6TH International Conference and Workshop on Engineering and Technology Research (ICWETR) LAUFET 2025: 448-455

Computational Fluid Dynamics Simulation of the Influence of Swirl Chamber Diameter Ratios on Spray Characteristics

[1]Akande F. B., [2]Taiwo A. and [3]Ayobamidele S. J.

[1]Department of Agricultural Engineering. Ladoke Akintola University of Technology, Ogbomoso, Nigeria.
[2]Department of Environmental Engineering, Igbinedon University, Okada, Nigeria.
[3] Department of Vocational and Technical Education, University of Ilesa, Ilesa, Nigeria.

https://www.laujet.com/

Keywords:

Swirl Chamber length, Spray Characteristics, Nozzle and Simulation

Corresponding Author:
sinatu_ayobamidele@unilesa.edu.ng
+2348135565615

ABSTRACT

Swirl nozzles are widely used in various agricultural and industrial applications, including fuel injection and fire suppression, due to their ability to generate fine droplets and uniform spray patterns. The geometry of the swirl chamber plays a crucial role in determining the spray characteristics, such as droplet size distribution, spray angle, atomization efficiency, and breakup length. This study investigates the influence of key geometric parameters such as chamber diameter, swirl length, and exit orifice size on the performance of swirl nozzles. Simulation analyses were conducted on three nozzles of different swirl chamber length diameter ratio (0.2, 0.5and1.0) to examine the influence of swirl chamber geometry on spray characteristics of the three swirl nozzles at 200kPa, showing the flow rate and pressure drops of the nozzles. Results indicate that in all cases, velocity starts low, increases gradually, and then experiences a sharp spike at approximately 0.13–0.15 meters along the flow path. The peak velocity varies slightly across different cases, ranging between 15.4 m/s and 15.7 m/s. The upstream pressure remains stable around 200 KPa in all cases. A sudden pressure drop occurs at 0.13–0.15 meters, coinciding with the velocity peak. the magnitude of pressure drop is consistent across cases, confirming that swirl intensity affects velocity more significantly than pressure loss.

INTRODUCTION

Swirl nozzles operate by imparting a rotational motion to the liquid before it exits the nozzle, generating a hollow cone or full cone spray pattern depending on the design. The efficiency of atomization and spray distribution is significantly influenced by the geometry of the swirl chamber, which governs the internal flow dynamics and droplet formation process. The swirl chamber is a critical component of the nozzle, where the interaction between centrifugal forces and fluid inertia determines the intensity of the swirl. Key design parameters, such as chamber diameter, inlet angle, swirl vane configuration, and exit orifice size, directly affect the resulting spray characteristics, including droplet size distribution, spray angle, breakup length, and uniformity. A well-optimized swirl chamber enhances atomization by increasing swirl intensity, leading to finer droplets and improved dispersion. However, excessive swirl strength or improper chamber design can lead to undesirable effects such as flow instabilities, droplet coalescence, and irregular spray patterns. Understanding the relationship between swirl chamber geometry and spray characteristics is essential for optimizing nozzle performance in applications such as agricultural spraying, fuel injection, fire suppression, and cooling systems. This study aims to investigate how variations in swirl chamber design influence spray performance using a simulation approach.

Zhang *et al.,* (2022) on their study on experimental and simulation analysis of spray and combustion characteristics in a swirl-chamber diesel engine combined experiments using a rapid compression and expansion machine with 3D-CFD simulations to visualize and analyze spray behavior and combustion processes within a

swirl chamber, the research provided insights into flow dynamics, spray formation, and ignition behavior, contributing to the validation of simulation models for swirl-chamber diesel engines. Lu *et al.* (2023) investigate how varying combustion chamber diameters affect combustion and emission performance in a lateral swirl combustion system through experiments and simulations. they established a linear relationship between optimal chamber diameter and fuel spray penetration distance, proposing the spray excess penetration coefficient (τ) as a matching parameter.

Fahim and Müller (2023) assessed how different geometrical parameters of a spray drying chamber influence internal airflow dynamics by quantifying the effects of design parameters. The research aimed to support the design of co-current spray chambers and provide a methodology for future parametric CFD studies. (Bashir and Ghadiri, 2024) works on utilizing computational fluid dynamics (CFD). This study explored how varying the geometrical configurations of swirls impacts flow dynamics in an industrial-scale annular lance. The research identified optimal designs that balance enhanced mixing with acceptable pressure drops. (Bhandari and Sharma, 2024) examined how fuel temperature variations influence spray characteristics in a pressure-swirl atomizer featuring a spiral path; their findings indicated that higher fuel temperatures lead to increased spray angles and variations in droplet size distribution. (Rahim and Zulkifli, 2023), experimental research focused on the spray angles produced by tangential-vane swirl atomizers. Results demonstrated that combining tangential inlets with swirl-generating vanes produces wider spray angles compared to using tangential inlets alone, emphasizing the role of swirling intensity over Reynolds number in determining spray angle.

Nozzle selection has a significant influence on whether or not the droplets reach the specific target location (Oskan, 2020). Chuech *et al.* (2018) indicated that conical nozzles with high swirl angles produce wider spray angles and improved fuel dispersion. Conversely, cylindrical nozzles tend to generate more coherent, longer penetration sprays. Swirl chambers generate recirculation zones that influence droplet size and distribution. Recent studies consistently highlight that swirl chamber geometry, particularly variations in diameter ratio, vane angle, and swirl intensity, plays a crucial role in determining spray characteristics such as spray angle, droplet size distribution, atomization efficiency, and evaporation rate. CFD simulations, often validated by experimental data, have proven to be effective tools in capturing the complex fluid dynamics within swirl chambers. Despite these advancements, limited research has specifically focused on varying swirl chamber diameter ratios under fixed operating pressures. Therefore, this study aims to fill this gap by conducting CFD simulations to investigate the effect of three distinct swirl chamber diameter ratios (0.2, 0.5, and 1.0) on spray characteristics at a constant pressure of 200 kPa.

METHODOLOGY

The geometry of the swirl chamber was designed with an inlet nozzle, swirl chamber, and exit orifice. The nozzle was designed to inject fluid tangentially into the chamber to create a swirling flow. The chamber's diameter was varied with the diameter ratios of 0.2, 0.5, and 1.0 corresponding to the ratio of the swirl chamber diameter to the nozzle exit diameter. The study utilizes Computational Fluid Dynamics (CFD) to simulate fluid flow and pressure drops under different swirl chamber geometries. The geometric were based on a swirl chamber with three different diameter ratios: 0.2, 0.5, and 1.0. These values were chosen to represent a range of common swirl chamber configurations in atomization systems. The simulations were conducted at a constant inlet pressure of 200 kPa to represent typical operating conditions in spray atomization systems. The fluid used in the simulations was

assumed to be a single-phase fluid, with properties representative of water. The chamber walls were modeled as non-slip walls, and the orifice was assumed to be smooth for this simulation. The inlet velocity profile was specified to mimic the tangential injection typical of swirl chambers, ensuring that the swirl was sufficiently generated within the chamber.

Design analysis of the swirl nozzle

According to Taiwo (2012), the radial and axial components of swirl nozzles are in the form of the Francis hydraulic turbine, which forms the basis of its design analysis. The Francis turbine is a form of reaction turbine that requires high pressure head and low flow rate. The Bernoulli Equation, satisfying the head between water surfaces in the reservoir to the centre of the outlet, is represented in Equation 1 (Taiwo and Oje, 2012)

$$\frac{V_i^2}{2g} + \frac{P_i}{\gamma} + Z_i + h_p = \frac{V_6^2}{2g} + \frac{P_o}{\gamma} + Z_o + \sum h \quad (1)$$

Where V_i, V_o are the average velocities at the water surface and outlet, respectively. P_1, P_o is the pressure intensity at the water surface and outlet, N/m^2. Z_i, Z_o is elevation at the water surface, outlet, m. h_p is energy supplied by the pump, Nm. g is acceleration due to gravity, m/s^2. γ is the specific weight of a flowing fluid, N/m^3. $\sum h$ is the summation of all head forces, Nm.

Neglecting all head losses and infinite kinetic energy in the tank and pressure gradients, the rate at which energy, E must be supplied by pump in terms of power required is given in Equation 2 (Carlson, 2000) given the recommended flow rate, Q through a long-jet orchard sprayer nozzle of 4.8 x 10^{-4} m^3/s and working pressure of 10 x 10^5 N/m^2 (Taiwo, 2012).

$$power\ (kW) = \frac{Q\gamma h_P}{1000} \quad (2)$$

Where Q is flow discharge = 4.8 x 10^{-4} m^3/s, γ is the specific weight of a flowing fluid (ρg), N/m^3, h_p is energy supplied by the pump, Nm. Energy supplied by the pump and the specific weight of fluid, γ can be derived from Equations 3 and 4 (Spinelli *et al.,* 2010)

$$h_p = \frac{P}{\gamma} \quad (3)$$

$$\gamma = \rho g \quad (4)$$

Where P is working pressure, ρ is the density of fluid (water), kg/m^3

The $h_p = 101.94\ m.$

Therefore, the required pump power is calculated using Equation 2 to be 0.48 kW, which is equivalent to 0.64 Hp Furthermore, simplifying the Bernoulli Equation, the velocity of flow, V_f can be calculated from Equation 5 (Taiwo, 2012), thus

$$V_f = \sqrt{2gh_p} \quad (5)$$

Where V_f is the velocity of flow, m/s. Substituting the already calculated pump head, h_p, V_f equals 44.72 m/s. From the continuity Equation, the flow rate at the nozzle exit orifice is a function of the area of the orifice and the velocity of flow at the orifice. Then, the diameter of the nozzle exit orifice, D_T can be calculated using the formula in Equation 6 (Taiwo and Oje, 2008).

$$A_o = \frac{\pi D_T^2}{4} \quad (6)$$

Where A_o is the area of the nozzle exit orifice, m^2. D_T is the diameter of the nozzle exit orifice, m.

Therefore,

$$D_T = \sqrt{\frac{4Q}{\pi V_f}} = \sqrt{\frac{4 \times 4.8 \times 10^{-4}}{\pi \times 44.72}} = 3.7\ mm$$

The width of the square-sectioned helical groove of the swirl element is calculated from Equation 7 (Taiwo and Oje, 2008), thus, From the continuity equation,$Q = A_o V_f = W_s^2 V_f$,

$$W_s = \sqrt{\frac{Q}{V_f}} \qquad 7$$

Where W_s is the width of square square-sectioned helical groove, m.

Hence, Ws = 3.27 mm, say 4mm.

The ratio of swirl element width to diameter is given in Equation 3 (Kim *et al.,* 2008)

$$n = \frac{B}{D} \qquad 8$$

Where n is the ratio of swirl element width to diameter. Ranges from 1.00 to 2.22 for swirl elements (Rajput, 1998). B is the width of the swirl element, m. D is the diameter of the swirl element, m. Figure 1 shows the diagram of the three swirl nozzles of different swirl chamber diameter ratios.

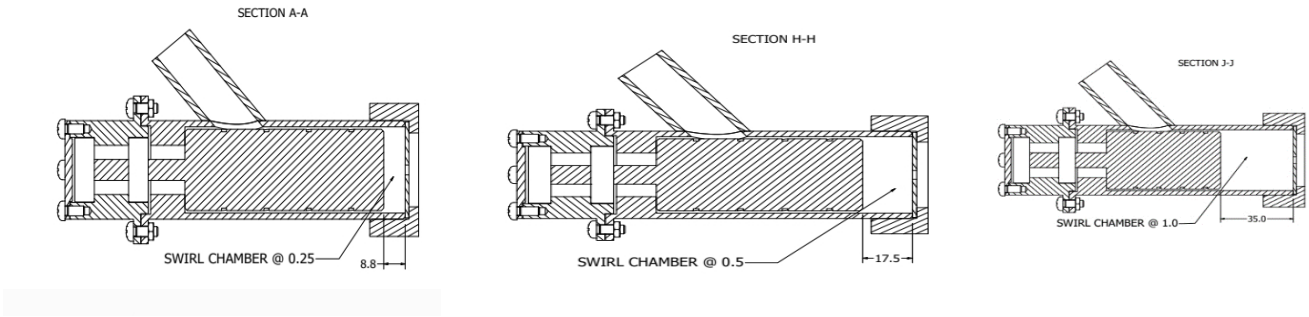

Figure 1: Swirl Nozzles of Different Swirl Chamber Diameters

RESULTS AND DISCUSSION

SWRIL CHAMBER 0.25 (200 Kpa)

SCL: D (0.2 Short Swirl Chamber)

The graph in Figure 2 shows the velocity magnitude along a parametric distance in a CFD simulation, steady region is from 0–0.1 m and the velocity remains relatively low with minor fluctuations, at 0.14–0.15 m there is a sharp velocity increase which suggests a critical flow transition, likely due to flow acceleration or a sudden contraction. After the Peak at 0.15–0.22 m, the velocity drops after the peak, indicating potential dissipation, turbulence, or flow separation, vortex breakdown downstream. Figure 3 represents the static pressure along the parametric distance; the pressure remains nearly constant at around 200 KPa, indicating a relatively stable inlet condition. There is a slight dip before the major drop due to swirl chamber effects or minor flow adjustments. There is a sudden pressure drop between 0.13 to 0.15 m pressure drop could also indicate the formation of a low-pressure core due to centrifugal forces from swirling motion. From 0.15 m onward, indicate the transition to a

high-speed jet or vortex breakdown region where energy is dissipated. This is in line with Bernoulli's principle if flow contracts, velocity must increase, causing pressure to drop.

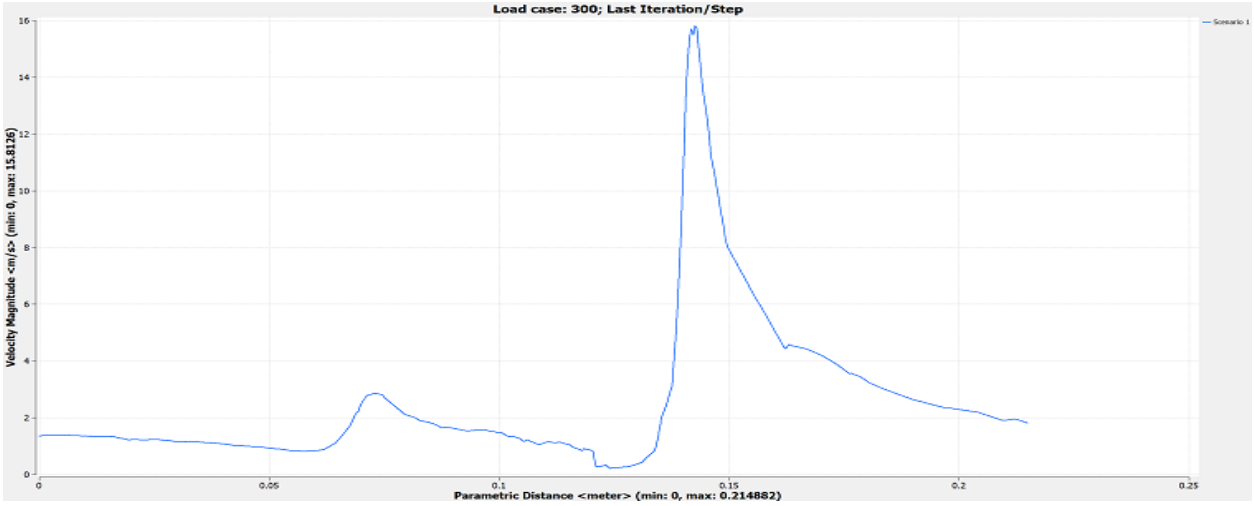

Figure 2: Velocity Magnitude along the Swirl Chamber of 0.25

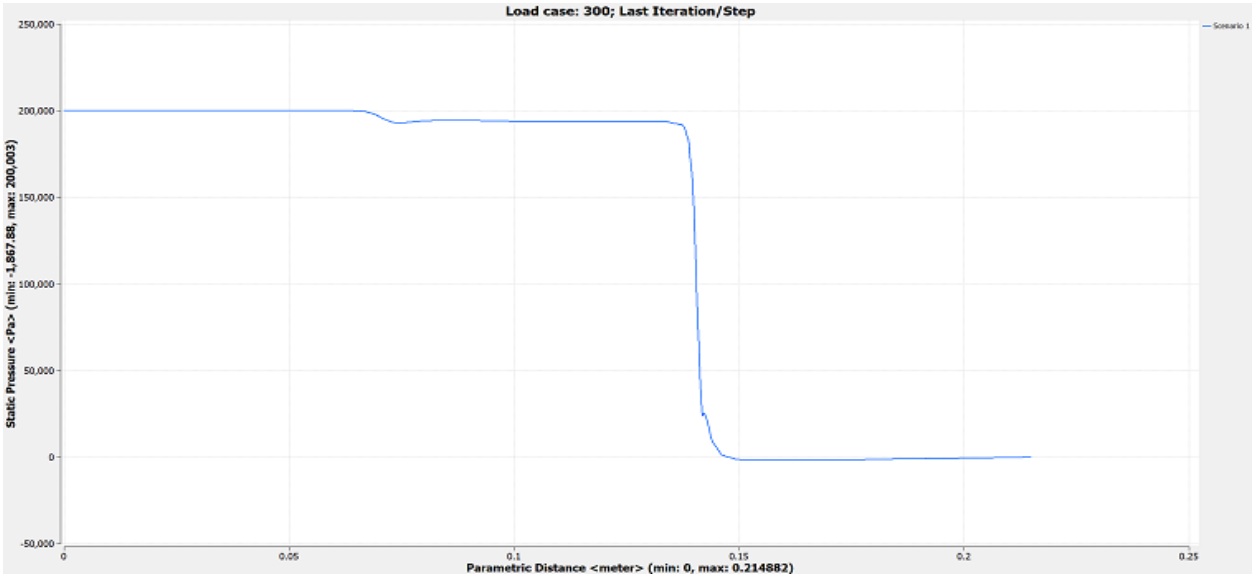

Figure 3: Static Pressure Distribution for SCLD: R 0.25

SWRIL CHAMBER 0.5 (200 Kpa)

SCL: D (0.5 Moderate Swirl Chamber)

Figure 4 shows a slightly higher peak velocity around 15.67 m/s and minor variations in flow behavior, at 0–0.1 m. The flow remains relatively slow, with minor fluctuations. There is a small bump around 0.06 m, possibly due to swirl-induced secondary flows. between 0.13–0.15 m, there is a flow acceleration due to swirl effects. Peak velocity is at 15.67 m/s, indicating a strong swirl-induced pressure drop, leading to high-speed flow, from 0.15 m onward, velocity drops after the peak, which might be due to vortex breakdown, energy dissipation, or flow expansion. This appears slightly different from the SCLD ratio of 0.25, which could indicate changes in swirl intensity or nozzle geometry**.** These findings align with those of Zhang *et al*. (2022), where the increased chamber volume was shown to produce a broader spray pattern due to the enhanced swirling motion and reduced axial velocity.

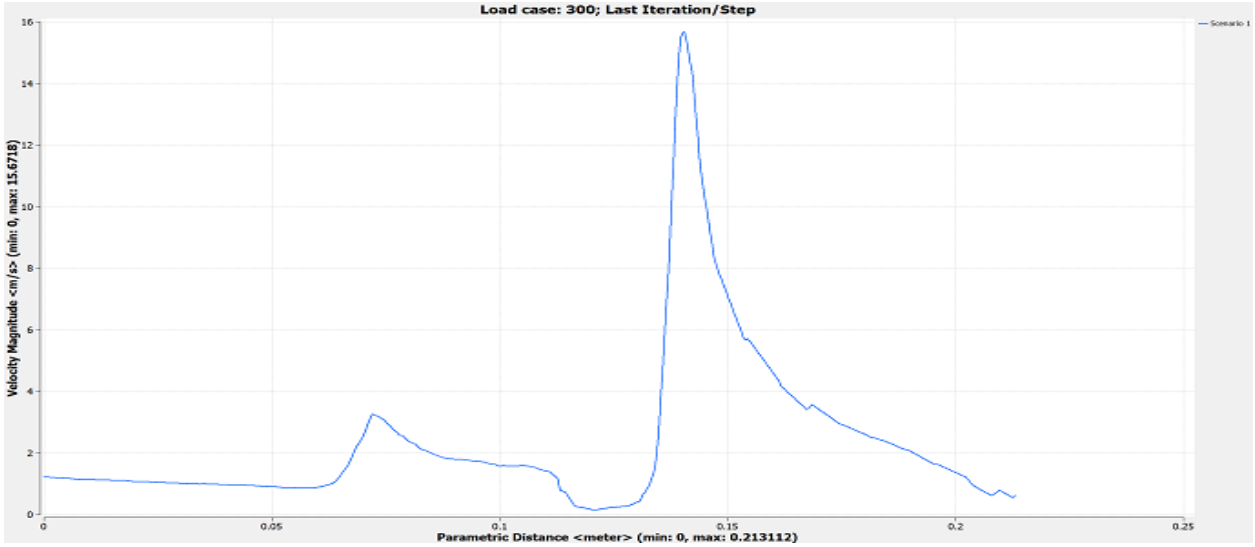

Figure 4: Velocity magnitude along the swirl chamber of 0.5

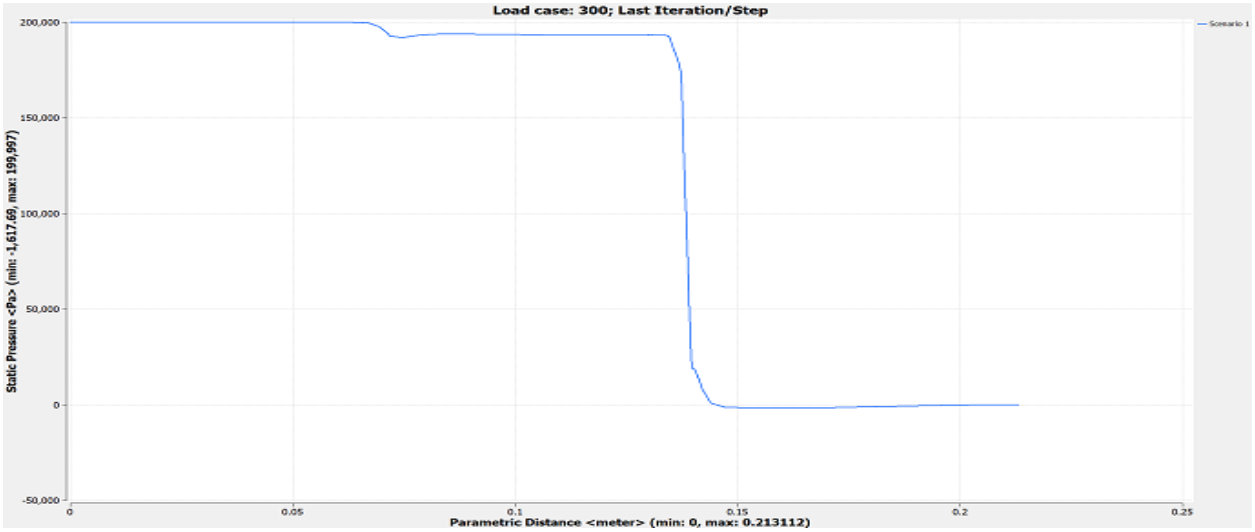

Figure 5: Static Pressure distribution for SCLD: R 0.5

SWRIL CHAMBER 1.0 (200 Kpa)

SCL: R (1.0 Long Swirl Chamber)

This velocity profile corresponds to Load Case 198 as shown in Figure 6 and it follows a similar trend to the previous velocity plots but with some noticeable differences. At 0–0.1 m, the velocity remains relatively low and stable. A small peak around 0.06 m suggests localized flow acceleration, likely due to swirl-induced recirculation or vortex formation. The velocity reaches a maximum of 15.42 m/s, which is slightly lower than the previous case (15.67 m/s). The peak again occurs at a critical transition point, likely at the nozzle throat, where contraction and swirl effects combine to accelerate the flow. The magnitude of this peak suggests that swirl is moderately strong but possibly lower than in the previous case, at 0–0.1 m the pressure remains relatively constant at 200 KPa in this region, the pressure remains nearly constant close to 0 Pa beyond 0.15 m, this suggests that the fluid has fully expanded and that the swirl jet is stabilizing downstream.

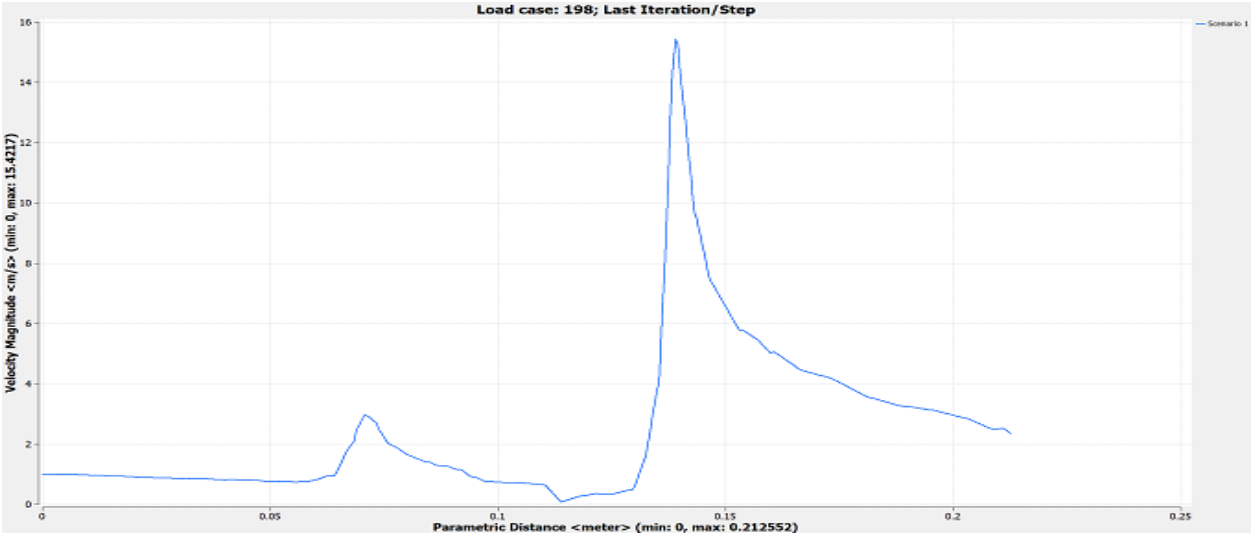

Figure 6: Velocity magnitude along the swirl chamber of 1.0

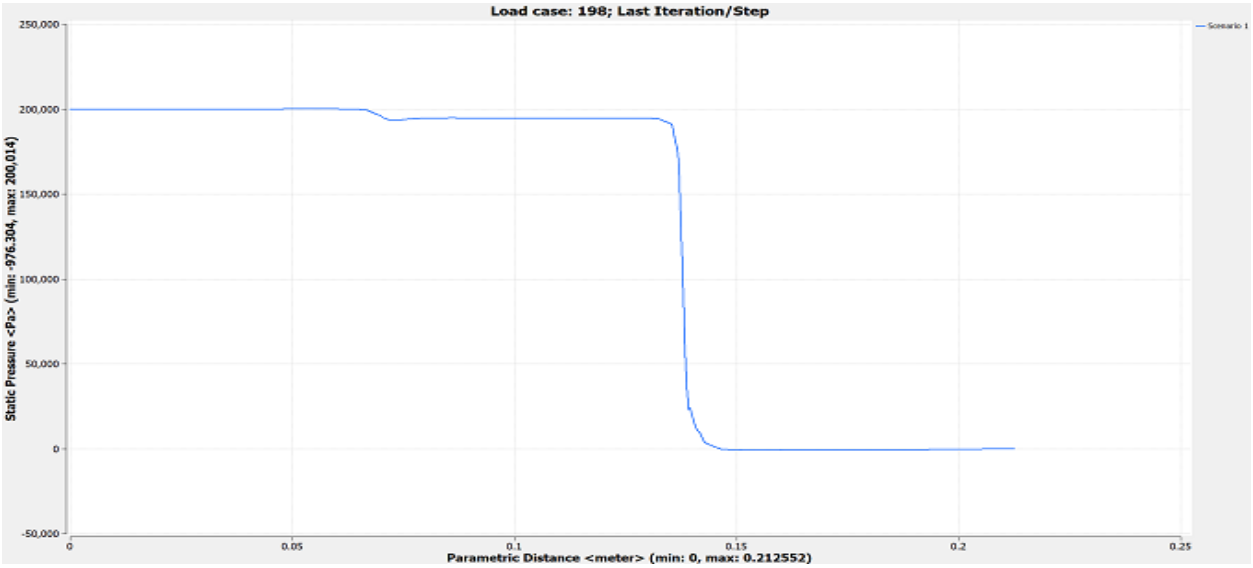

Figure 7: Static Pressure distribution for SCLD: R 1.0

CONCLUSION

This CFD simulation study successfully demonstrates the significant impact of swirl chamber geometry, specifically, diameter ratios, on spray characteristics under a constant pressure of 200 kPa. The velocity graphs suggest that stronger swirl intensity will lead to a higher velocity peak**.** The pre-peak fluctuations in velocity indicate that higher swirl chamber ratios induce greater pre-nozzle turbulence and the Pressure drop trends remain similar, implying that swirl influences flow acceleration. These changes in spray behavior suggest that larger swirl chamber diameters can enhance atomization efficiency, improve fuel-air mixing, and potentially optimize combustion processes. Future studies could expand on these findings by exploring a wider range of operating pressures, fluid properties, and swirl chamber geometries to further optimize spray atomization systems.

REFERENCES

Bashir, M. S., and Ghadiri, M. (2024). *CFD Investigation of Flow Hydrodynamics and Optimization in an Industrial-Scale Annular Lance with Swirl Flow*. **The Canadian Journal of Chemical Engineering**, 102(3), 1227–1242. https://doi.org/10.1002/cjce.25274

Bhandari, A., and Sharma, R. (2024). *Numerical Investigation of the Impact of Fuel Temperature on Spray Characteristics in a Pressure-Swirl Atomizer with Spiral Path*. **Propulsion and Power Research**, https://doi.org/10.1007/s42757-024-0198-x

Chuech, S., Schick, R., Wensing, M., and Leipertz, A. (2018). Impact of nozzle geometry on spray formation. *Fuel, 216*, 63–72. https://doi.org/10.1016/j.fuel.2017.11.121

Fahim, I., and Müller, J. (2023). *Parametric CFD Study of Spray Drying Chamber Geometry: Part I—Effects on Airflow Dynamics*. **Chemical Engineering and Technology**, 46(1), 5–16. https://doi.org/10.3390/chemosensors9010005

Kim, S., Khil, T., Kim, D. and Yoon, Y. (2008). Effect of geometric parameters on the liquid film thickness and air core formation in a swirl injector. Measurement Science and Technology, 20(1):15-43.

Lu, Z., Zhang, H., Ma, Y., and Cheng, Q. (2023). *A Method of Matching the Combustion Chamber Diameter Based on Spray Characteristics for Lateral Swirl Combustion System*. Beijing Institute of Technology. https://pure.bit.edu.cn/en/publications/a-method-of-matching-the- combustion-chamber-diameter-based-on-spraying.

Oskan, E. (2020). Best Practices for Effective and Efficient Pesticide Application. Department of Food, Agricultural and Biological Engineering. The Ohio State University. FABE-532.

Rahim , M. S. A., and Zulkifli, A. H. (2023). *Experimental Analysis of Tangential-Vane Swirl Atomizer Spray Angle*. **Journal of Fluid Mechanics and Thermal Sciences**, 10(1), 88–97. https://semarakilmu.com.my/journals/index.php/fluid_mechanics_thermal_sciences/articl e/view/1229

Rajput, R.K. (1998). Hydraulics for Engineering Students of Degree and Diploma. S. Chand and Company Ltd., Rain Nagar, New Delhi- 11055.

Taiwo, A. (2012). Study of some performance Parameters of a high-pressure Swirl nozzle: An experimental pressure agro-forestry swirl nozzle for use in studying the geometric and performance parameters.

Taiwo, A. and Oje, K. (2008). Development and testing of a swirl Chamber Nozzle. Journal of Agricultural engineering and technology (JAET), 16(1): 11-23

Taiwo, A. and Oje, K. (2012). Theoretical performance simulation of a high pressure agro-forestry swirl nozzle. International Journal of Agricultural and Biological Engineering, 5(4), 31-38.

Zhang, X., Ren, Y., Liu, J., and Liu, Q. (2022). *Experimental and Simulation Analysis of Spray and Combustion Characteristics in a Swirl-Chamber Diesel Engine*. SAE Technical Paper 2022-01-1049. https://doi.org/10.4271/2022-01-1049