Sphinx Journal of Mathematical Sciences

1(1), 24–39, 2026

DOI: http://dx.doi.org/10.64389/sjms.2026.011104

https://sphinxsp.org/journal/index.php/sjms/

Research article

Investigation concentric flow conical burner mixing field by species transports model

Mohamed k. Hasanin[a*], Mohy.S. Mansour[b]

a. Higher institute of engineering, El shrouk city, Cairo, Egypt
b. Mechanical Power Engineering Department, Faculty of Engineering, Cairo University, Egypt

* **Correspondence:** mkhalidmmh@gmail.com

ARTICLE INFO

Keywords:
Combustion stability, combustion modelling, concentric flow burner, species-Transport -model, flow field velocity, reverse flow circulation

Mathematics Subject Classification:
68Q04, 76B75

Important Dates:
Received: 1 December 2025
Revised: 25 December 2025
Accepted: 29 December 2025
Online: 1 January 2026

ABSTRACT

This study investigates flame stabilization in a concentric flow conical burner operating under partially premixed combustion conditions. The burner configuration was designed to generate controlled inhomogeneous partially premixed environments with improved stability characteristics. Experimental measurements and numerical simulations based on the species transport model were conducted to evaluate the effects of mixing level, Reynolds number, and equivalence ratio on combustion performance. The numerical model was validated using highly resolved Particle Image Velocimetry (PIV) measurements, demonstrating good agreement with experimental observations. Flow-field structures, velocity distributions, and temperature profiles were analyzed to assess the influence of the conical geometry on flame stabilization mechanisms. A new parameter, (A_mix/D^2), was introduced to quantify the ratio of the mixture-region area to the total mixing-field area, including recirculation zones. Results showed that increasing axial distance enhanced mixing and flame stability. The most stable operating condition was achieved at XD = 1 and LD = 10, where symmetric velocity and temperature distributions were maintained. Methane exhibited greater stability than hydrogen due to its lower reactivity.

1. Introduction

Combustion is a chemical process in which different types of fuels react with specific oxidizers to produce heat and high temperatures. Depending on the mixing conditions between the fuel and oxidizer, combustion can generally be classified into three main types. The first type is fully premixed combustion, where the fuel and oxidizer are completely mixed prior to ignition. The second type is non-premixed combustion, where the fuel reacts with a surrounding oxidizer without prior mixing; this mode is often referred to as diffusion combustion. The third type is partially premixed combustion, where partial mixing occurs before the reaction. This mode hasbeen widely reported to exhibit improved stability compared with the other combustion regimes [1]. Achieving stable combustion is a key objective in many industrial applications. Among the different combustion modes, partially premixed combustion has been identified as one of the most stable mechanisms [1,2]. Consequently, numerous research groups have developed various burner configurations with different stabilization techniques, such as piloted flames and reverse-flow burners [3,4]. To obtain better control of combustion characteristics, new burner designs have been proposed that allow adjustment of the mixing level using simple operational parameters. One such design is capable of producing combustion powers up to 250 kW while maintaining a compact burner size [5]. This burner consists of two co-axial tubes, an inner and an outer tube, and incorporates a conical nozzle at the exit to enhance flame stabilization. The influence of the nozzle geometry on flame structure and stability has therefore been investigated in several studies [6]. The proposed burner design allows control of several important parameters, including the degree of premixing (L/D), Reynolds number (Re), and equivalence ratio (Φ). By integrating a conical quartz tube, it became possible to apply advanced laser-diagnostic techniques. For instance, Planar Laser-Induced Fluorescence (PLIF) of the CH radical has been used to characterize flame structures and compare experimental results with computational simulations using Large Eddy Simulation (LES) models [7]. Another diagnostic technique, Laser-Induced Plasma Spectroscopy (LIPS), has been employed to measure the local equivalence ratio in turbulent partially premixed combustion systems [8]. Most previous investigations were conducted using methane–air mixtures, which highlights the importance of studying burner performance with low-calorific- value fuels. Compared with jet flames, conical flames exhibit lower sensitivity to variations in fuel type, jet velocity, and stoichiometric conditions [9]. Optimization of the conical burner geometry, particularly the premixing degree and cone angle, has been investigated using LPG fuel, and the results indicated that flame stability improves as the cone angle decreases [10]. To better understand the stabilization mechanism inside the conical section, measurements of the velocity field using Particle Image Velocimetry (PIV) and temperature measurements using thermocouples have been performed. These studies demonstrated that the dominant stabilization mechanism arises from the formation of a reverse flow region near the inner wall of the cone [11]. However, local flame extinction remains a significant challenge for combustion modeling. Experimental investigations showed that increasing the premixing air in the fuel stream leads to the growth of local extinction zones, eventually resulting in flame blowout [12]. Further studies examined the flow field at the burner exit without the conical nozzle using Rayleigh scattering measurements, while also evaluating the influence of co-flow conditions. The results showed that co-flow enhances flame stability, making this burner configuration attractive for industrial applications. Additionally, Rayleigh diagnostics enabled identification of the most stable operating conditions in terms of L/D, Φ, and Re [13, 14]. Velocity fields have also been analyzed using PIV measurements, which revealed radial and axial velocity profiles for propane fuel with co-flow. These measurements confirmed that the reversed flow near the cone walls generates vortical structures of different scales that enhance flame stability [15]. In parallel, numerous studies have investigated combustion characteristics in different burner geometries using Computational Fluid Dynamics (CFD) models. For example, CFD simulations comparing propane and methane fuels under non-premixed combustion conditions demonstrated that propane can provide higher efficiency for certain burner configurations [16]. Other numerical studies examined methane combustion to evaluate the influence of flow mixing on flame temperature and CO_2 formation, enabling optimization of operating conditions [17]. The effects of Reynolds number and equivalence ratio in hydrogen–air combustion have also been investigated, showing that premixed flames tend to stabilize closer to the burner compared with non-premixed flames, particularly in swirl-stabilized burners [18]. Additional CFD studies have explored the influence of Reynolds number and equivalence ratio in propane–air premixed combustion using a Swiss-roll combustor, where simulations based on the k–ε turbulence model and reduced chemical kinetics indicated that the highest temperatures occur in the swirl region [19]. Comparative studies involving hydrogen, methane, and propane fuels revealed that hydrogen can produce the most stable flames in micro- scale burners [20]. Furthermore, numerical analysis of partially premixed methane–air combustion in a two-dimensional burner configuration showed that the maximum temperature occurs in the combustion zone and gradually decreases toward the outlet [21]. From the reviewed literature, it can be concluded that the Conical Flow Control Burner (CFCB) can operate under highly turbulent conditions and with different types of fuels. This burner design provides considerable flexibility in controlling the degree of mixing and equivalence ratio. The addition of a conical nozzle significantly improves flame stability, while the presence of a co-flow system further enhances turbulence intensity. Moreover, previous CFD investigations across different burner geometries indicate that the most influential parameters include fuel type, Reynolds number, equivalence ratio (Φ), flow velocity field, and temperature distribution. Therefore, the objective of the present work is to develop a comprehensive understanding of the CFCB burner configuration in order to address existing research gaps also to extract quantitative parameter can describe mixing field in conical region

1. Experimental set up

The burner consists of two coaxial stainless-steel tubes. The inner tube has an inner diameter of 4mm and a wall thickness of 1 mm, while the outer tube has an inner diameter of 9.7 mm with the same wall thickness [1]. The inner tube can be moved vertically, allowing precise control of the mixing length (L). This configuration also provides flexibility to interchange the fuel and air inletsdepending on the experimental requirements. In some experiments, the flow field at the burner exitwas investigated without the conical section, where measurements were performed only 2 mm above the burner tip to study the exit flow structure [11]. However, to perform a more comprehensive analysis of the flow field using advanced laser diagnostic techniques such as Particle Image Velocimetry (PIV) and to achieve a more stable flame, a quartz conical section canbe attached to the burner outlet, as illustrated in Fig. 1. The half-cone angle of this section was previously optimized and selected to be 26°, while the mixing length (L) was optimized at 5 mm.The conical section has a height (H) of 70 mm and a nozzle exit diameter of 73 mm [7]. In the present study, the mixing length L is represented in dimensionless form as L/D, while the cone height H is expressed as X/D. The present investigation is conducted using a two-dimensional analysis, where variations in the flow field are examined along the radial direction (r) and the axialdirection (x).

(a) Dimensions of burner (b) Represent quartz tube and flame

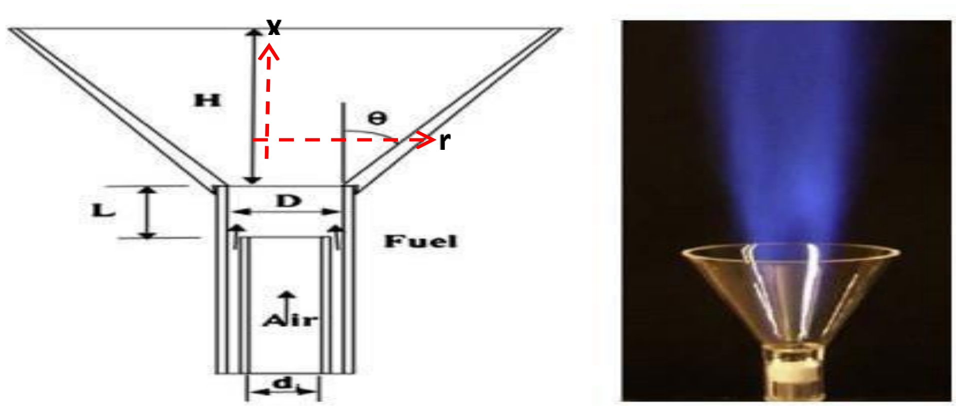


Fig (1):- represent schematic diagram for burner dimension and also with quartz nozzl

2. CFD model description

The computational analysis in this study was performed using ANSYS Fluent, which solves the governing equations of fluid flow using the finite volume method (FVM) for spatial discretization.The main numerical approaches and modeling parameters used in the simulations are summarizedin Table 1. The geometrical dimensions of the burner were described in the previous section. A complete schematic of the computational domain used in the simulations is presented in Fig. 2(a).The burner configuration contains two separate inlets. The inner inlet supplies the fuel, while the outer inlet is used for the air stream. By varying the flow rates at these two inlets, different operating conditions can be achieved, including variations in the Reynolds number (Re) and the equivalence ratio (Φ). The mixing process begins inside the outer tube when the fuel and air streams interact. This mixing continues along a defined mixing length (L/D) before reaching the nozzle exit, where combustion is initiated. Therefore, investigating the influence of the mixing length (L/D) and the behavior of the mixture as it passes through the nozzle at different axial positions (X/D) represents a key objective of this study. For mesh generation, an unstructured triangular mesh with smooth transition and inflation layers near the walls was applied to accuratelycapture the flow behavior, as illustrated in Fig.2 (b). A mesh sensitivity analysis was also conducted to ensure the independence of the numerical results from mesh resolution. To improvemesh quality, the computational domain was divided into three main regions during the meshing process, as shown in Fig. 3, which allows better control of element distribution and improves numerical accuracy. Initially, a mesh containing approximately 55,000 elements and 70,000 nodeswas tested; however, the predicted flow characteristics were still influenced by the mesh density. Consequently, further refinement was performed until mesh-independent results were achieved. The final computational grid consisted of approximately 168,600 elements and 170,000 nodes, which was adopted for the CFD simulations in this study as shown in Fig.3 (d).

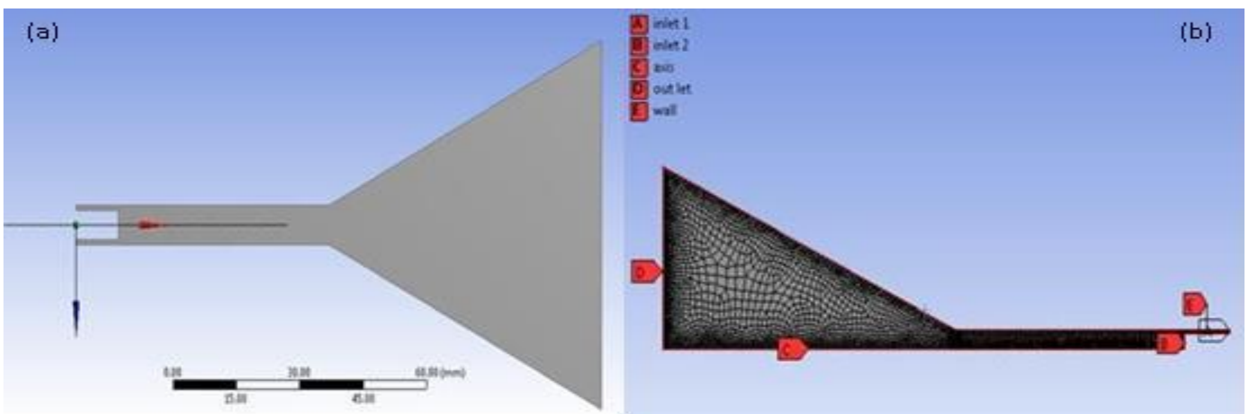
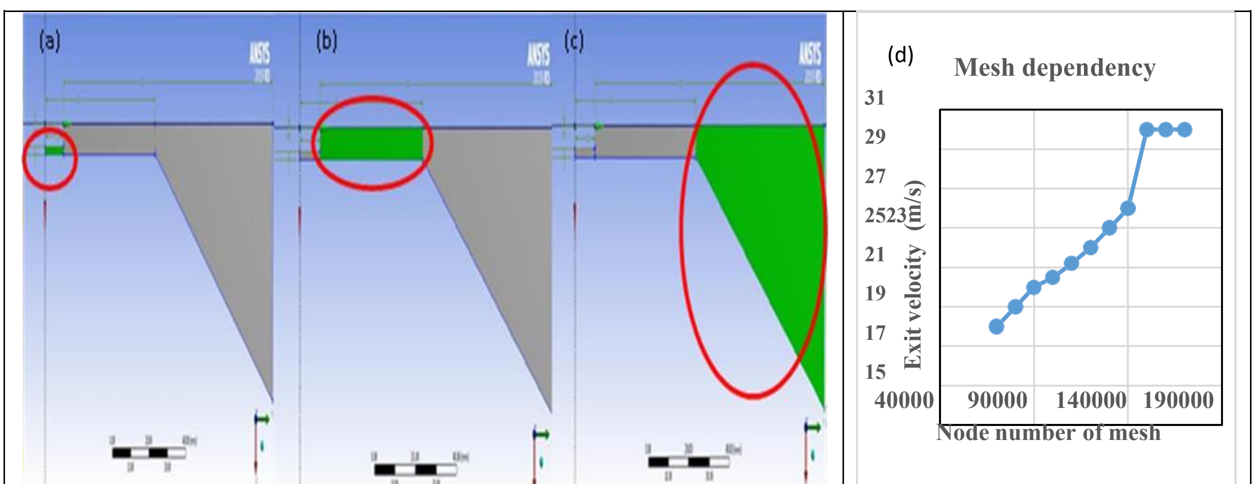


Fig (2):- (a)-geometry of burner with conical section, (b) mesh and inputs ofstudy



Fig(3):- (a)-represent the inlet of fuel (b) representthe inlet of air and mixing length LD (c) represent the conical section with certain XD,(d) mesh sensitivity

2.1. Model and governing equations

The numerical simulations were performed using a pressure-based solver combined with the k– epsilon turbulence model to represent turbulent flow behavior. The governing transport equationsfor turbulent kinetic energy and its dissipation rate used in the simulations are presented in Eqs.
(1) and (2), as reported in [22]. To describe the transport and interaction of chemical species duringcombustion, the Species Transport Model was employed. This model enables tracking of the convection, diffusion, and reaction of chemical species within the flow field. The governing equation for species transport is provided in Eq. (3). Accurate representation of the interaction between turbulence and chemical reactions was achieved using the Eddy Dissipation Model, whichrelates the reaction rate to the turbulent mixing rate of reactants. The formulation of this model involves two governing expressions, presented in Eqs. (4) And (5), following the approach described in [23–24]. This method is particularly suitable for evaluating the turbulent reaction rateRi in combustion systems. Mass diffusion of chemical species in the numerical simulations was modeled according to Fick's Law of Diffusion, which is expressed in Eq. (6). However, because the present study deals with turbulent flow conditions, the turbulent diffusion formulation was adopted. Therefore, the modified diffusion expression shown in Eq. (7) was used to account for the combined effects of molecular and turbulent diffusion, as described in [23–24].

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b + \rho \mathcal{E} - Y_M + Y_k \quad (1)$$

$$\frac{\partial}{\partial t}(\rho \mathcal{E}) + \frac{\partial}{\partial x_j}(\rho \mathcal{E} u_i) = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial k}{\partial x_j}\right] + C_{1\mathcal{E}}\,\frac{\mathcal{E}}{k}\,(G_k + C_{2\mathcal{E}} G_b) - C_{2\mathcal{E}}\,\rho\,\frac{\mathcal{E}^2}{k} + S\mathcal{E} \quad (2)$$

$$\frac{\partial}{\partial t}(\rho Y_i) + \nabla.(\rho \vec{v} Y_i) = -\nabla.\vec{J_i} + R_i + S_i \quad (3)$$

$$R_{i,r} = V'_{i,r}\, M_{w,i}\, A\, \rho\, \frac{\varepsilon}{k} \min_R\left(\frac{Y_R}{V'_{R,r}\, M_{w,R}}\right) \quad (4)$$

$$R_{i,r} = V'_{i,r}\, M_{w,i}\, AB\, \rho\, \frac{\varepsilon}{k} \frac{\sum_P Y_P}{\sum_J^N V''_{J,r}\, M_{w,J}} \quad (5)$$

$$\vec{J_i} = -\left(\rho D_{i,m} \nabla Y_i + D_{T,i}\, \frac{\nabla T}{T}\right) \quad (6)$$

$$\vec{J_i} = -\left(\rho D_{i,m} + \frac{\mu_t}{Sc_t}\right) \nabla Y_i - D_{T,i}\, \frac{\nabla T}{T}) \quad (7)$$

The turbulence characteristics in the present simulation were modeled using the k–epsilon turbulence model, which is widely applied in combustion simulations because of its robustness and relatively low computational cost. The model constants and transport coefficients were selected according to the standard values commonly reported in the literature. These parameters include the empirical turbulence constants, turbulent Prandtl numbers for kinetic energy, dissipation rate, and energy transport, in addition to the turbulent Schmidt number for species diffusion. To represent the turbulence–chemistry interaction in the combustion process, the EddyDissipation Model was adopted. This model assumes that the chemical reaction rate is controlled primarily by the turbulent mixing rate of reactants. The empirical constants associated with the model were also selected from standard recommended values to ensure stable and reliableprediction of flame behavior in the partially premixed combustion system considered in this studyas shown in table (2).

Table (2):- illustrate solving equation constants

(k-ϵ) model parameters	Magnitude
C_μ	0.09
$C_{1\varepsilon}$	1.44
$C_{2\varepsilon}$	1.92
PrTKE	1
PrTDR	1.3
PrEnergy	0.85
PrWall	0.85
Sct	0.7
Eddy dissipation parameters	
A	0.4
B	0.5

3. Results and conclusion

3.1. Stability and model validation with (PIV)

In the present study, the primary controlled parameter is the degree of mixing, which is physicallyrepresented by the dimensionless ratio L/D. The flow velocities of both the inner fuel stream and the outer air stream can be independently adjusted. These velocities may also be expressed in termsof their corresponding Reynolds numbers, namely the inner Reynolds number and the outer Reynolds number. However, for simplification and consistency in the analysis, a mixture Reynoldsnumber calculated based on the jet velocity is used to characterize the flow conditions. In addition,the relative proportion between the fuel and air flow rates is defined by the equivalence ratio (Φ),which represents the fuel–air ratio of the jet mixture. An important aspect of this study is the evaluation of burner stability in order to identify a representative operating condition suitable for CFD modeling. A stability map of the burner, shown in Fig. 4, was constructed to determine stable flame regions. Based on this map, a stable operating point was selected at an equivalence ratio ofΦ=3and a Reynolds number of Re=8214, with a mixing length ratio of L/D=5. Validation of the numerical model with experimental data represents a critical step in the present investigation. Particle Image Velocimetry (PIV) measurements revealed the presence of a reverse flow region near the conical wall, where a portion of the air stream moves upstream and forms vertical structures. This recirculation zone contributes to a self-stabilization mechanism of the flame, enhancing the overall stability of the combustion process. The comparison between numerical predictions and experimental velocity measurements is presented in Fig. 5. The strong agreement between the two datasets indicates that the numerical model successfully reproduced the main flowstructures observed in the experiments. In particular, the accurate prediction of the recirculation zone confirms that the selected turbulence–combustion models effectively captured the relevant interaction mechanisms. This validation step was essential to establish the reliability of the computational framework before proceeding with further parametric investigations. Furthermore,different values of L/D were investigated at a fixed X/D location to evaluate their influence on theflow and combustion characteristics. The effects of equivalence ratio (Φ) and Reynolds number were also analyzed for different fuel types. The detailed boundary conditions and solver settingsused in the numerical simulations are summarized in Tables 3 and 4, respectively.

Table (3):- illustrate solving system and boundary conditions

Parameters	Solving scheme
Scheme used	couple
Spatial Discretization	
Pressure	Second-Order Upwind
Momentum	Second-Order Upwind
Turbulent Kinetic Energy	Second-Order Upwind
Turbulent Dissipation Rate	Second-Order Upwind
CH4/O2/CO2/H2O/Energy	Second-Order Upwind
Solution Initialization	Hybrid Initialization

Table (4):- boundary conditions of burner

Parameters	Conditions
A-inlet (1)	Velocity inlet
B- inlet(2)	Velocity inlet
C- axis	Axis symmetry
D-out-let	Pressure out let with ,Backflow Turbulent Intensity: 5% Backflow Turbulent Viscosity Ratio: 10%
E- wall	Stationary Wall with Standard Wall Roughness

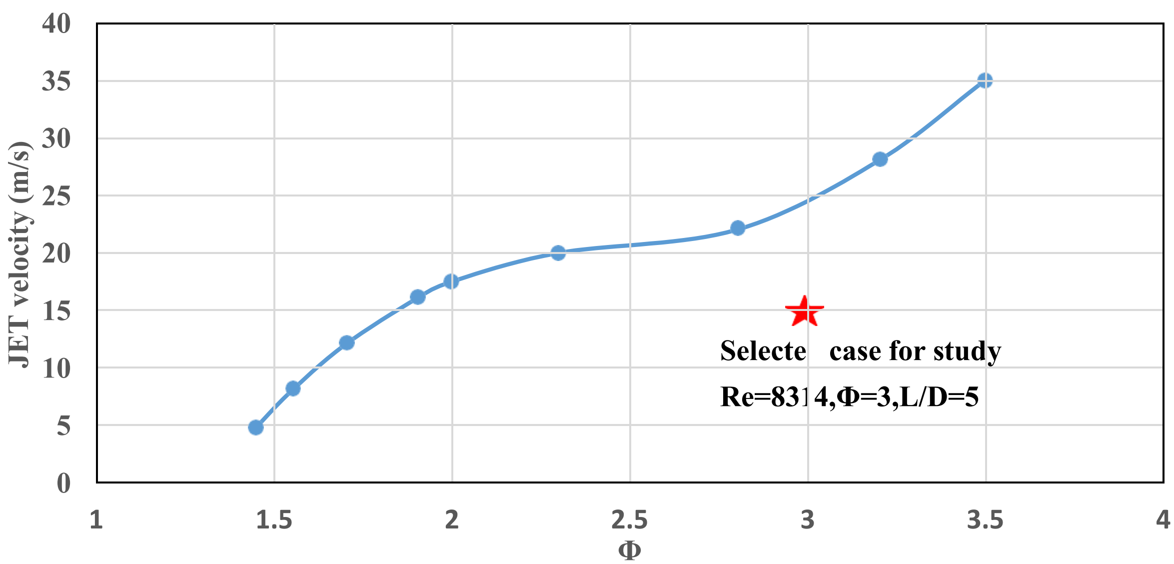


Fig (4):- represent the stability curve of CFCB at LD=5, the red circle represent selected casestudy

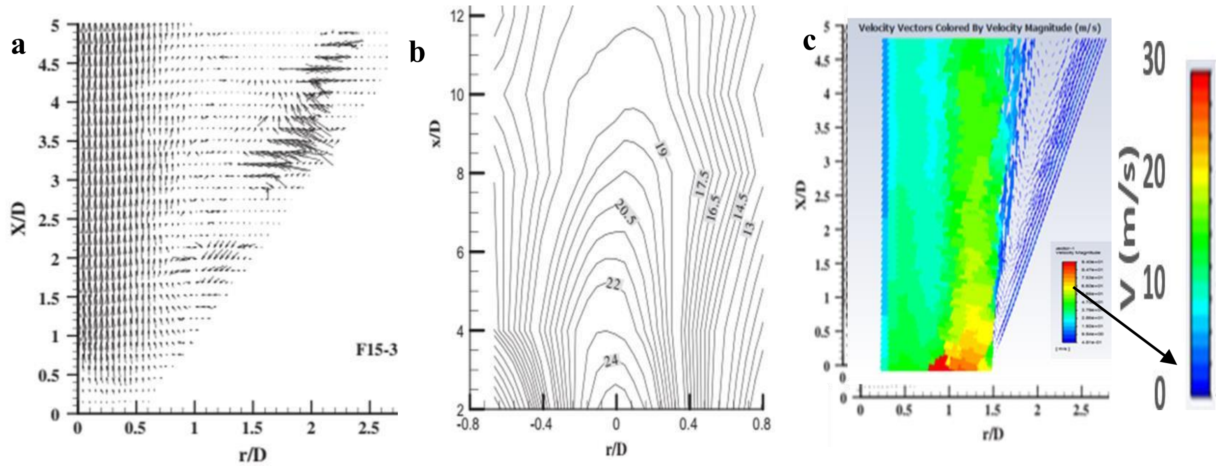


Fig (5):- (a) illustrate experimental PIV recirculation eddies that make stabilization mechanismin burner, (b) represent stream lines contours for same region (c): represent the shape of same region extracted from CFD results

4.2 different effects of parameters

After the validation stage, a series of numerical simulations was conducted to examine the effectsof the main operating parameters. Different mixing lengths (L/D) were considered while keepingthe nozzle exit position (X/D) constant to evaluate the influence of the fuel–air interaction lengthon flame stabilization and combustion behavior. In addition, the effects of equivalence ratio (Φ) and Reynolds number were systematically investigated for various fuel compositions, providing further insight into the interaction between jet momentum, mixing intensity, and combustion stability. The solver configuration and boundary conditions used in this phase of the study are summarized in Tables 3 and 4.

4.2.1 Flow field velocity effect of LD

At the operating condition of Re = 8314, Φ = 3, and an axial distance of X/D = 0, the influence ofvarying the mixing length (L/D) on the velocity field and temperature distribution was systematically examined, as shown in Fig. 6. The results demonstrate a strong dependence of the near-wall flow structures on L/D. At the smallest mixing length, both velocity and temperature profiles display relatively uniform distributions near the wall. As L/D increases, these profiles develop pronounced peaks, indicating stronger gradients and enhanced mixing within the boundary region. With further increases in L/D, the profiles gradually become flatter again, suggesting a tendency toward flow re-stabilization and increased homogenization. At low mixinglengths (L/D), the inlet velocity profile remains nearly uniform, while species concentrations are either uniform or only weakly stratified. At this stage, the diffusion timescale is relatively short and the boundary layers have not yet significantly developed. Consequently, species transport predicts limited diffusion, resulting in weak momentum exchange and small velocity gradients near the wall. As L/D increases, species diffusion becomes more pronounced, which enhances momentum diffusion and promotes boundary layer growth. This process leads to stronger velocitycurvature, higher wall shear stress, and maximum velocity gradients, indicating intensified mixing within the flow field. The region corresponding to intermediate mixing lengths (approximately L/D = 5–10) therefore represents a peak-gradient zone where mixing and momentum transfer aremost active. At higher mixing lengths (L/D ≈ 14–20), the species transport process approaches a balance between convection and diffusion, and the flow gradually becomes both hydro dynamically and chemically fully developed. As a result, the velocity and species gradients begin to diminish, leading to flatter and more stabilized profiles.

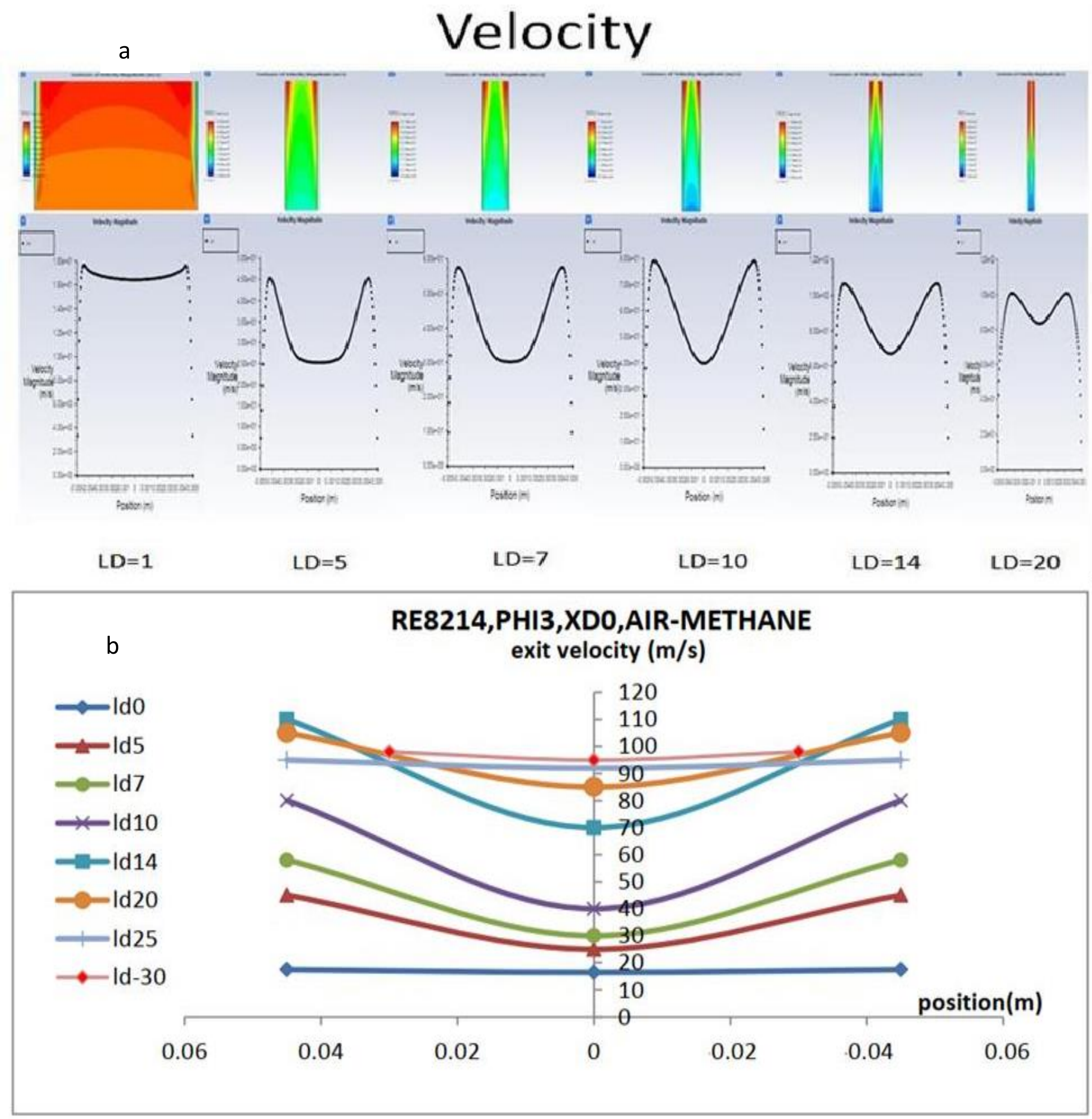


Fig. 6. Exit velocity distribution profiles of the burner at varying mixing lengths (LD).(a).CFD model flow field ,(b) quantitative analysis for mixing field

4.2.2:- Flow field velocity effect of XD

The influence of increasing axial distance XD was examined at Re=8214and Φ=3, with a focuson the conical section and a representative mixing length of LD=5, as illustrated in Fig. 7.The results indicate that, under these conditions, the flow field progressively shifts away from the centerline, producing an asymmetric velocity distribution. This asymmetry can be attributed to pressure imbalance, with reduced pressure on the displaced side and elevated pressure on the opposite side. Furthermore, the deviation occurs more rapidly at LD=10, emphasizing the sensitivity of flow stability to mixing length. To address this, the optimal axial distance that preserved symmetry across different LD values was identified as XD=1. As demonstrated in Fig. 8, the velocity profile remains symmetric even at larger mixing lengths, extending up to LD=10.

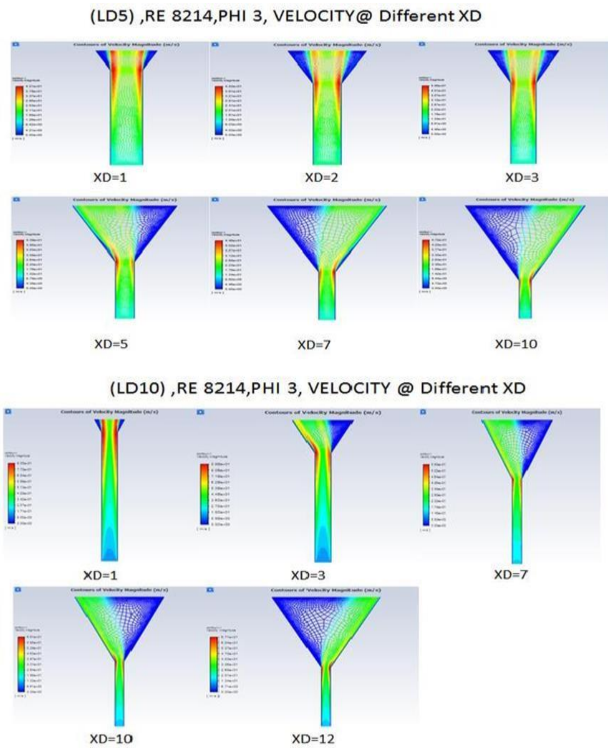


Fig (7):- comparison between different LD =5&10 for different XD

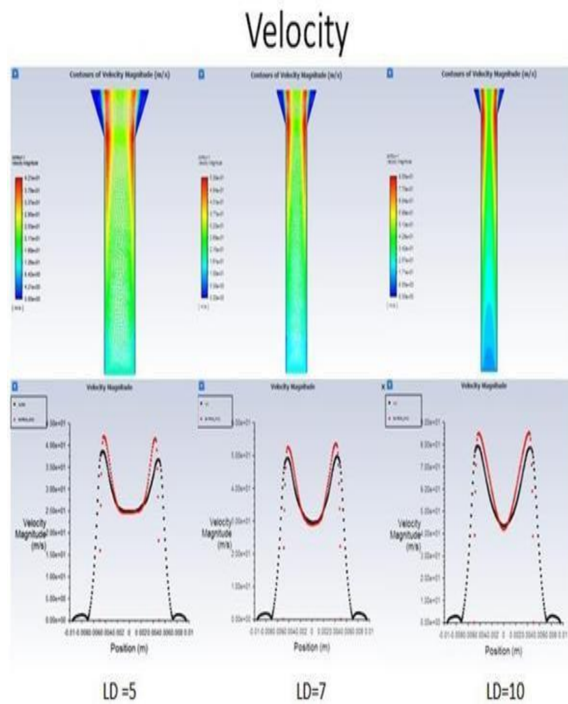


Fig (8):- comparison between different LDat XD=1

For velocity and temperature distribution quantitative analysis via mixing field new parametersare introduced like mixing area ratio also velocity and temperature gradient.as shown in fig 9 area ratio between mixture to total mixing field area increase linearly with XD for different LDalso its noticeable that for LD=10 after XD =10 this ratio become constant that's means reverseflow due to conical section become constant also higher ratio mean lower reverse flowstabilization. Velocity gradient near wall and velocity at center line also are calculated for different LD and XD as shown in fig (10.a) it's noticeable that by increasing XD mixture regionarea decrease for different LD. In fig (10b) clear proof that adding cone emphasize the stabilityby forming reverse flow region behave like stabilizer for flame.

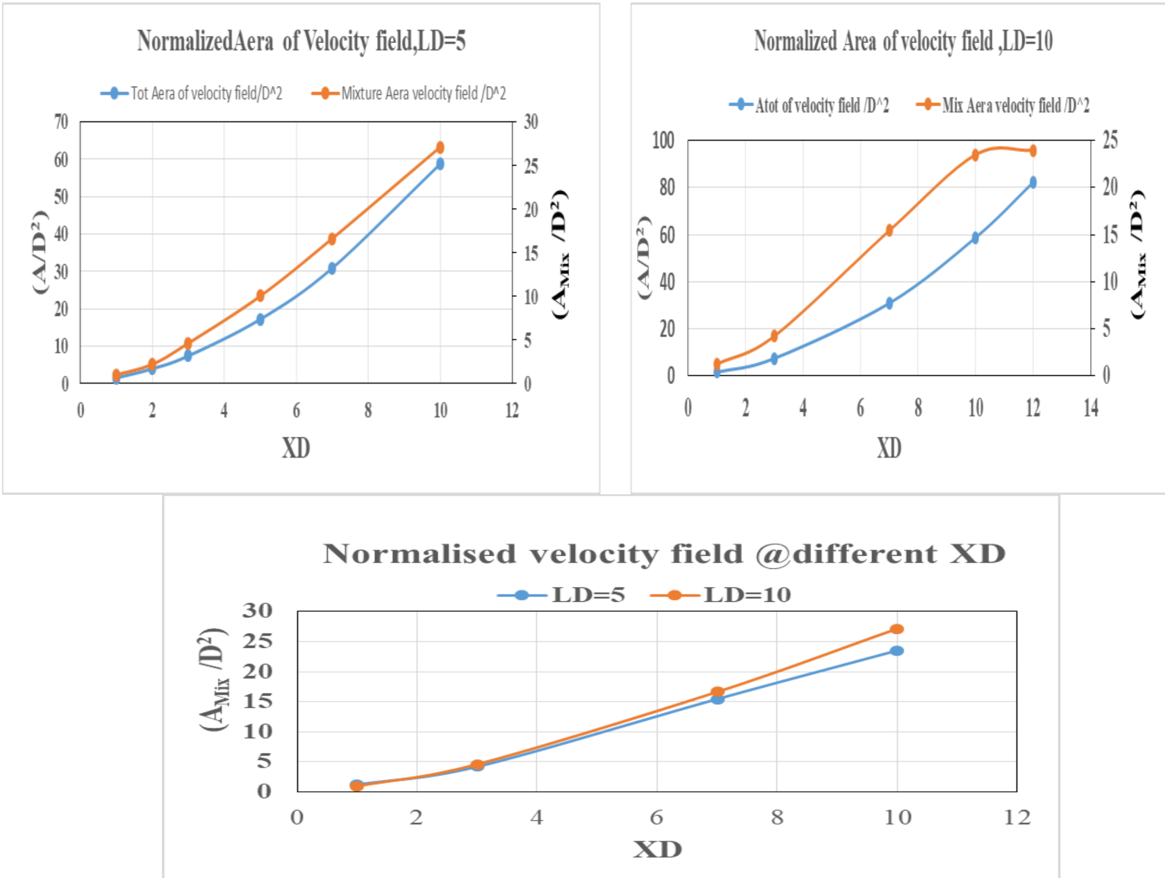


Fig (9) represent mixture area percent according total mixing field for different LD =5,LD=10

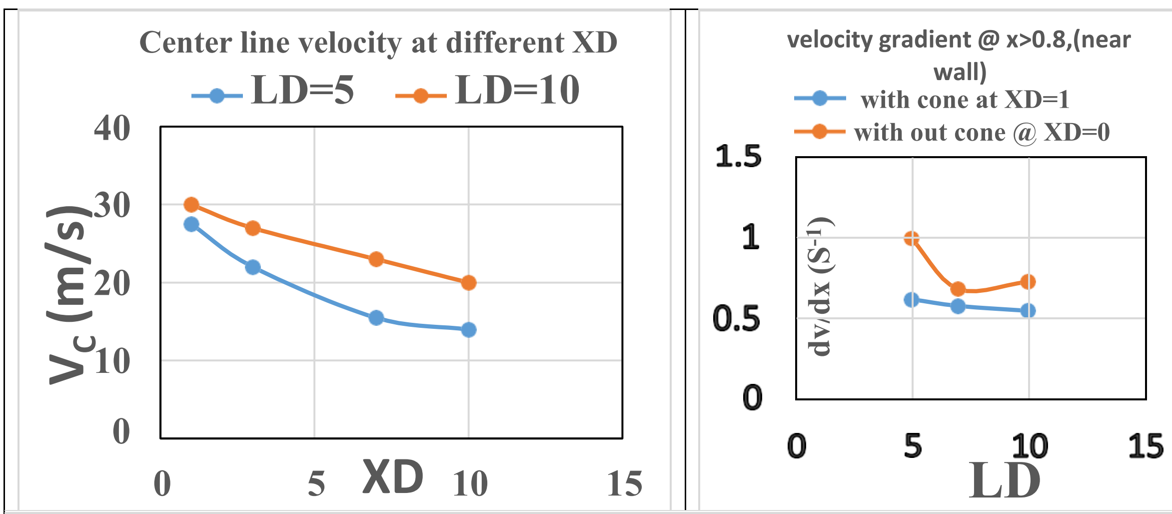


Fig 10. (a) center velocity for different XD,(b) gradient of velocity near wall for different LD with and without cone

4.2.3. Flow field temperature effect of LD

Figure 11 presents a comparison of temperature distributions for L/D 5, 7, and 10 in the combustion system. The results indicate that increasing the length-to-diameter ratio enhances thermal uniformity within the flow field. Higher L/D values produce smoother and more symmetric temperature profiles, accompanied by reduced temperature gradients and fewer localized hot spots. The contour maps further illustrate improved heat distribution and diminished edge cooling as L/D increases, which reflects enhanced mixing and greater flame stability. Among the examined cases, the L/D=10 configuration exhibits the most uniform temperature field, suggesting superior combustion performance. Overall, these observations demonstrate that longer nozzles facilitate more stable and efficient combustion by enabling morecomplete flow development and improved heat retention, making them advantageous forapplications targeting high efficiency and reduced emissions.

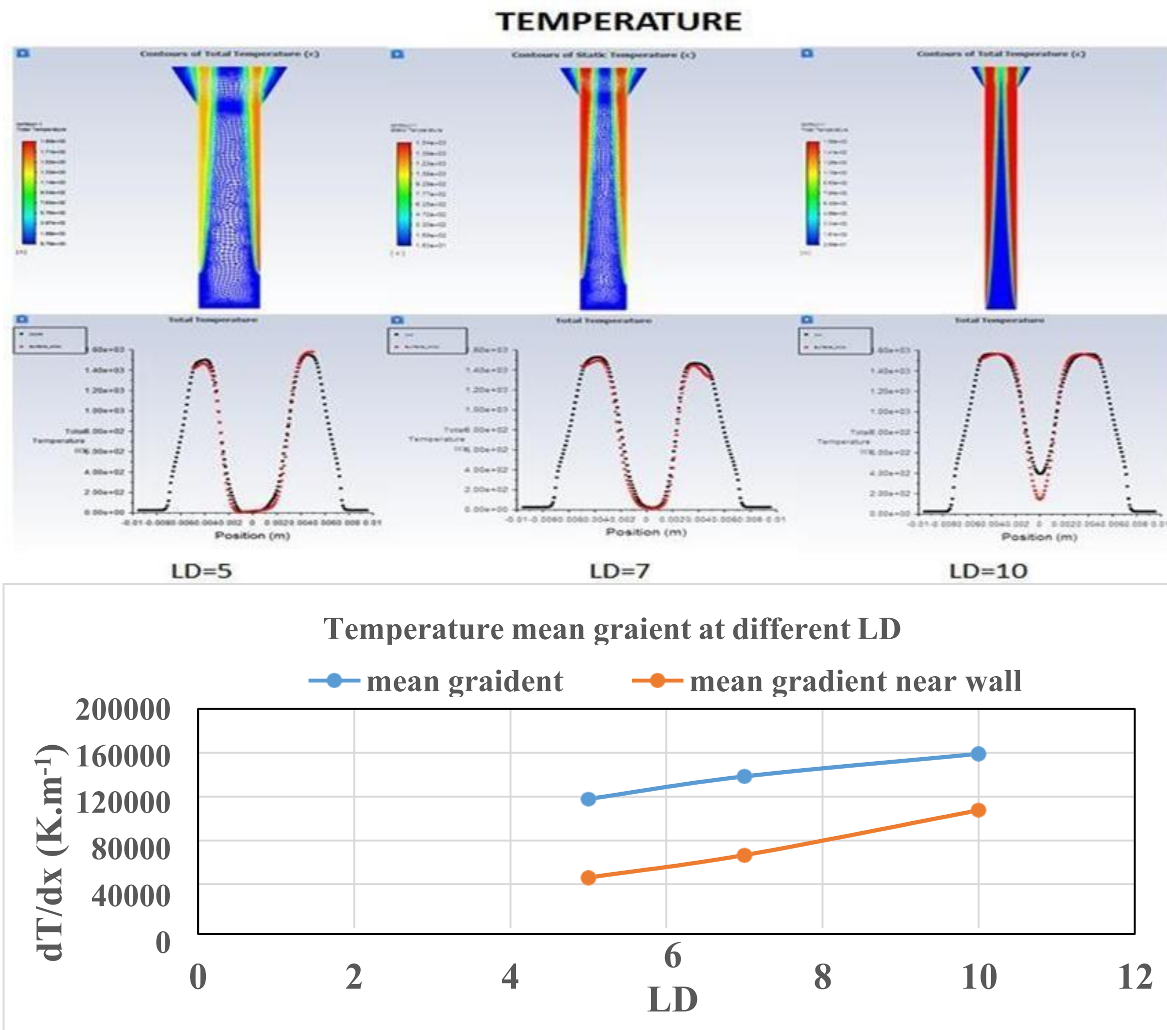


Fig (11). Illustrate temperature profiles distribution for XD=1& different LD

4.3. Flow field velocity and temperature effect of fuel type

According fuel comparison figure 12 presents a comparative investigation of methane–air and hydrogen–air combustion in a coaxial burner, emphasizing the influence of mixing length L/D andaxial distance X/D on velocity and temperature distributions. Methane, the primary constituent ofnatural gas, was adopted as the reference fuel because of its extensive use in combustion studies. At moderate L/D values, methane produced relatively stable flame structures; however, as the mixing length increased, the velocity and temperature profiles developed noticeable central depressions and asymmetries, indicating the formation of recirculation regions and a reduction inflame stability. In contrast, hydrogen exhibited greater uniformity and robustness over a broader range of geometric conditions. Although its lower density and higher laminar flame speedgenerated steeper velocity and temperature gradients at short L/D, hydrogen maintained more symmetric and continuous distributions at larger mixing lengths, thereby avoiding the instabilitiesobserved in methane flames. These characteristics suggest that hydrogen can perform more effectively in systems with longer residence times, where stable mixing and efficient combustion are required. Despite these advantages, hydrogen's high reactivity introduces practical challenges,particularly its strong propensity for flashback. This phenomenon promotes upstream flame propagation, which can destabilize the combustion process, especially in cone-stabilized burners.Therefore, while hydrogen presents clear benefits as a low-carbon alternative to methane, its application necessitates careful burner design and appropriate control strategies to minimize flashback risks. Overall, the results underscore hydrogen's potential as a sustainable energy carrier, while highlighting the importance of optimizing burner geometry, operating conditions, and stabilization mechanisms to fully realize its combustion benefits.

4. Conclusion

This study investigated flame stabilization in a Concentric Flow Conical Burner (CFCB) through an integrated experimental and computational methodology. Although numerous experimental studies have been conducted on this burner configuration, computational analyses remain relatively scarce. The present results demonstrate that the species transport model can serve as an effective and computationally economical diagnostic approach, successfully reproducing velocity and temperature distributions in close agreement with Particle Image Velocimetry (PIV) measurements. The conical burner geometry was found to induce recirculation zones that enhanceflame stabilization by generating reverse flow patterns analogous to those produced by swirl. Flame stability was strongly dependent on the operating parameters, particularly the Reynolds number (Re), equivalence ratio (Φ), mixing length (LD), and axial position (XD). The most stableoperating condition occurred at XD = 1 with LD = 10, where symmetric velocity and temperatureprofiles were maintained. A comparison of fuels indicated that methane exhibited greater flame stability than hydrogen due to its lower chemical reactivity, although the numerical model capturedthe behavior of both fuels with good accuracy.

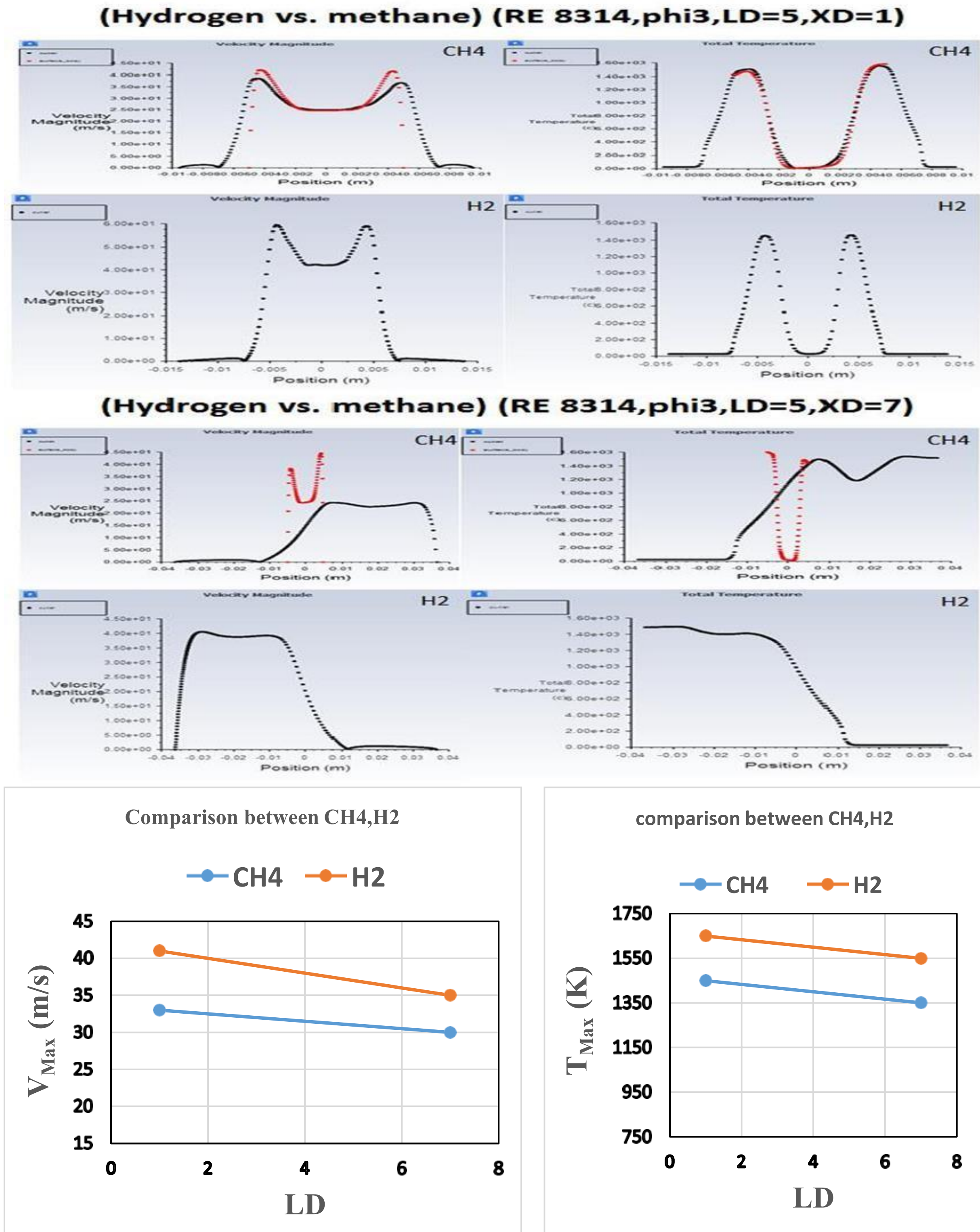


Fig (12).fuel type effect ,comparison between velocity and temperature for hydrogen andmethane

References:-

[1] R.W. Bilger, combust. Sci .Technol 13(1976) 155-170.

[2] Peters, N. 2000. Turbulent Combustion Cambridge University Press Cambridge,UK.

[3] Mansour, M.S et al, (1989) A Reverse Flow Reactor for Turbulence/ChemistryInteraction Studies, Combustion Science and Technology, 65:1-3, 83-101

[4] Masri, A.R. 2015. Partial premixing and stratification in turbulent flames. Proc.Combust. Inst., 35, 1115–1136

.

[5] M.S.Mansour, Combust.Sci.Technol152 (2000)115–145.

[6] Masri, A.R. 2015. Partial premixing and stratification in turbulent flames. Proc.Combust. Inst., 35, 1115–1136

[7] Fawzy El-Mahallawy, Ahmed Abdelhafez Mohy S. Mansour (2007) Mixing and Nozzle Geometry Effects on Flame Structure and Stability, Combustion Science And Technology, 179:1-2, 249-263

[8] B. Li et al. / Proceedings of the Combustion Institute 32 (2009) 1811–1818

[9] M.S. Mansour et al. / Spectrochimica Acta Part B 64 (2009) 1079–1084

[10] B. Yan et al. / Experimental Thermal and Fluid Science 34 (2010) 412–419

[11] Mansour, M.S., Elbaz, A., and Samy, M. 2012.The stabilization mechanism ofhighly stabilized partially premixed flames in a concentric flow conical nozzle burner. Exp. Therm. Fluid Sci., 43, pp. 55–62.

[12] E. Baudoin Flow Turbulence Combust (2013) 90:269–284

[13] A.M. Elbaz et al. The flow field structure of highly stabilized partially premixed flames in a concentric flow conical nozzle burner with co-flow, Exp. Therm. FluidSci. (2015)

[14] M.S. Mansour et al., Effect of the mixing fields on the stability and structure of turbulent partially premixed flames in a concentric flow conical nozzle burner,Combustion and Flame (2016)

[15] Elbaz, A.M., Experimental Thermal and Fluid Science (2018), https://doi.org/10.1016/j.expthermflusci.2018.01.010

[16] A.S. Nair, B. Mohan Krishna and S. Ajith Kumar, Study of non – Premixed combustion of propane and methane using CFD,Materials Today: Proceedings, https://doi.org/10.1016/j.matpr.2020.09.713

[17] Guessab Ahmed, Aris Abdelkader, Abdelhamid Bounif, Iskander Gökalp Reduced chemical kinetic mechanisms: Simulation of turbulen non-premixed CH4-air flame, Jordan J. Mech. Ind. Eng., Volume 8 Number 2, April. 2014 ISSN 1995-6665.

[18] Wang, Shixuan, Zili Yuan, and Aiwu Fan. "Experimental investigation on nonpremixed CH4/air combustion in a novel miniature Swiss-roll combustor." Chemical Engineering and Processing-Process Intensification 139 (2019): 44-50.

[19] Aikun Tang, Yiming Xu, Chunxian Shan, Jianfeng Pan, Yangxian Liu, A comparative study on combustion characteristics of methane, propane and hydrogen fuels in micro-combustor,

School of Energy and Power Engineering, Jiangsu University, Zhenjiang 212013, China.

[20] Lucky Anetor, Edward Osakue, Christopher Odetunde, Reduced mechanism approach of modeling premixed propane-air mixture using ANSYS fluent, Eng. J.
Volume 16 Issue 1

[21] Mr. Ankit Rajak, Mr. Amit Datta, Mr. Vishwajeet Kureel, Analysis of combustion characteristics of methane/air mixture in coaxial combustor, Int. J. Adv.
Res. Sci. Eng.,Vol.no.5, Issue-2015

[22] A.S. Nair, B. Mohan Krishna and S. Ajith Kumar, Study of non – Premixed combustion of propane and methane using CFD,Materials Today: Proceedings, https://doi.org/10.1016/j.matpr.2020.09.713

[23] Wan, Jianlong, Yongjia Wu, and Haibo Zhao. "Excess enthalpy combustion of methane-air in a novel micro non-premixed combustor with a flame holder and preheating channels." Fuel 271 (2020): 117518.

[24] Mikulčić, Hrvoje, Jakov Baleta, Xuebin Wang, Jin Wang, Fengsheng Qi, and Fan Wang. "Numerical simulation of ammonia/methane/air combustion using reduced chemical kinetics models." International Journal of Hydrogen Energy (2021).

[25] ANSYS Modeling and Meshing Guide002114 November 2004 ANSYS, Inc. is aUL registered ISO 9001: 2000 Company