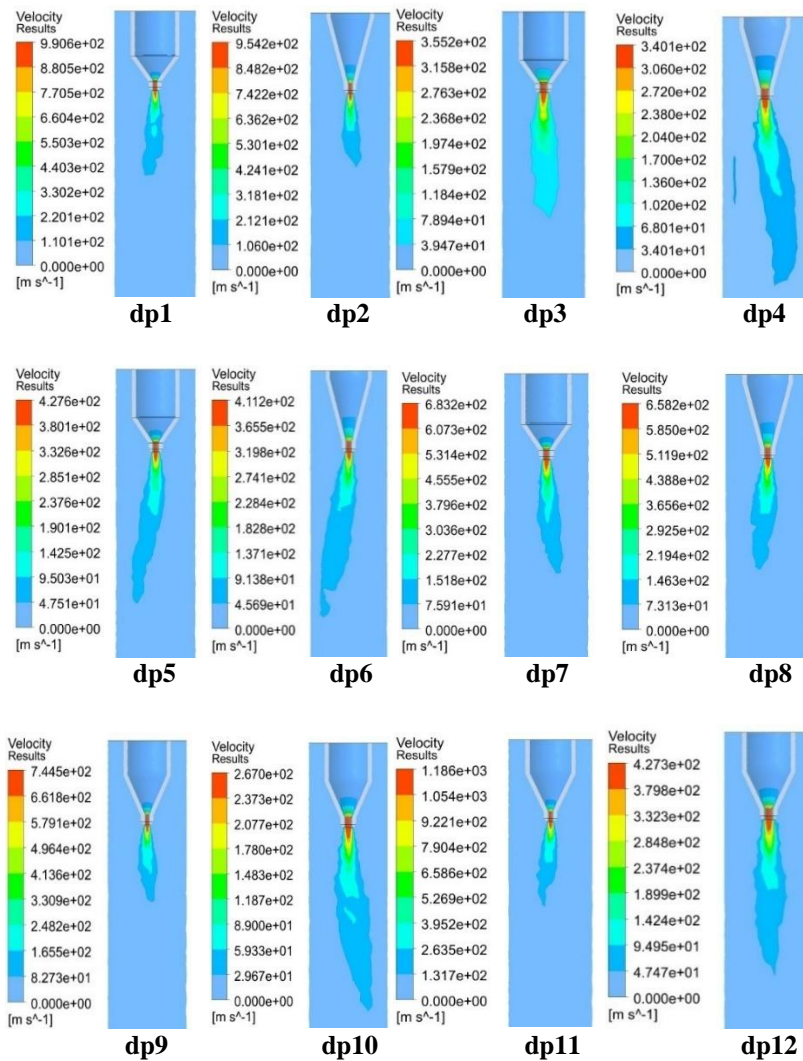


Figure 3: Velocity at plane 1

3.2. Pressure

Analysis revealed that pressure values at the nozzle exit increased significantly with lower outlet angles and smaller exit diameters. The highest pressure, 614029 Pa, was achieved at a 145° nozzle angle, 0.6 mm exit diameter and 13 m/s outlet velocity, as shown in Table 3. The pressure distribution at the nozzle exit varies considerably based on the combinations of nozzle geometry and inlet velocity. In narrow angle designs, the flow compression increases pressure; for this design, this occurs at 145°, where turbulent flow is also higher compared to other configurations.

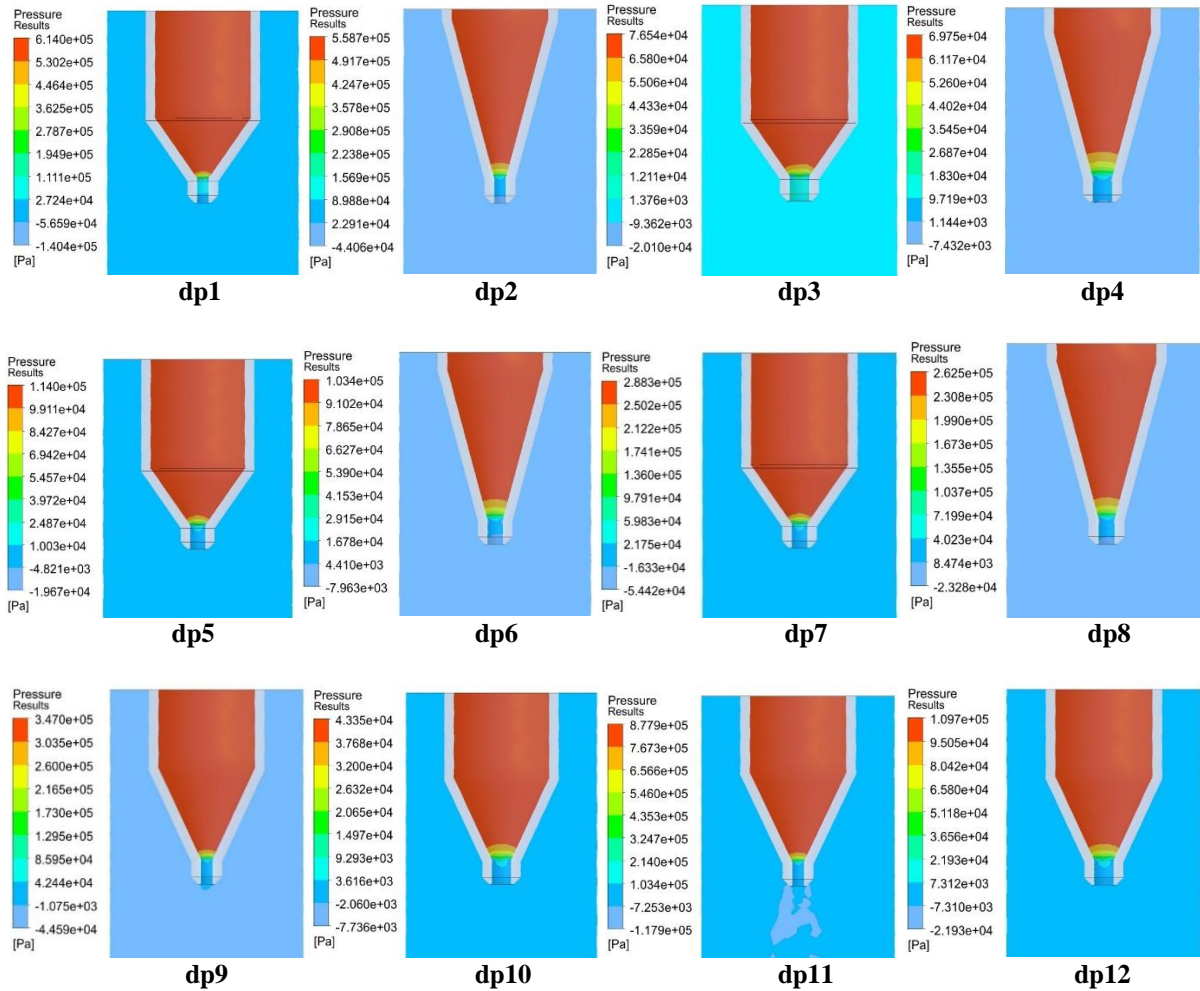


Figure 4. Pressure at plane 1

3.3. Turbulence kinetic energy

The findings showed that designs with narrower nozzle diameters and higher outlet velocities generated more turbulence kinetic energy. The highest turbulence kinetic energy, 24260 J, was calculated at a 145° angle, 0.6 mm nozzle exit diameter and 13 m/s inlet velocity, as shown in Table 3.

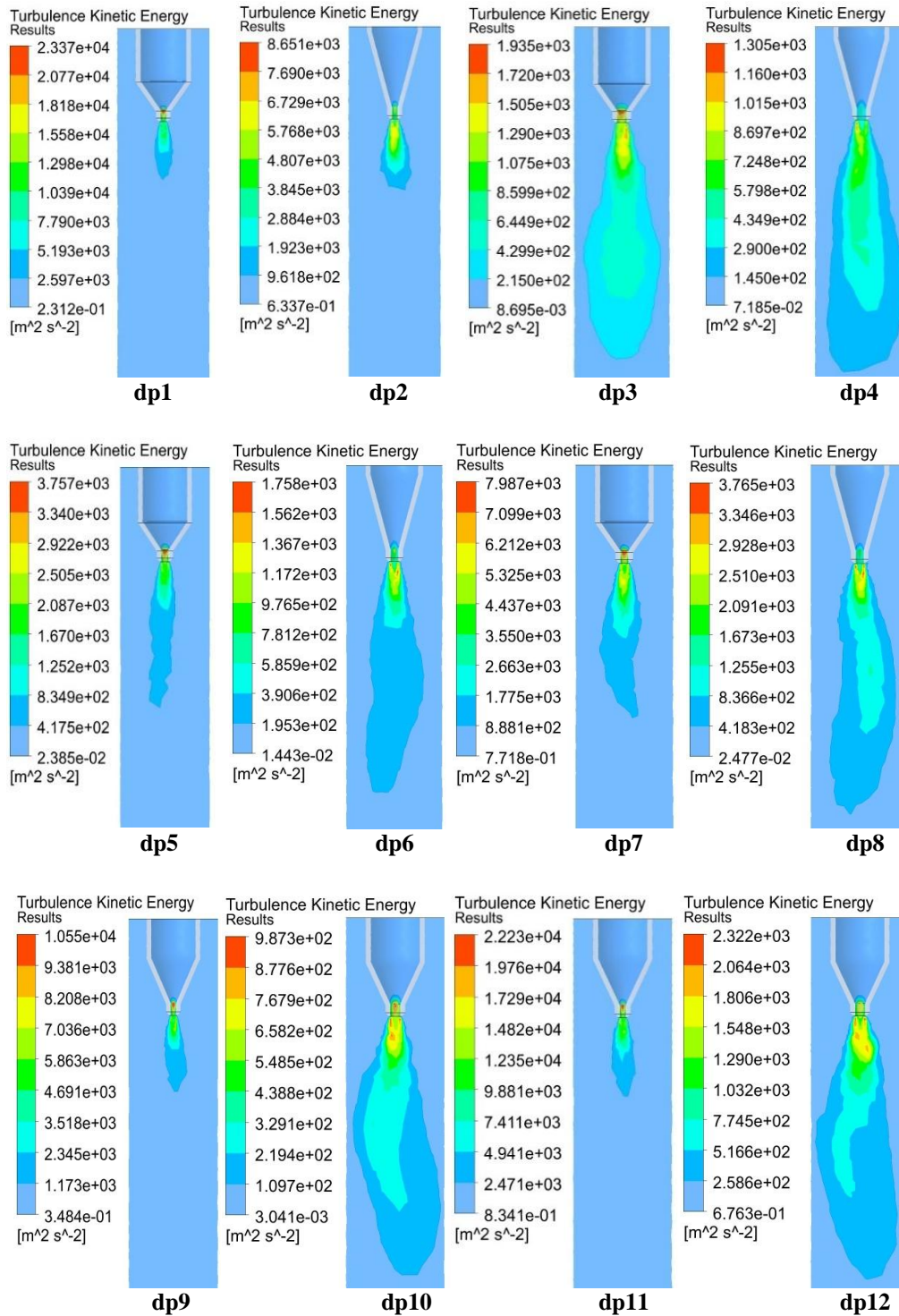


Figure 5. Turbulence kinetic energy at plane 1

3.4. Eddy viscosity

Eddy viscosity reflects the viscous friction effects of turbulent flow. The highest eddy viscosity, measured at 0.02339 Pa·s, was observed at a 155° angle. 0.6 mm exit width and 16 m/s inlet velocity as shown in Figure 6. This finding indicates that turbulent flow intensifies at high nozzle exit velocities and narrower nozzle angles.

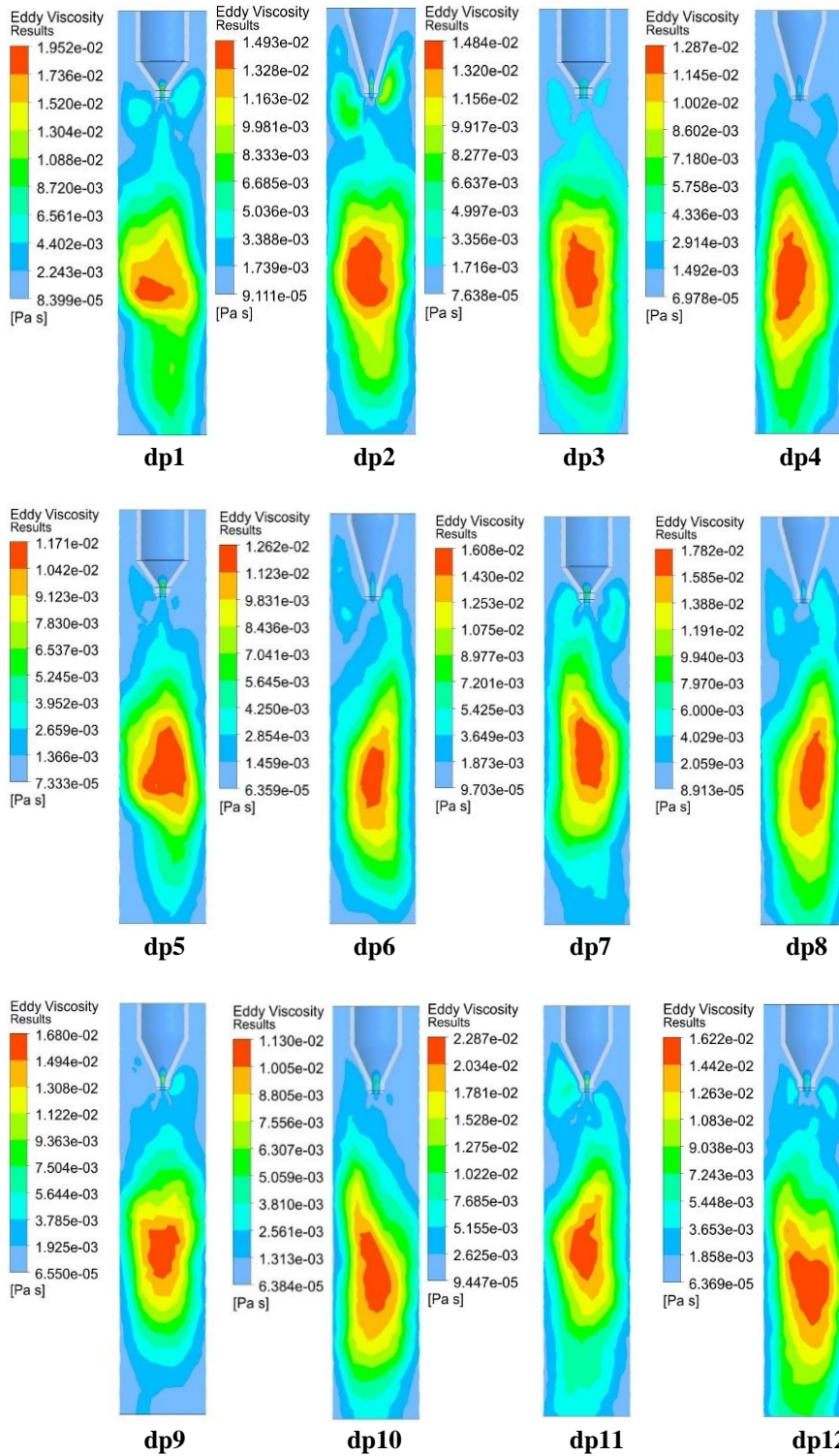


Figure 6. Eddy viscosity at plane 1

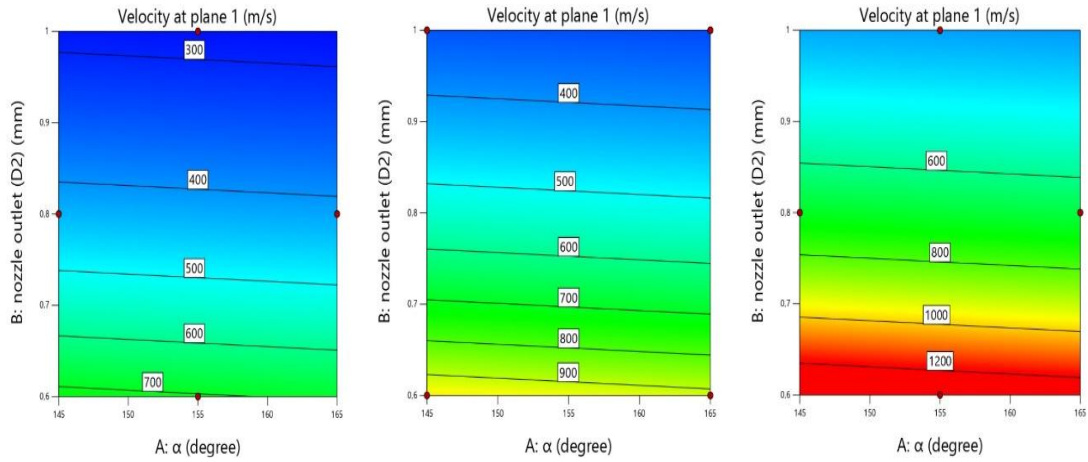


Figure 7: Velocity at plane 1 (velocity inlet; a:10 m/s. b:13 m/s. c:16 m/s)

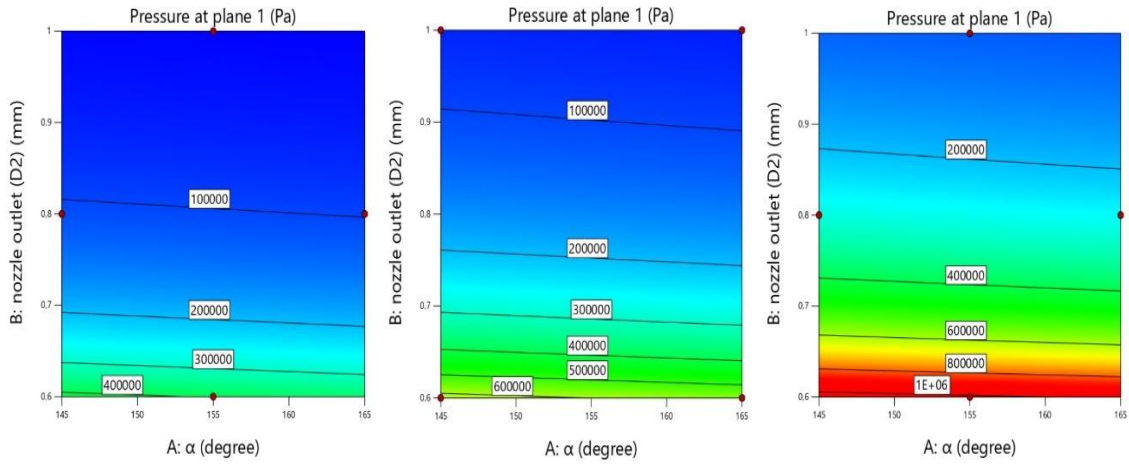


Figure 8. Pressure at plane 1 (velocity inlet; a:10 m/s. b:13 m/s. c:16 m/s)

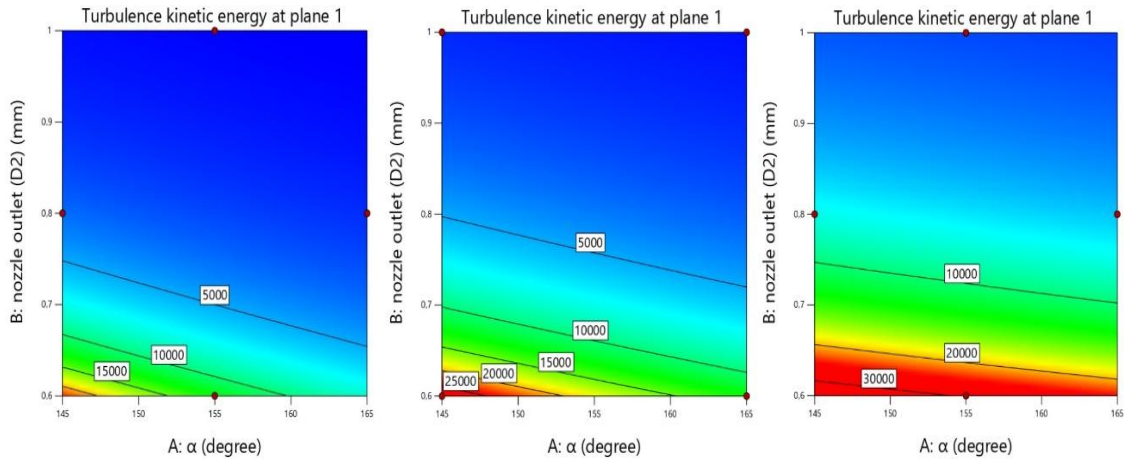


Figure 9. Turbulence kinetic energy at plane 1 (velocity inlet; a:10 m/s. b:13 m/s. c:16 m/s)

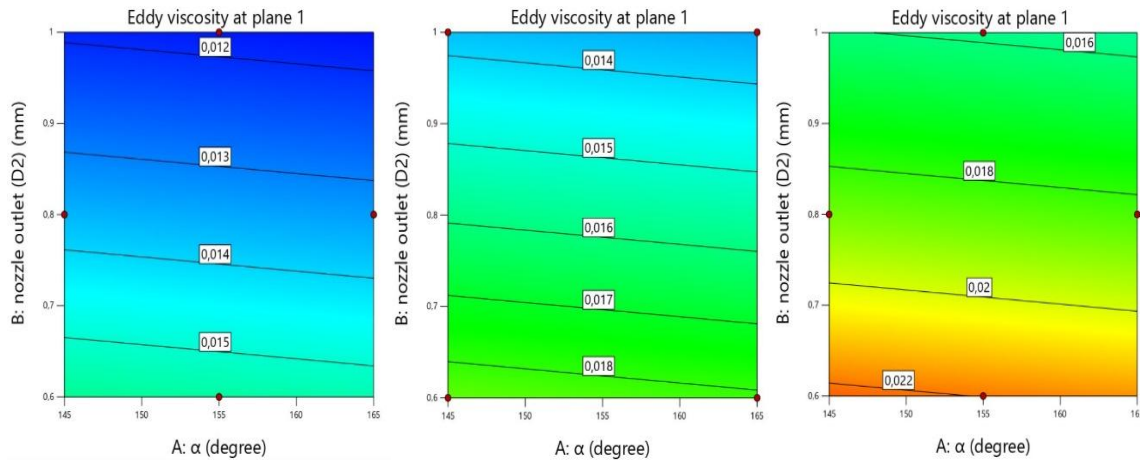


Figure 10. Eddy viscosity at plane 1 (velocity inlet; a:10 m/s. b:13 m/s. c: 16 m/s)

4. Conclusions

This essay covers the effects of the nozzle geometry and outlet velocity on the spraying performance. The analysis indicated that, both a smaller diameter nozzle and higher output velocities will augment flow velocity, pressure, the turbulent kinetic energy and Eddy viscosity. The high turbulence value on the other hand, can deteriorate the flow regimes in some applications. Interaction effects between nozzle angle, outlet diameter and inlet velocity were analyzed using the RSM model. The results showed that the combined influence of small outlet diameters and high inlet velocities significantly increased both turbulence kinetic energy and Eddy viscosity. Additionally, the interaction between nozzle angle and velocity had a notable impact on pressure distribution and flow uniformity. These interactions emphasize the non linear behavior of spray dynamics and validate the usefulness of the RSM approach for capturing complex parameter dependencies. The limitations related to computational modeling are the key limitations of this study. The fluid was modeled as being incompressible in its property and the simulations were done in a steady state which can be a great source of missing pick up data for the transient spray dynamics. Also, temperature effects, phase change and droplet break up mechanisms were not explicitly modeled. The material properties were treated as constant and the turbulence parameters also were treated as constant, which could lead to unrealistic situations while being compared to the real world. In spite of these limitations, the form of the trends and ranges of the values were consistent with theoretical behavior and published data as well, which proved the general validity of the analysis. The results obtained are consistent with the previous studies in the literature. The main findings of the study are as follows:

- A smaller nozzle diameter (0.6 mm) and higher inlet velocities (16 m/s) have been reached the maximum flow velocity, pressure, the highest turbulent kinetic energy and Eddy viscosity values, respectively.
- Using a larger nozzle (1 mm) and low exit velocities (10 m/s), the flow velocity and pressure values of the flow regime decreased and thus offered a more balanced flow.
- The nozzle angle seems to exert a significant impact on the turbulence by altering the kinetic energy and the viscosity. Wider angles (165°) were linked to an increase in the turbulence levels.

The results showed that turbulence, eddy viscosity, and nozzle diameter are important factors in controlling atomization at high inlet velocities, but they may also contribute to spray instability. Also, the increase in turbulence because of the smaller nozzle angles, results in the break up and distribution of the droplets being affected. The implications of these findings in terms of the optimal quality of the spray process that should be ensured by a proper and careful selection of geometric and operational parameters were a focus of the study.

The study was able to make a good comparison between the CFD results and the physical behavior that has been previously established through methods such as the use of the trends reported in previous studies. The interrelationships between the nozzle geometry, the flow velocity, the turbulence characteristics and the spray quality were found in the previous studies Zhang et al. [19], Du et al. [21], and Wang et al. [23]. Model outputs, such as the increase in turbulence kinetic energy when the inlet velocity is increased and the outlet diameter is decreased, agree well with theory. These findings reinforce the confidence in the numerical results obtained from the simulations conducted in this study.

The study outcomes demonstrate that in the applications requiring high needle velocities and pressure, the smaller diameter nozzles and higher exit velocities would be more efficient. To improve spray performance, turbulence kinetic energy and Eddy viscosity should be kept as low as possible. Small diameter nozzles may provide an advantage in high-speed applications, but the increased turbulence energy losses and the more challenging spray control may result. On the other hand, larger nozzle diameters can perhaps produce a more stable flow regime at lower velocities. Through this, therefore, a careful optimization in the nozzle design is required to be able to achieve an ideal spray performance. Future research should address transient effects and phase changes for more realistic predictions, and the implications of these findings should be leveraged to achieve improved spray performance.

5. Author Contribution Statement

F.T. contributed to the design, analysis of the data, interpretation of the results, spelling check and content and A.E.D. contributed to the creation of the idea, literature review, analysis of the data, interpretation of the results, spelling check and checking the paper in terms of content.

6. Ethics Committee Approval and Conflict of Interest

There is no need for an ethics committee approval in the prepared paper. There is no conflict of interest with any person/institution in the prepared paper.

7. Ethical Statement Regarding the Use of Artificial Intelligence

No artificial intelligence-based tools or applications were used in the preparation of this study. The entire content of the study was produced by the author in accordance with scientific research methods and academic ethical principles.

8. References

- [1] M. Gorokhovski, and M. Herrmann, "Modeling primary atomization." *Annu. Rev. Fluid Mech.* vol. 40. no. 1. pp. 343–366. Jan. 2008.
- [2] D. P. Schmidt, and M. L. Corradini, "The internal flow of diesel fuel injector nozzles: A review." *Int. J. Engine Res.* vol. 2. no. 1. pp. 1–22. Feb. 2001.
- [3] J. Dukowicz, "A particle fluid numerical model for liquid sprays." *J. Comput. Phys.* vol. 35. no. 2. pp. 111–566. Apr. 1980.
- [4] L. Pickett, J. Manin, C. Genzale, D. Siebers, M. Musculus, and C. Idicheria, "Relationship between diesel fuel spray vapor penetration/dispersion and local fuel mixture fraction." *SAE Int. J. Engines.* vol. 4. no. 1. pp. 764–799. Jun. 2011.
- [5] D. Siebers, "Liquid-phase fuel penetration in diesel sprays." *SAE Tech. Paper.* vol. 107. no. 3. pp. 1205–1227. 1998.
- [6] D. Siebers, "Liquid-phase fuel penetration in diesel sprays based on mixing-limited vaporization." *SAE Trans.* vol. 108. pp. 703–728. 1999.
- [7] J. Abraham, "What is adequate resolution in the numerical computations of transient jets?." *SAE Trans. J. Engines.* vol. 106. pp. 141–151. 1997.
- [8] V. Iyer, and J. Abraham, "Penetration and dispersion of transient gas jets and sprays." *Combust. Sci. Technol.* vol. 130. no. 1. pp. 315–334. Sep. 1997.
- [9] W. Ning, R. Reitz, R. Diwakar, and A. Lippert, "An Eulerian-Lagrangian spray and atomization model with improved turbulence modeling." *Atomization Sprays.* vol. 19. no. 8. pp. 727–739. Jan. 2009.
- [10] X. Zhou, "Characterization of interactions between hot air plumes and water sprays for sprinkler protection." *Proc. Combust. Inst.* vol. 35. no. 3. pp. 2723–2729. Jan. 2014.
- [11] K. McGrattan, A. Hammins, and D. Stroup, "Sprinkler. smoke and heat vent. draft curtain interaction-large scale experiments and model development." *Nat. Inst. Standards Technol.* p. 148. Sep. 1998.
- [12] A. W. Marshall, and M. di Marzo, "Modelling aspects of sprinkler spray dynamics in fires." *Process Saf. Environ. Prot.* vol. 82. no. 2. pp. 97–104. Mar. 2004.
- [13] C. T. Crowe, "Modeling spray–air contact in spray-drying systems." in *Adv. Drying.* vol. 1. A. S. Mujumdar. Ed. Washington. DC: Hemisphere. 1980. pp. 63–99.
- [14] D. E. Oakley, R. E. Bahu, and D. Reay, "The aerodynamics of cocurrent spray dryers." in *Proc. 6th Int. Drying Symp. IDS. Versailles. France.* 1988. pp. 373–378.
- [15] K. Inthavong, Z. F. Tian, H. F. Li, J. Y. Tu, W. Yang, and C. L. Xue, "A numerical study of spray particle deposition in a human nasal cavity." *Aerosol Sci. Technol.* vol. 40. no. 11. pp. 1034–1045. Feb. 2006.
- [16] J. S. Kimbell, R. A. Segal, B. Asgharian, B. A. Wong, J. D. Schroeter, and J. P. Southall, "Characterization of deposition from nasal spray devices using a computational fluid dynamics model of the human nasal passages." *J. Aerosol Med.* vol. 20. no. 1. pp. 59–74. Mar. 2007.
- [17] P. W. Longest, and M. Hindle, "Evaluation of the Respimat Soft Mist Inhaler using a concurrent CFD and in vitro approach." *J. Aerosol Med.* vol. 22. no. 2. pp. 99–112. Nov. 2008.
- [18] P. W. Longest, G. Tian, R. L. Walenga, and M. Hindle, "Comparing MDI and DPI aerosol deposition using in vitro experiments and a new stochastic individual path (SIP) model of the conducting airways." *Pharm. Res.* vol. 29. no. 6. pp. 1670–1688. Jan. 2012.
- [19] T. Zhang, B. Dong, X. Chen, Z. Qiu, R. Jiang, and W. Li, "Spray characteristics of pressure-swirl nozzles at different nozzle diameters." *Appl. Therm. Eng.* vol. 121. pp. 984–991. 2017.
- [20] B. Önen, E. Altuncu, and A. Çınar, "Erozif aşınma testlerinde farklı nozul tasarımlarının PMMA üzerinde aşınma izi alanı ve yüzey pürüzlülüğüne etkilerinin incelenmesi." *AKU J. Sci. Eng.* vol. 21. pp. 755–763. 2021.
- [21] C. Du, M. Andersson, and S. Andersson, "Effects of nozzle geometry on the characteristics of an evaporating diesel spray." *SAE Int. J. Fuels Lubr.* vol. 9. no. 3. pp. 493–513. 2016.
- [22] M. H. H. Ishak, F. Ismail, S. C. Mat, M. Z. Abdullah, M. S. Abdul Aziz, and M. Y. Idroas, "Numerical analysis of nozzle flow and spray characteristics from different nozzles using diesel and biofuel blends." *Energies.* vol. 12. no. 2. p. 281. 2019.

- [23] F. Wang, Z. He, J. Liu, and Q. Wang, "Diesel nozzle geometries on spray characteristics with a spray model coupled with nozzle cavitating flow." *Int. J. Automot. Technol.*. vol. 16. pp. 539–549. 2015.
- [24] X. Hu, H. Wang, Y. Liu, and J. Zhu, "Comparative analysis of spray performance between blade-structured and conventional twin-fluid nozzles using CFD." *Heliyon*. vol. 11. no. 4. p. 107108. 2025.
- [25] T. Yilmaz, and M. Kaya, "Numerical investigation of atomization performance in pneumatic nozzles with aerofoil modifications." *Comput. Fluids*. vol. 275. p. 107540. 2025.
- [26] L. Chen, Y. Zhang, and Q. Xu, "Coupled analysis of internal nozzle flow and external spray characteristics in diesel injectors." *Fuel Commun.*. vol. 18. p. 100560. 2024.
- [27] J. Collazo, J. Porteiro, D. Patino, J. L. Miguez, E. Granada, and J. Moran, "Simulation and experimental validation of a methanol burner." *Fuel*. vol. 88. no. 2. pp. 326–334. Feb. 2009.
- [28] M. Fogliati, D. Fontana, M. Garbero, M. Vanni, G. Baldi, and R. Donde, "CFD simulation of paint deposition in an air spray process." *J. Coat. Technol. Res.*. vol. 3. pp. 117–125. Apr. 2006.
- [29] T. H. Shih, W. W. Liou, A. Shabbir. Z. Yang, and J. Zhu, "A new $k-\epsilon$ eddy viscosity model for high Reynolds number turbulent flows." *Comput. Fluids*. vol. 24. no. 3. pp. 227–238. Mar. 1995.
- [30] P. K. Senecal, D. P. Schmidt, I. Nouar, C. J. Rutland, R. D. Reitz, and M. L. Corradini, "Modeling high-speed viscous liquid sheet atomization." *Int. J. Multiph. Flow*. vol. 25. no. 6–7. pp. 1073–1097. Sep.–Nov. 1999.