Numerical and experimental study on the flame flow field near the fuel nozzle tip

Hoang Long Duong[1], Tien Van Truong[1], Quoc Cuong Dang[1], Quang Dinh Phan[1], Thi Kim Loan Au[2], Duc Thong Hong[3], Hung Truyen Luong[1,*]

[1]Nguyen Tat Thanh University, Viet Nam
[2]Ho Chi Minh University of Technology and Education, Viet Nam
[3]Ho Chi Minh University of Technology, Viet Nam
[*]lhtruyen@ntt.edu.vn

Abstract

The generation of a recirculation zone in jet flow has recently been demonstrated to influence flame stability and soot formation. In this study, we conducted both experiments and numerical simulations to elucidate the mechanisms underlying vortex formation in laminar coflow diffusion flames. Numerical models were developed by incorporating various parameters, including fuel density, jet velocity, fuel temperature, and ambient air temperature. The flow patterns near the nozzle indicated how these parameters affect the development of the recirculation zone. Simulation results were validated against experimental data. Among the studied parameters, fuel density, jet velocity, and nozzle diameter emerged as the primary factors influencing recirculation flow, whereas fuel and ambient air temperatures exerted secondary effects. Detailed analysis of each parameter unveiled critical conditions at which the recirculation zone forms.

Received 12/09/2025
Accepted 19/10/2025
Published 28/11/2025

Keywords

diffusion flame, negative buoyancy, jetflame, recirculation

1 Introduction

The study of laminar diffusion flames has become a common subject of fundamental research, with well-established theoretical frameworks, models, and boundary conditions, particularly for investigations into soot formation and toxic emissions. The scope of applications has expanded to include topics such as jet flame geometry design, the effects of different fuel types, parameters controlling flame size and shape, and factors influencing soot production and counterflows [1]. Additionally, research on coflow diffusion flames has examined the influence of fuel dilution, inlet velocity, gravity, and density on near-nozzle fuel flow and soot formation through both experimental and numerical analyses [2-5].

Early investigations into the dynamic structure of buoyant jet diffusion flames combined numerical and experimental approaches to better understand the unsteady interactions between flames and associated vortex motions [6]. A key factor influencing the distortion of the flow field is the buoyant force exerted on fuel and burnt gases, an effect present in most practical combustors. The buoyant force plays a significant role in flame flickering behavior. Recent studies on the hydrodynamic structure of flames with various hydrocarbon fuels have shown deflected flow patterns near the nozzle that vary depending on the fuel

https://doi.org/10.55401/yb881381

type. Fuels with densities greater than air generate buoyant forces acting on both the fuel and burnt gases, leading to the formation of a recirculation zone [7, 8]. The vortex formation in the previous study [1, 7, 8] affects the stability of the coflow diffusion flame. This could influence the accuracy of soot measurements. Further study is needed to define the critical conditions to suppress the recirculation zone and stabilize the flame. In this work, we investigated the mechanism of vortex formation in the fuel flow field near the nozzle exit, focusing on the effects of parameters such as fuel density, fuel inlet temperature, fuel inlet velocity, and nozzle diameter. Both numerical simulations and experimental studies were conducted to characterize the transition from a non-recirculating to a recirculating fuel stream. The threshold value of each parameter for the vortex formation was found out. These results enrich the dataset on factors influencing vortex flow formation, enabling a clearer definition of vortex flow mechanisms.

2 Experiment and numerical simulation

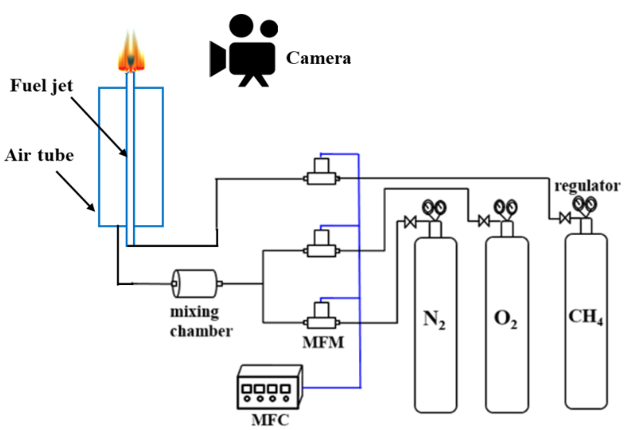

Figure 1 Experimental set up for coflow buner.

The experimental setup for the coflow diffusion flame [8] was adapted from a previous study as shown in figure 1. The inner diameters of the fuel nozzle and coflow section were 10.8 mm and 180 mm, respectively. The nozzle length was 855 mm to ensure fully developed laminar flow. To achieve a uniform velocity profile at the nozzle exit, a layer of glass beads was placed at the bottom of the coflow section, and a ceramic honeycomb was installed near the nozzle exit. We experimentally studied the effect of fuel density on the recirculation zone by using mixtures of methane and n-butane, with molar masses (*Mf*) ranging from 16.0 g/mol (methane) to 58.1 g/mol (n-butane). Air was used as the primary coflow gas. To investigate the influence of mean jet velocity (*uj*) on vortex formation, jet velocities were varied between 4.45 cm/s and 22.5 cm/s, while the coflow velocity (*uco*) was maintained at 6.2 cm/s.

Flow visualization was performed using a continuous-wave diode laser (Coherent Inc., Genesis 532-2000 S OPSL) operating at 532 nm, with titanium dioxide (TiO_2) seeding particles (~ 0.3 μm) to illuminate the flow field. A digital camera (Canon 50D) captured the elastically scattered laser light and flame luminosity. A 105 mm f/2.8 macro lens (Sigma DG MACRO) was used to magnify selected areas of interest, and shutter speeds were set between 1/8 s and 1/30 s to record the path lines of seeding particles effectively.

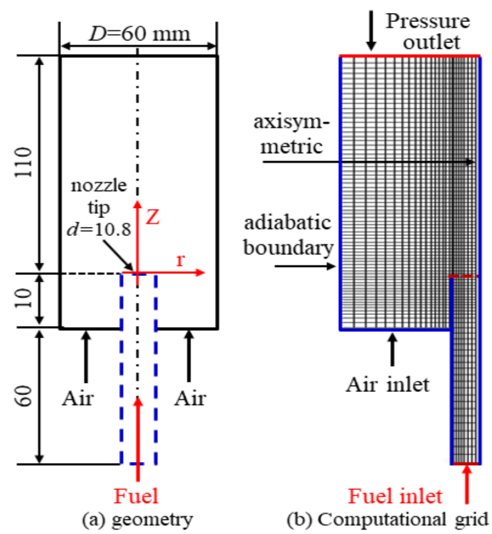

Figure 2 Computational domain and grids of the coflow burner.

To gain a more detailed understanding of the flow characteristics, numerical simulations were performed using ANSYS Fluent [9] to model the transport, diffusion, and chemical reactions of species. Figure 2 illustrates the schematic geometry and computational grid of the coflow burner. The jet diameter was 10.8 mm, while the confined coflow had a diameter of 60 mm and a height of 120 mm. The fuel inlet was positioned 70 mm below the nozzle tip to capture a realistic velocity profile at the nozzle exit, and the coflow inlet was located 10 mm below the fuel nozzle tip.

A grid independence study was performed across four mesh densities ranging from 4,000 to 30,000 elements. The simulation of cold flow, which involves no chemical reaction, was performed for all grid models. The cold flow velocity results show that the results of the 12,200-element grid differed by less than 0.5% in velocity compared to the 30,000-element mesh, confirming mesh adequacy. Therefore, a computational grid consisting of 12,200 quadrilateral elements was adopted for further simulations.

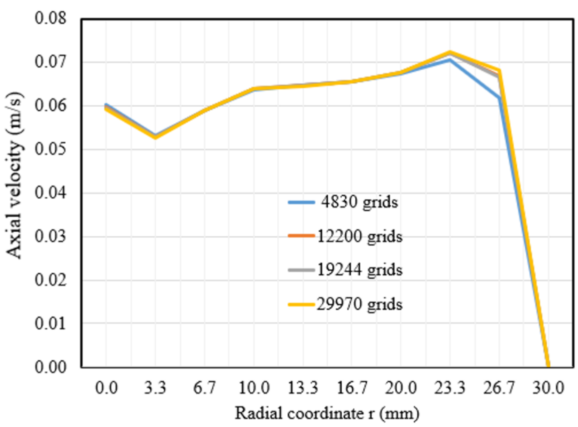

Figure 3 Axial velocity at 10 mm above the nozzle tip

The numerical simulation solved the steady-state, two-dimensional governing equations for continuity, momentum, energy, and species transport using appropriate numerical schemes [10, 11]. A previous study [8] demonstrated that a one-step overall reaction model is sufficient to predict detailed flow characteristics within fuel streams, as numerical results agreed well with experimental data for methane and butane flames. Based on this, the present simulation employed the one-step global reaction mechanism for methane [8] as the reference chemical reaction for fuel mixtures of this study, expressed as follows:

$$CH_4 + 2O_2 \rightarrow CO_2 + 2H_2O. \quad (1)$$

Its reaction rate can be expressed in the Arrhenius form,

$$k = A_k . e^{\frac{E_a}{RT}\left(\frac{T_a}{T_{ao}}\right)} . C_{CH_4} . C_{O_2} , \quad (2)$$

where C_i is the concentration of species i, and a pre-exponential factor (A_k), activation energy (E_a), and the gas constant (R), are 6.9 × 10^{11} kg/m^3·s, 1.32 × 10^{11} J/kmol, and 8,314 J/kmol.ºK, respectively. T_a and T_{ao} denote activation temperature and constant activation temperature, respectively. The temperature ratio, T_a/T_{ao}, was determined by the function of equivalence ratio, Φ, and a mixture fraction, Z, as follows,

$$\phi = 4\frac{Y_{CH4,F}}{Y_{O2,A}}\frac{Z}{(1-Z)}, \quad (3)$$

$$\phi \le 0.64: \quad T_a/T_{ao} = 1 + 8.25(\phi - 0.64)^2,$$
$$0.64 \le \phi \le 1.07: \quad T_a/T_{ao} = 1, \quad (4)$$
$$\phi \ge 1.07: \quad T_a/T_{ao} = 1 + 1.443(\phi - 1.07)^2,$$

Where, Y_{CH4} and Y_{O2} are the mass fraction of methane and oxygen in the fuel and oxidizer stream, respectively.

A parabolic velocity profile (u_j) was imposed as the fuel inlet boundary condition, while the coflow velocity (u_{co}) was set uniformly at 6.2 cm/s at the coflow inlet. The inlet temperatures were maintained at 300 ºK. Adiabatic and impermeable boundary conditions were applied at the peripheral walls, and a constant atmospheric pressure was specified at the axial outlet. To simulate varying fuel densities, the molar mass of methane in the governing equations was adjusted while keeping all other properties and chemical reaction parameters unchanged. This approach enabled an isolated investigation of the effect of fuel density ($\rho_f = M_f P/RT$) on vortex formation, providing a cost-effective analysis without the complexity of modeling multi-species fuel mixtures, where. Where M_f is fuel molar mass, R is the universal gas constant, and P and T are the fuel pressure and temperature, respectively.

3 Results and discussion

In this part, we separately examined and discussed each effect of the fuel density, the jet velocity, the nozzle diameter and the temperature of ambient air and fuel on the formation of vortex formation. Through the investigation, we can withdraw a solution to stabilize the laminar flame by using a coflow burner from the effect of all the parameters on the recirculation zone.

3.1 Effect of fuel density

First, we investigated the influence of the fuel density on the vortex formation. Figure 4 compares experimentally obtained flow path-lines and numerically calculated flow fields for selected fuel molar masses in the range of 16.0 (M_{CH4}) ≤ M_f ≤ 58.1 (M_{C4H10}) g/mol at fixing jet velocity u_j = 4.45 cm/s and the ambient air with coflow velocity u_{co} = 6.2 cm/s.

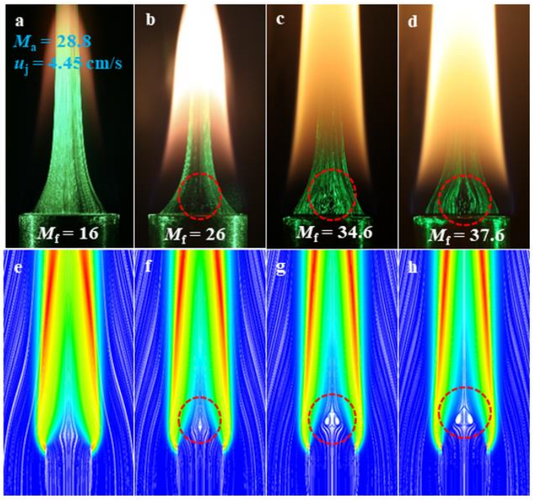

Figure 4 Flow characteristics for selected fuel molar masses showing experimental (a–d) and numerical results (e–h): u_j = 4.45 cm/s, M_f = (16 ~ 46.2) g/mol (fuel), M_a = 28.8 g/mol (air), and u_c = 6.2 cm/s.

For the low fuel density as methane, no recirculation zone was found. When we increase fuel density, the circulation zone comes out and gets larger with higher fuel density. The agreement of numerical and experimental results attributes the recirculation zone in the flow stream to the increased fuel density.

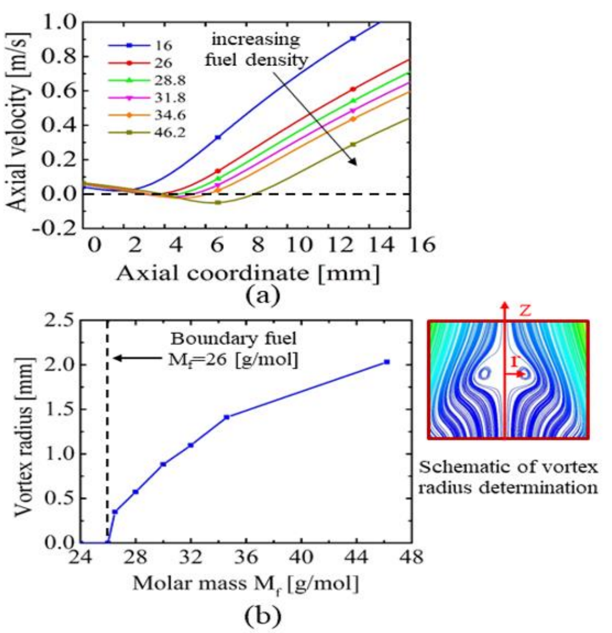

Figure 5. Numerical results with various molar mass; u_j = 4.45 cm/s, M_f = (16 ~ 46.2) g/mol (fuel), M_a = 28.8 g/mol (air), and u_c = 6.2 cm/s. (a: profiles of the axial velocity at r = 0) and (b: vortex radius as function of fuel molar mass)

The extraction of the axial velocity profile on the central axis of the nozzle shows that there is the reverse flow near the nozzle tip. We found out the critical fuel density at M_f = 26 g/mol at which the axial velocity decreased to zero then increased again. For the higher fuel density, the magnitude of reverse flow became larger as shown in Figure 5(a). At this critical fuel density, the reverse flow can be observed in numerical results more clearly than that in experiment results. To evaluate the size of the recirculation zone, we performed the measurement of the recirculation zone radius which was determined from its center to the central axis of the nozzle as described in Figure 5(b).

The numerical result of the flow stream near the nozzle tip shows that the size of the recirculation zone increased with the increased fuel density. As the fuel density increased, the negative buoyant force acted on the fuel core by pulling the fuel stream down toward the nozzle. The recirculation zone pushed the radial location of the flame further away from the jet center. Therefore, the elevated velocity flow, caused by the flame, less affects the fuel core. This phenomenon also affects the vortex formation on the flow stream.

3.2 Effect of jet velocity

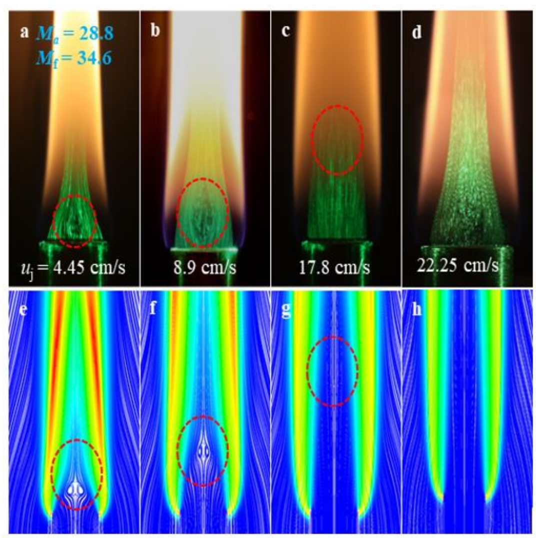

Figure 6. Flow characteristics for selected fuel molar masses showing experimental (a–d) and numerical results (e–f): u_j = (4.45 ~ 22.25) cm/s, M_f = 34.6 g/mol (fuel), M_a = 28.8 g/mol (air), and u_c = 6.2 cm/s.

In previous study [7], the vortex formation can be described by Richardson number Ri = $(\rho_f - \rho_a)gd/(\rho_f u^2_f)$; Ri > 0 for recirculation zone and Ri < 0 for no recirculation zone. Since velocity directly changes the jet momentum which prevails the buoyant force acting on the fuel flow. In this regard, we performed an investigation on various jet velocity (u_j = 4.45, 8.9, 17.8, and 22.5) cm/s for the fuel mixture at M_f = 34.6 g/mol, which showed the vortex formation in both experimental and numerical result. When the jet velocity increased, the recirculation zone lifted up higher and higher and disappeared as shown in Figure 6.

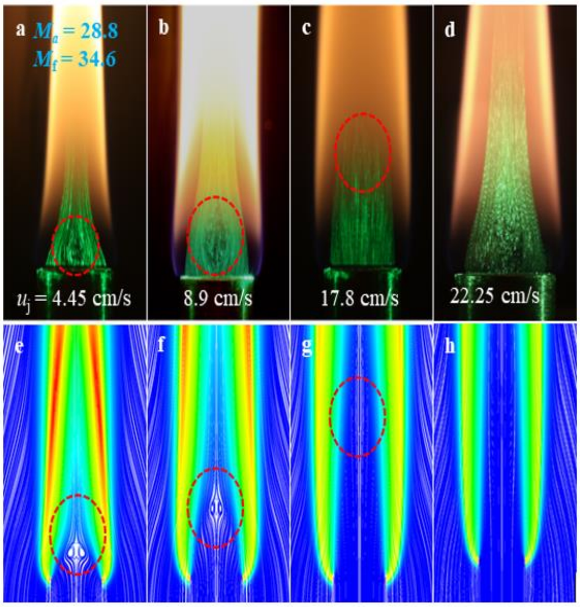

Figure 6 Flow characteristics for selected fuel molar masses showing experimental (a–d) and numerical results (e–f): u_j = (4.45 ~ 22.25) cm/s, M_f = 34.6 g/mol (fuel), M_a = 28.8 g/mol (air), and u_c = 6.2 cm/s.

The axial location of the recirculation zone moved toward the nozzle exit exhibiting the smaller size of the recirculation as compared to that at u_j = 4.45 cm/s and u_j = 8.9 cm/s as shown in Figure 6(a,b) for experiment and Figure 6(e,f) for numerical simulation.

When the jet velocity was increased to 22.5 cm/s, both experimental and numerical results confirmed that the recirculation zone disappeared in Figure 6(d,h). This indicates that there must be a competition between the negative buoyancy acting on the fuel stream and jet momentum pushing it away from the nozzle as it is clearly defined in Ri. Thus, as the fuel density becomes higher, it should require higher jet velocity to prevent the vortex formation. The experimental and numerical results clearly show that jet velocity was the key factor for the modification in the recirculation zone.

3.3 Effect of fuel inlet temperature

Temperature is a parameter that directly changes the fuel density as shown in the ideal gas law equation ($\rho_f = M_f P/RT$). In this aspect, the temperature can contribute to the mechanism of vortex formation. we performed the investigation on various fuel inlet temperature in a range of (300 ~ 500) °K at a fixed jet velocity u_j = 4.45 cm/s, the fuel density (M_f = 44.1 g/mol) of propane was candidate fuel for testing since this fuel density is larger the critical fuel density (M_f = 26 g/mol) and showed the recirculation zone at the room temperature in the previous examination [7]. The streamline obtained from the numerical results of various fuel inlet temperatures (380, 400, 420, 425, and 430) °K showed that the recirculation zone got smaller and smaller when the temperature increased, as shown in Figure 7.

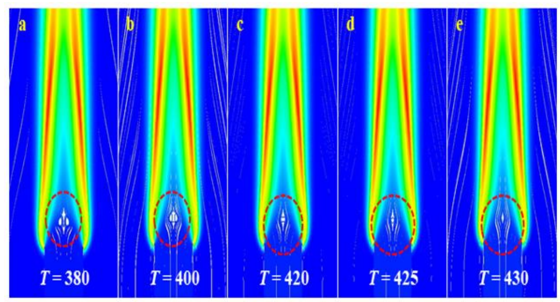

Figure 7 Selected flow characteristics showing the effect of fuel inlet temperature with M_a = 28.8 g/mol (air): (a) T = 380 °K, (b) T = 400 °K, (c) T = 420 °K, (d) T = 425 °K, (e) T = 430 °K.

Since the increase of fuel inlet temperature makes the fuel density become lower, therefore the relative difference between fuel density and air density also becomes smaller. As a result of that, the buoyant force affecting fuel and burnt gas also reduced and made the recirculation zone become much smaller.

A measurement of vortex size on the flow stream obtained from numerical simulation near the nozzle tip shows that the size of the recirculation zone reduced with the increased fuel inlet temperature as shown in Figure 8. As the temperature increased to a critical value, the recirculation zone on the flow field near the nozzle disappeared. As indicated in Figure 8, the boundary temperature for the vortex formation was 425 °K at which lower fuel inlet temperature could form the recirculation zone. On the other hand, we could eliminate the recirculation zone by increasing fuel inlet temperature higher than the critical temperature of 425 °K.

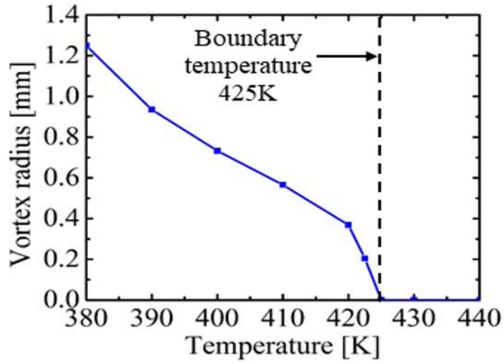

Figure 8 Vortex radius as a function of fuel inlet temperature.

3.4 Effect of ambient air temperature

To clarify the effect of temperature on the vortex formation, we additionally investigated the exhibition of recirculation zones with various ambient air temperatures. We applied the same conditions as described in section 3.3, but fixed the fuel temperature of 300 °K. Then, we conducted numerical simulation for various ambient air temperatures in a range of (300 ~ 500) °K. Figure 9 shows the size of the recirculation zone obtained from the numerical simulation as a function of ambient air temperature. The size of the recirculation zone is sensitive to the temperature change. Thus, changing temperatures changed the ambient density and affected the vortex formation. The recirculation zone gets larger as the temperature increases. Besides fuel density and jet velocity, the temperature is a secondary parameter affecting the vortex formation.

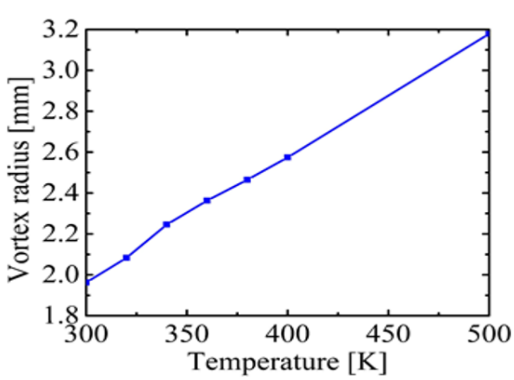

Figure 9 Vortex radius as a function of ambient air temperature

3.5 Effect of nozzle diameter

Regarding the parameters which include in the Richardson number $Ri = (\rho_f - \rho_a)gd/(\rho_f u^2_f)$, nozzle diameter d is a geometry parameter that composing to Ri number. We performed numerical simulations to examine the effect of nozzle diameter on the vortex formation with various nozzle diameters (d = 5.4, 10.8 and 21.6) mm. And we fixed all other conditions (M_f = 34.6 g/mol, M_a = 28.8 g/mol, u_j = 4.45 cm/s, and u_{co} = 6.2 cm/s) at which the recirculation appeared in both the experiment and the simulation when d = 10.8 mm as shown in Figure 2(c,g).

Figure 10 shows that there is no vortex formation for a smaller nozzle (d = 5.4 mm) (Figure 10a), whereas a larger recirculation zone occurred with a larger nozzle (d = 21.6 mm) (Figure 10c). The exhibition of flow stream extracted from numerical simulation clearly shows the effect of nozzle diameter on vortex formation. This result also signifies that nozzle diameter d is an important factor of vortex formation.

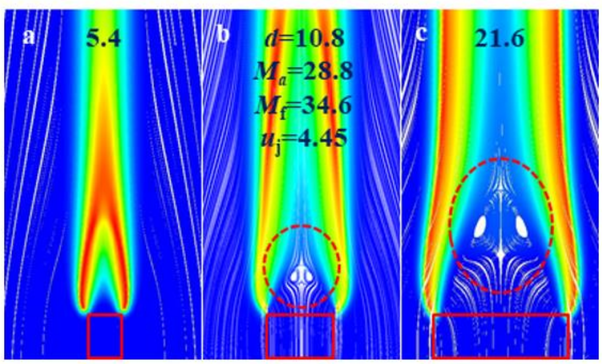

Figure 10 Selected flow characteristics showing the effect of nozzle diameter with u_j = 4.45 cm/s, M_f = 34.6 g/mol (fuel), M_a = 28.8 g/mol (air), and u_c = 6.2 cm/s. (a) d = 5.4 mm, (b) d = 10.8 mm, (c) d = 21.6 mm.

The axial profile of the axial velocity obtained from the numerical simulation shows that, as the nozzle diameter increased, the negative axial velocity increased as shown in Figure 11a. The negative axial velocity formed the negative buoyancy which pulled the fuel stream down and created the recirculation zone. A measurement of the recirculation zone size shows that the radius of the recirculation zone increased with increased nozzle diameter as shown in Figure 11b.

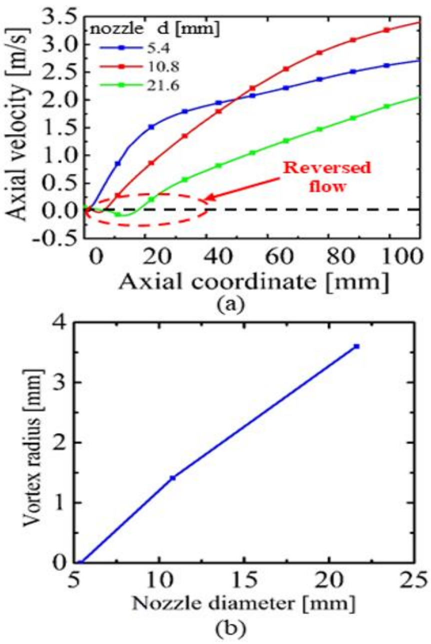

Figure 11 Numerical results with various nozzle diameter; d = (5.4 ~ 21.6) mm. (a: profiles of the axial velocity at r = 0) and (b: vortex radius as function of nozzle diameter).

4 Conclusion

We experimentally and numerically performed the investigation on the effect of fuel density and jet velocity on the vortex formation. Both experiment and numerical simulation results agreed well as they exhibited the recirculation zone in the same boundary conditions. We investigated the effect of temperature and nozzle diameter on the recirculation zone. The prediction of the recirculation zone can be attributed to the variables of the Richardson number (Ri). The results indicated that the fuel density and jet velocity and nozzle diameter are the primary impacts on the vortex formation while the effects of temperature and chemical reaction are secondary.

The findings in this study provided the solutions, by control the parameters of fuel density, fuel jet velocity, and nozzle diameter, to remove the recirculation zone and stabilize the fuel flow stream of coflow flame. The results provide the guidlines to remove the vortex flow and stablize the coflow diffusion flame.

Acknowledgement

This research is funded by Nguyen Tat Thanh University, Ho Chi Minh City, Viet Nam under grant number 2025.01.123.

References

1. K.T. Kang, J.Y. Hwang, S.H. Chung, W. Lee. (1997). Soot zone structure and sooting limit in diffusion flames: Comparison of counterflow and co-flow flames. *Combustion and Flame*, 109 (1-2), 266-281. https://doi.org/10.1016/S0010-2180(96)00163-0

2. D.R. Snelling, F. Liu, G.J. Smallwood, Ö.L. Gülder. (2004). Determination of the soot absorption function and thermal accommodation coefficient using low-fluence LII in a laminar coflow ethylene diffusion flame. *Combustion and Flame*, 136 (1-2), 180-190. https://doi.org/10.1016/j.combustflame.2003.09.013

3. M.D. Smooke, R.E. Mitchell, D.E. Keyes. (1986). Numerical Solution of Two-Dimensional Axisymmetric Laminar Diffusion Flames. *Combustion Science and Technology*, 67 (4-6), 85-122. https://doi.org/10.1080/00102208908924063

4. J. Buckmaster, N. Peters. (1988). Flame Stability. *Proceedings of the 21st (International) Symposium on Combustion,* 21, 1829-1836.

5. T. Yuan, D. Durox, E. Villermaux. (1994). An analogue study for flame flickering. *Experiments in Fluids*, 17, 337-349. https://doi.org/10.1007/BF01874414

6. R.W. Davis, E.F. Moore, W.M. Roquemore, L.D. Chen, V. Vilimpoc, L.P. Goss. (1991). Preliminary Results of a Numerical-Experimental Study of the Dynamic Structure of a Buoyant Jet Diffusion Flame. *Combustion and Flame*, 83 (3-4), 263-270. https://doi.org/10.1016/0010-2180(91)90074-L

7. Y. Xiong, M.S. Cha, S.H. Chung. (2015). Fuel density effect on near nozzle flow field in small laminar coflow diffusion flames. *Proceedings of the Combustion Institute*, 35 (1), 873-880. https://doi.org/10.1016/j.proci.2014.06.025

8. M.S. Cha, J.W. Son, S.H. Yoon, H.T. Luong, D.A. Lacoste. (2019). Vortex formation mechanism within fuel streams in laminar nonpremixed jet flames. *Combustion and Flame*, 199, 46-63. https://doi.org/10.1016/j.combustflame.2018.10.015

9. ANSYS Inc. (2017). ANSYS Free Student Software.

10. E.F. Tarrazo, A.L. Sánchez, A. Linan, F.A. Williams. (2006). A simple one-step chemistry model for partially premixed hydrocarbon combustion. *Combustion and Flame*, 147 (1-2), 32-38. https://doi.org/10.1016/j.combustflame.2006.08.001

11. E.F. Tarrazo, M. Vera, A. Lina'n. (2006). Liftoff and blowoff of a diffusion flame between parallel streams of fuel and air. *Combustion and Flame*, 144 (1-2), 261-276. https://doi.org/10.1016/j.combustflame.2005.07.012

Nghiên cứu bằng mô phỏng và thực nghiệm trường dòng ngọn lửa gần đầu vòi phun nhiên liệu

Dương Hoàng Long[1], Trương Văn Tiến[1], Đặng Quốc Cường[1], Phan Quang Định[1], Âu Thị Kim Loan[2], Hồng Đức Thông[3], Lương Hùng Truyện[1,*]

[1]Trường Đại học Nguyễn Tất Thành, Việt Nam
[2]Trường Đại học Sư phạm Kỹ thuật, Thành phố Hồ Chí Minh, Việt Nam
[3]Trường đại học Bách Khoa, Thành phố Hồ Chí Minh, Việt Nam
[*]lhtruyen@ntt.edu.vn

Tóm tắt Việc tạo ra vùng tuần hoàn trong trường dòng sử dụng vòi phun nhiên liệu đã được những nghiên cứu gần đây cho thấy có ảnh hưởng đến độ ổn định của ngọn lửa và sự hình thành bồ hóng. Nghiên cứu này tiến hành cả thí nghiệm và mô phỏng số để làm rõ các cơ chế cơ bản của sự hình thành vùng tuần hoàn trong ngọn lửa khuếch tán dòng chảy tầng. Các mô hình mô phỏng số được thiết lập bằng cách kết hợp nhiều thông số khác nhau, bao gồm mật độ nhiên liệu, vận tốc vòi phun, nhiệt độ nhiên liệu và nhiệt độ không khí xung quanh. Các hình ảnh trường dòng của dòng chảy gần vòi phun cho thấy các thông số này ảnh hưởng lớn đến sự phát triển của vùng tuần hoàn. Kết quả mô phỏng đã được xác thực dựa trên dữ liệu thực nghiệm. Trong số các thông số được nghiên cứu, mật độ nhiên liệu, vận tốc tia phun và đường kính vòi phun là những yếu tố chính ảnh hưởng đến dòng tuần hoàn, trong khi yếu tố ảnh hưởng phụ là nhiệt độ nhiên liệu và không khí môi trường xung quanh vòi phun. Phân tích chi tiết từng thông số đã làm sáng tỏ các điều kiện quan trọng mà tại đó vùng tuần hoàn hình thành.

Từ khóa ngọn lửa khuếch tán, lực nổi, ngọn lửa phản lực, sự tuần hoàn.