บทความวิจัย

การวิเคราะห์การไหลและสมรรถนะการเผาไหม้ของหัวเผาแก๊สชนิดฝักบัวที่ใช้เชื้อเพลิง LPG และ CNG ด้วยการจำลอง CFD

อาทิตย์ แสงโสภา อนิรุตต์ มัทธุจักร์* และ ธนรัฐ ศรีวีระกุล
ห้องปฏิบัติการการประยุกต์ใช้ลำเจ็ตและการเผาไหม้ ภาควิชาวิศวกรรมเครื่องกล คณะวิศวกรรมศาสตร์ มหาวิทยาลัยอุบลราชธานี
สุทธิศักดิ์ พงศ์ธนาพาณิช
ภาควิชาเทคโนโลยีวิศวกรรมเครื่องกล วิทยาลัยเทคโนโลยีอุตสาหกรรม มหาวิทยาลัยเทคโนโลยีพระจอมเกล้าพระนครเหนือ

* ผู้นิพนธ์ประสานงาน โทรศัพท์ 0 4535 3309 อีเมล: Anirut.m@ubu.ac.th DOI: 10.14416/j.kmutnb.2025.10.001
รับเมื่อ 4 พฤษภาคม 2568 แก้ไขเมื่อ 26 มิถุนายน 2568 ตอบรับเมื่อ 21 กรกฎาคม 2568 เผยแพร่ออนไลน์ 24 ตุลาคม 2568

บทคัดย่อ

ความต้องการพลังงานอย่างยั่งยืนที่เพิ่มขึ้นในอุตสาหกรรมเซรามิกของประเทศไทยส่งผลให้เกิดความจำเป็นในการเพิ่มประสิทธิภาพการทำงานของหัวเผาโดยใช้เชื้อเพลิงที่สะอาดกว่า แม้ว่าแก๊สปิโตรเลียมเหลว (LPG) จะเป็นเชื้อเพลิงที่นิยมใช้อย่างแพร่หลายเนื่องจากค่าความร้อนที่สูงแต่แก๊สธรรมชาติอัด (CNG) ให้ข้อดีในด้านการปล่อยมลพิษที่ต่ำกว่าและต้นทุนที่ลดลง งานวิจัยนี้ศึกษาลักษณะการเผาไหม้และการไหลของหัวเผาเซรามิกชนิดฝักบัวที่ใช้เชื้อเพลิง LPG และ CNG โดยใช้การจำลองพลศาสตร์ของไหลเชิงคำนวณ (CFD) ควบคู่กับการทดลองจริงเพื่อเพิ่มสมรรถนะของหัวเผาการจำลอง CFD ดำเนินการในสองขั้นตอน ได้แก่ การทดสอบแบบเย็นเพื่อวิเคราะห์การผสมเชื้อเพลิงกับอากาศ และการทดสอบแบบร้อนเพื่อศึกษาพฤติกรรมการเผาไหม้ โดยใช้โปรแกรม FLUENT 2021 R2 ร่วมกับโครงข่าย Tetrahedro และแบบจำลองความปั่นป่วน RNG k–ε การทดลองยืนยันผลดำเนินการที่ความดันเชื้อเพลิง 4 ปอนด์ต่อตารางนิ้ว โดยใช้เทอร์โมคัปเปิลชนิด เค (Thermocouple K-type) สำหรับวัดอุณหภูมิ ผลการทดลองสอดคล้องกับการจำลองโดยมีความคลาดเคลื่อนเฉลี่ยของความเร็ว 6.52% และอุณหภูมิ 5.30% การเผาไหม้ของ LPG ให้ค่าอุณหภูมิสูงสุดมากกว่า 1,375.61 เคลวิน เมื่อเทียบกับ CNG 1,332.83 เคลวิน ขณะที่ CNG แสดงความเร็วการไหลสูงกว่าจากความหนาแน่นที่ต่ำกว่า LPG ยังให้ความสม่ำเสมอของอุณหภูมิและความเข้มของการเผาไหม้ที่ดีกว่า ซึ่งเป็นสิ่งสำคัญต่อกระบวนการเผาเซรามิกอย่างคงที่ งานวิจัยนี้เน้นย้ำถึงความสำคัญของการออกแบบหัวเผาอย่างเหมาะสมและการประยุกต์ใช้ CFD เพื่อส่งเสริมการใช้เชื้อเพลิงสะอาดเพิ่มประสิทธิภาพพลังงานและความยั่งยืนด้านสิ่งแวดล้อมในอุตสาหกรรมเซรามิกขนาดเล็กของประเทศไทย

คำสำคัญ: พฤติกรรมการเผาไหม้ LPG CNG CFD หัวเผาเซรามิก

การอ้างอิงบทความ: อาทิตย์ แสงโสภา, อนิรุตต์ มัทธุจักร์, ธนรัฐ ศรีวีระกุล และ สุทธิศักดิ์ พงศ์ธนาพาณิช, "การวิเคราะห์การไหลและสมรรถนะการเผาไหม้ของหัวเผาแก๊สชนิดฝักบัวที่ใช้เชื้อเพลิง LPG และ CNG ด้วยการจำลอง CFD," *วารสารวิชาการพระจอมเกล้าพระนครเหนือ*, ปีที่ 36, ฉบับที่ 1, หน้า 1–14, เลขที่บทความ 261-067982, ม.ค.–มี.ค. 2569, doi: 10.14416/j.kmutnb.2025.10.001.

Research Article

Flow and Combustion Performance Analysis of LPG and CNG Shower-Type Burners Using CFD Modeling

Arthit Sangsopha, Anirut Matthujak* and Thanarath Sriveerakul
Combustion and Jet Application Research Laboratory (CJARL) Department of Mechanical Engineering, Faculty of Engineering, Ubon Ratchathani University, Ubon Ratchathani, Thailand
Sutthisak Phongthanapanich
Department of Mechanical Engineering Technology, College of Industrial Technology, King Mongkut's University of Technology North Bangkok, Bangkok, Thailand

* Corresponding Author, Tel. 0 4535 3309, E-mail: Anirut.m@ubu.ac.th DOI: 10.14416/j.kmutnb.2025.10.001
Received 4 May 2025; Revised 26 June 2025; Accepted 21 July 2025; Published online: 24 October 2025

Abstract

The increasing demand for sustainable energy in Thailand's ceramic industry has led to the need for improving burner performance using cleaner fuels. Although Liquefied Petroleum Gas (LPG) is commonly used due to its high calorific value, Compressed Natural Gas (CNG) is considered more environmentally friendly and cost-effective. In this study, the combustion and flow characteristics of a shower-type ceramic burner using LPG and CNG were analyzed through Computational Fluid Dynamics (CFD) simulations and validated by experiments. The CFD work was divided into two parts: a cold-flow analysis to examine fuel–air mixing, and a hot-flow simulation to study combustion behavior. ANSYS Fluent 2021 R2 was used with a tetrahedral mesh and the RNG k–ε turbulence model. Experiments were carried out at 4 psi fuel pressure, and a K-type thermocouple was used to measure flame temperature. The simulated results agreed well with the experimental data, showing average deviations of 6.52% in velocity and 5.30% in temperature. LPG produced a slightly higher peak temperature (1,375.61 K) than CNG (1,332.83 K), while CNG showed higher flow velocity due to its lower density. Moreover, LPG provided a more uniform temperature distribution, which is beneficial for consistent ceramic firing. These results suggest that although CNG is cleaner, LPG offers better combustion characteristics under the current burner design. The findings highlight the potential for improving burner design through CFD modeling to support fuel switching in small-scale ceramic industries in Thailand.

Keywords: Combustion Behavior, LPG, CNG, CFD, Ceramic Burners

Please cite this article as: A. Sangsopha, A. Matthujak, T. Sriveerakul, and S. Phongthanapanich, "Flow and combustion performance analysis of LPG and CNG shower-type burners using CFD modeling," *The Journal of KMUTNB*, vol. 36, no. 1, pp. 1–14, ID. 261-067982, Jan.–Mar. 2026 (in Thai), doi: 10.14416/j.kmutnb.2025.10.001.

1. Introduction

In Thailand, Liquefied Petroleum Gas (LPG) remains a primary fuel used across households, commercial kitchens, and industrial burners due to its high calorific value and ease of handling [1]–[3]. However, with growing concerns about energy sustainability and environmental impact, alternative fuels such as Compressed Natural Gas (CNG) have gained attention for their lower emissions and potential cost advantages. Several studies have examined the combustion characteristics of these fuels using various burners. One study investigated LPG combustion in a swirl burner and reported improved thermal efficiency with lower CO emissions [4]. Another developed a high-efficiency natural gas burner, achieving 57% thermal efficiency through design modifications [5]. CFD was also used to evaluate syngas combustion in ceramic kilns, emphasizing how burner geometry affects NO_x formation. These studies reflect ongoing efforts to enhance fuel flexibility, thermal performance, and emission control through integrated experimental and computational approaches [6]–[9]. Notably, studies have demonstrated that the use of natural gas in high-temperature air combustion can enhance combustion characteristics while reducing pollutant formation, suggesting its suitability for industrial furnace applications [10]. Moreover, advancements in porous radiant burner technologies show promise for low-emission and efficient cooking applications [11], [12], while numerical investigations on Synthetic Natural Gas (SNG) compatibility with LPG burners highlight the need for design adaptations to maintain optimal combustion performance [13]. These developments underscore the ongoing transition toward cleaner fuel alternatives and more sustainable combustion practices, which are crucial for Thailand's energy-dependent sectors. Beyond addressing the rising demand for cleaner fuels, it is essential to consider the development of more efficient combustion technologies that promote complete fuel oxidation and minimize pollutant formation. Incomplete combustion not only reduces thermal efficiency but also contributes to elevated emissions of Carbon Monoxide (CO) and unburned hydrocarbons. Advanced burner design strategies particularly those informed by Computational Fluid Dynamics (CFD) have demonstrated potential in optimizing flame stability and fuel–air mixing [14]–[16], thereby supporting both energy efficiency and environmental sustainability.

Despite such progress, several research gaps persist in the context of small-to medium-scale industries, particularly within Thailand's ceramic sector. While numerous studies have examined burner design and performance using natural gas and LPG [17], limited research has focused on adapting these innovations for small ceramic enterprises that often lack access to technological advancements. For instance, although the adaptation of LPG cooking stoves to biogas has demonstrated improved thermal efficiency [18], the conversion process is complex and may not be easily scalable. Similarly, burner designs for natural gas applications have shown high thermal performance achieving up to 57% thermal efficiency at optimal operating pressures [19] but their implementation remains constrained by fuel supply infrastructure and cost. Furthermore, studies on premixed LPG burners highlight the importance of secondary air in stabilizing flames and extending combustion limits [20], while comparative studies indicate that although CNG combustion yields higher efficiency than LPG, it necessitates richer fuel mixtures [21]. Research on burner flow dynamics using CFD has also shown that modifications to burner structure and airflow can significantly enhance thermal efficiency and combustion temperatures [22]–[24]. However, these studies rarely address the specific needs and operational constraints of small-scale ceramic factories in Thailand.

Small-scale ceramic factories in Thailand face design constraints such as basic control systems, fixed gas supply pressures, and limited installation space. These factors require simple, low-cost burner designs that can perform reliably under static and resource-limited conditions.

This study addresses existing research gaps by analyzing the flow dynamics and combustion performance of shower-type burners using CNG, through both Computational Fluid Dynamics (CFD) and experimental validation. Focusing on burner designs common in Thailand's small ceramic enterprises, the research offers practical insights into improving energy efficiency, reducing emissions, and enhancing flame stability. This study combines CFD simulations with experimental analysis to reveal key insights into flow behavior, temperature distribution, and fuel–air interaction. Unlike large-scale burners that operate with high-pressure supply and sophisticated control systems, small burners face constraints such as limited space and simplified setups. The findings highlight how fuel characteristics particularly density and flame speed directly influence performance under these compact operating conditions. informing effective burner redesign. The findings support Thailand's transition to cleaner fuels in underserved sectors and contribute to broader goals of energy efficiency and environmental sustainability.

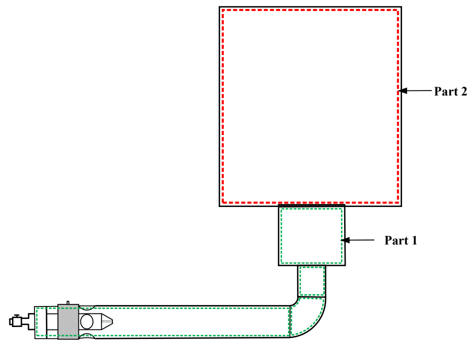


Figure 1 Computational domain used for cold and hot flow analysis of the shower-type burner.

2. Materials and Methods

CFD simulations were performed using ANSYS Fluent 2021 R2 to study fuel–air mixing and combustion in a shower-type ceramic burner. The process was divided into two stages: 1) cold flow for mixing analysis and 2) hot flow for combustion behavior, as shown in Figure 1.

2.1 Cold Flow Simulation (Non-Combustion Case)

The objective of the cold flow simulation (Part 1) is to analyze the mixing behavior of fuel and air within the burner without combustion. This simulation validates the velocity field with experimental results and generates mass fraction and mass flow rate data for key species, including LPG (C_3H_8, C_4H_{10}, N_2, O_2) and CNG (CH_4, C_2H_6, CO_2, N_2), which serve as boundary inputs for the subsequent combustion simulation. The burner's three-dimensional geometry was modeled using actual dimensions and discretized with an unstructured tetrahedral mesh of 1,080,312 elements to capture complex flow behavior. Cold flow simulations were performed using pressure inlet boundary conditions corresponding to thermal loads of 10.74, 17.40, 27.05, and 30.20 kW for both LPG and CNG, consistent with experiments. Primary and secondary air were also set as pressure inlets, the outlet as a pressure outlet, and all walls as no-slip boundaries. Key parameters are summarized in Table 1.

Turbulent flow is modeled using the RNG k–ε turbulence model, with additional species transport modeling to simulate fuel–air mixing without chemical reactions. The fuel composition for LPG is set at 70:30 propane-to-butane by volume, while CNG consists of CH_4:C_2H_6:CO_2:N_2 in a 77:6:14.8:2.2 ratio, as provided by PTT Public Company Limited (2022). The simulation parameters for Part 1 are summarized in Table 1, using a steady-state, pressure-based solver with standard wall functions for near-wall treatment.

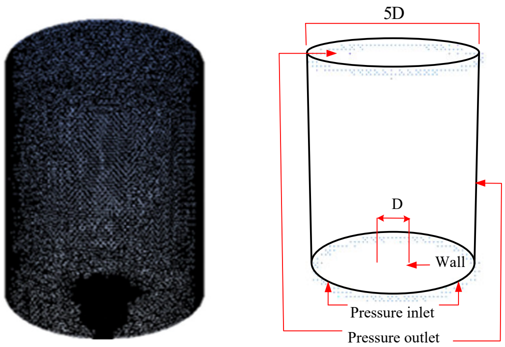
Table 1 Boundary Conditions for Part 1

Boundary Condition	Model
Inlet boundary condition	Air gauge pressure inlet = 0 Pa LPG gauge pressure inlet = 4 psi
Outlet boundary condition	Pressure outlet (air gauge pressure outlet = 0 pa)
Solver	Pressure base
Time	Steady state
Near-wall treatment method	Standard wall function
Turbulence model	RNG k-ε model
Other	Species transport
C_3H_8 : C_4H_{10}	70 : 30

2.2 Combustion Simulation (Hot Test Case)

The hot flow simulation (Part 2) investigated combustion behavior within the shower-type burner and compared numerical results with experimental temperature measurements. This phase accounted for key physical processes including fuel–air mixing, chemical reactions, thermal radiation, and convective-conductive heat transfer. The burner geometry remained consistent with Part 1 and was discretized into 1,486,135 unstructured tetrahedral elements, as illustrated in Figure 2(a). Boundary conditions, including mass flow inlets and pressure outlets, are detailed in Figure 2(b). Mass fractions and flow rates from the validated cold-flow simulation were imposed at the burner inlet, while all walls were defined as no-slip and thermally solid boundaries.

Combustion was modeled using the Eddy Dissipation Model (EDM), which is suitable for turbulence–chemistry interactions under fast-reaction conditions. Radiative heat transfer was captured using the Discrete Ordinates (DO) model, and turbulence effects were treated with the RNG k–ε model. All simulations employed a steady-state, pressure-based solver. Near-wall treatment used the standard wall function approach, with Y^+ values maintained between 30 and 300 to ensure accurate prediction of wall-adjacent flow and heat transfer. A mesh independence study was conducted using coarse (0.88 million), medium (1.48 million), and fine (2.2 million) element grids. The medium grid yielded results within 2% of the fine mesh, balancing accuracy with computational cost, and was therefore selected for all final simulations. These configurations ensured numerical stability and reliable resolution of combustion dynamics.



(a) Mesh (b) Boundary conditions

Figure 2 (a) Mesh and (b) Boundary conditions for the computational domain.

Table 2 Boundary conditions for Part 2

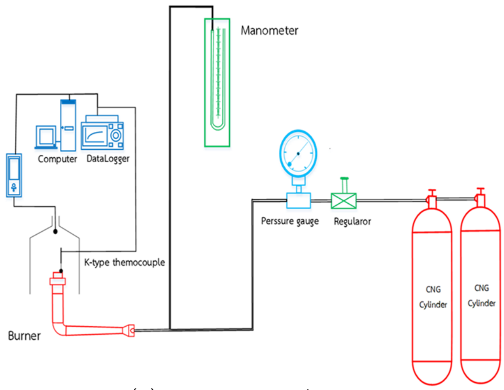
Boundary Condition	Model
Inlet boundary condition	LPG gauge mass inlet Mass fraction of C_3H_8, C_4H_{10}, N_2 and O_2 (part 1)
Outlet boundary condition	Pressure outlet (air gauge pressure outlet = 0 pa)
Solver	Pressure base
Time	Steady state
Near-wall treatment method	Standard wall function
Turbulence model	RNG k-ε model
Radiation model	Discrete Ordinates
Combustion model	Eddy dissipation model

The conditions in Part 2 focus on combustion simulation. Heat transfer in this phase is primarily governed by convection from hot flue gases and combustion products, along with radiation emitted by the flame. To capture turbulent flow behavior at high Reynolds numbers, the RNG k–ε turbulence model was employed. The Eddy Dissipation Model was used to simulate turbulence–chemistry interactions under fast-reaction conditions. Given the significant temperature gradients present, the Discrete Ordinates radiation model was applied to accurately account for radiative heat transfer, in line with previous recommendations in the literature [12]. The corresponding boundary conditions and simulation parameters are summarized in Table 2. Pressure–velocity coupling was handled using the SIMPLE algorithm to ensure stable convergence. Boundary pressures were set to 0 Pa gauge, representing atmospheric pressure, allowing realistic open-boundary flow behavior. This two-stage CFD approach beginning with a non-reactive flow analysis followed by reactive flow simulation ensures both numerical stability and physical accuracy. It also significantly reduces computational time while preserving the fidelity of the flow and combustion phenomena observed in real-world operations. The methodology provides a robust framework for optimizing burner design to enhance thermal efficiency in small-scale ceramic industries using alternative fuels such as CNG

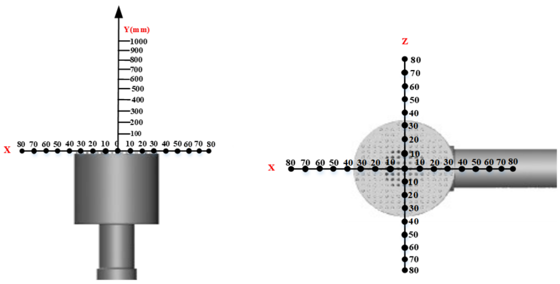
2.3 Temperature Measurement

To validate the accuracy of the CFD simulation results, experimental measurements of the flame temperature were conducted and compared with the numerical data. The experimental setup is illustrated in Figure 3(a). Prior to data acquisition, the burner was ignited and operated at its maximum flame setting for a duration of 15 minutes. This preheating phase was intended to eliminate any residual coating materials or dust within the burner that could otherwise interfere with temperature measurements.

Flame temperature measurements were performed at multiple axial and radial positions, as depicted in Figure 3(b), using a K-type thermocouple



(a) Experimental setup



(b) Temperature measurement position

Figure 3 Experimental setup for measurements validation.

with an accuracy of ±1.1°C or 0.4% of the measured value. A data logger was employed to continuously record the temperature data throughout the experiment. The tests were conducted using LPG as the fuel at a supply pressure of 4 psi, corresponding to a thermal power output of 10.74 kW. All measurements were carried out under ambient room temperature conditions with sufficient ventilation and without the influence of strong external airflows to minimize experimental uncertainties.

3. Results and Discussions

3.1 Comparison of Flow Velocity at Burner Exit

The comparison between CFD simulation and experimental results regarding flow velocity at the

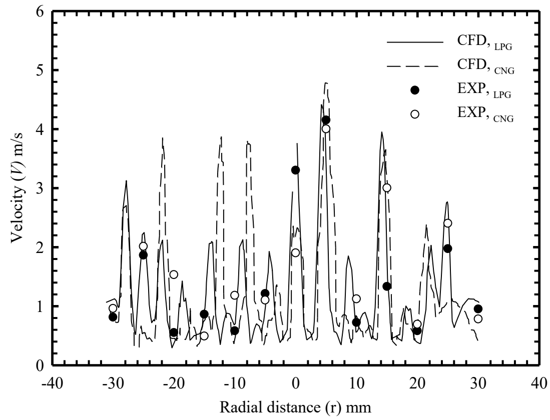


Figure 4 A comparison of the flow velocity.

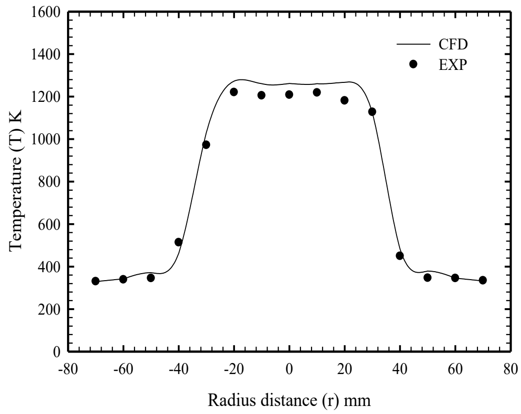
burner exit is presented in Figures 4. The velocity profiles obtained from both CFD and experimental analysis showed similar trends, confirming the accuracy of the model. In particular, lower velocities were observed near the burner wall, increasing progressively toward the center due to jet-like behavior, a pattern consistent with findings reported in [12]. The maximum axial velocity at the burner exit reached 4.4 m/s in the CFD results, compared to 4.0 m/s in the experimental data. The average deviation between CFD and experimental results was less than 6.52% for velocity and 5.30% for temperature. This level of agreement validates the effectiveness of the numerical method. Linear regression showed strong correlation, with R^2 = 0.982 for velocity and 0.976 for temperature, as also supported by [24], who demonstrated that flow field predictions in LPG burners can be accurately modeled using the RNG k–ε approach. Furthermore, the higher velocity consistently observed with CNG is attributable to its lower gas density, which enhances exit velocity under equivalent pressure conditions a trend that aligns with theoretical and empirical studies on gaseous jet flow dynamics [17].

As shown in the combined graph (Figure 4), LPG and CNG exhibit distinct velocity profiles due to their physical properties. CNG, mainly methane, has a lower molecular weight and density than LPG, leading to higher flow velocities under equal inlet pressure. Its simpler molecular structure also reduces viscosity, affecting momentum transfer and jet behavior. These factors explain the consistently higher velocities observed for CNG.

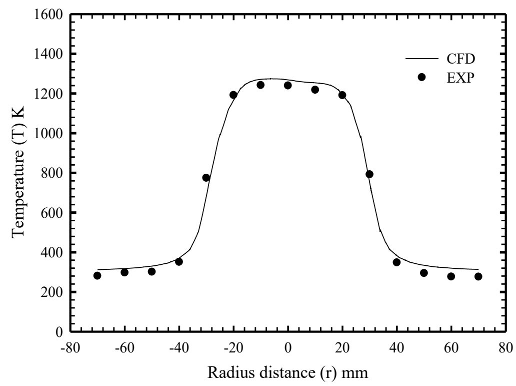
3.2 Comparison of Temperature

The comparison of temperature distributions obtained from CFD simulations and experimental measurements is illustrated in Figures 5(a)–(b), focusing on various radial positions from the center of the burner. The results indicate that the simulated temperature profiles are in good agreement with the experimental data. It was observed that the temperature around the burner is relatively uniform in the near-field region and gradually decreases with increasing radial distance from the burner center. The average deviation between the CFD-predicted temperatures and the measured values did not exceed 5.30%, confirming the reliability of the numerical model in capturing the thermal behavior of the burner under combustion conditions. Such consistency with experimental data is in line with previous CFD validation studies on burner systems [22], [24].

The stable temperature zone corresponds to regions with optimal fuel–air mixing. Peak temperatures occur near the center, where fuel concentration is highest, while temperatures decline radially as gas disperses. LPG's broader high-temperature area indicates more uniform mixing, whereas CNG's



(a) LPG



(b) CNG

Figure 5 Shows the comparison of temperature (a) LPG and (b) CNG.

narrower zone reflects steeper concentration gradients affecting thermal distribution.

The flow field characteristics under non-reacting conditions are visualized in Figure 6, showing velocity vectors at the burner's mid-plane.

3.3 Flow Behavior of Fluid and Combustion

The simulation results presented in Figure 6 illustrate the velocity vector fields at the mid-plane of the burner, corresponding to the cold-flow simulation (Part 1). The velocity distribution indicates

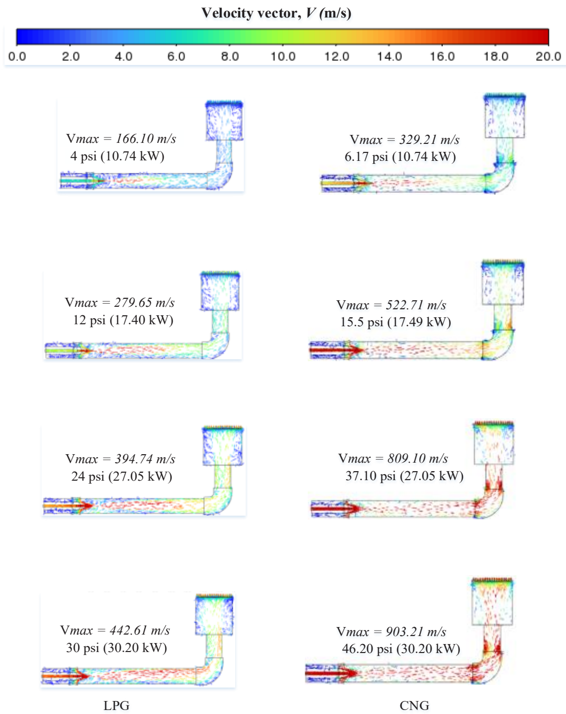


Figure 6 Comparison of simulated velocity distributions for Shower burners at mid-plane.

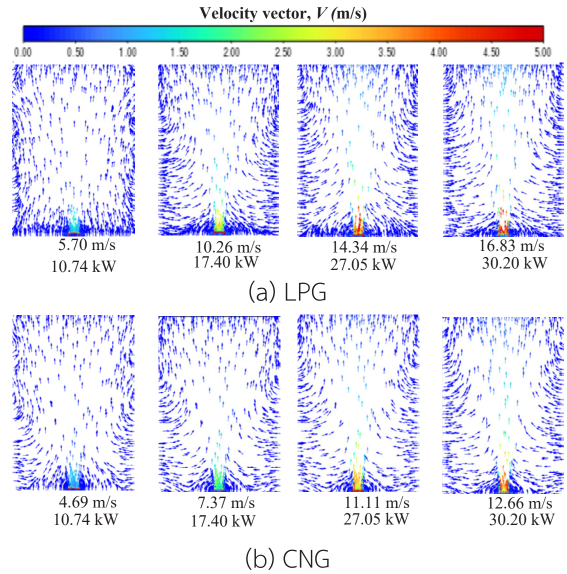


Figure 7 Comparison of the velocity distributions of Shower burner at mid-plane.

that the highest velocities occur in the vicinity of the fuel injector. This is attributable to the release of fuel gas either LPG or CNG under high pressure from the gas supply system, resulting in a pronounced jet effect at the injector outlet for all tested pressure levels. As the high-pressure fuel exits the injector, it entrains primary air into the mixing tube, thereby promoting efficient mixing before combustion. The simulation results further show that velocity magnitudes near the burner exit holes are higher than those in upstream regions. This is the critical region where fuel–air mixing is most intensive, enhancing the conditions necessary for stable combustion [19]. As the input gas pressure increases, a corresponding increase in exit velocity is observed. For LPG, the exit velocities at increasing power levels are approximately 4.3 m/s, 6.7 m/s, 12.2 m/s, and 14.9 m/s, while for CNG, the velocities are higher at 6.3 m/s, 12.7 m/s, 16.8 m/s, and 19.9 m/s, respectively. The consistently greater velocities observed with CNG at all power levels are primarily due to its lower gas density compared to LPG, which results in higher flow velocities under identical pressure conditions [14]. The consistently greater velocities observed with CNG at all power levels are primarily due to its lower gas density compared to LPG, which results in higher flow velocities under identical pressure conditions.

The influence of turbulent mixing on combustion assumes that the chemical kinetics are rapid with respect to the mixing rate. The exothermic reaction releases heat into the flow field, which affects the temperature distribution and flame stabilization.

Figure 7(a)–(b) presents the velocity vector fields at the mid-plane of the burner under varying

LPG and CNG gas supply pressures, corresponding to increasing power outputs. The results indicate that the maximum velocity of the hot combustion gases exiting the burner nozzle reaches 16.83 m/s for LPG at a thermal power of 30.20 kW, whereas for CNG at the same power level, the maximum velocity is slightly lower at 12.66 m/s.

The combustion process generates thermal buoyancy and induces the entrainment of secondary air, which plays a crucial role in supporting and sustaining the combustion reaction within the burner chamber. Regions proximal to the burner where secondary air induction is more effective exhibit enhanced combustion intensity, contributing to more complete oxidation of the fuel. This behavior is consistent with the characteristic flow and flame structures typically observed in shower-type burners, where the interplay between induced air and high-velocity fuel jets promotes stable and efficient combustion. The simulation results presented in Figure 6 illustrate the velocity vector fields at the mid-plane of the burner, corresponding to the cold-flow simulation (Part 1). The velocity distribution indicates that the highest velocities occur in the vicinity of the fuel injector. This behavior is consistent with the characteristic flow and flame structures typically observed in shower-type burners, where the interplay between induced air and high-velocity fuel jets promotes stable and efficient combustion.

To analyze the thermal field under reacting conditions, the temperature contours derived from CFD are presented in Figure 8(a)–(b) shows that, despite similar input conditions, LPG produces a broader, more uniform high-temperature flame, indicating stronger combustion and heat release.

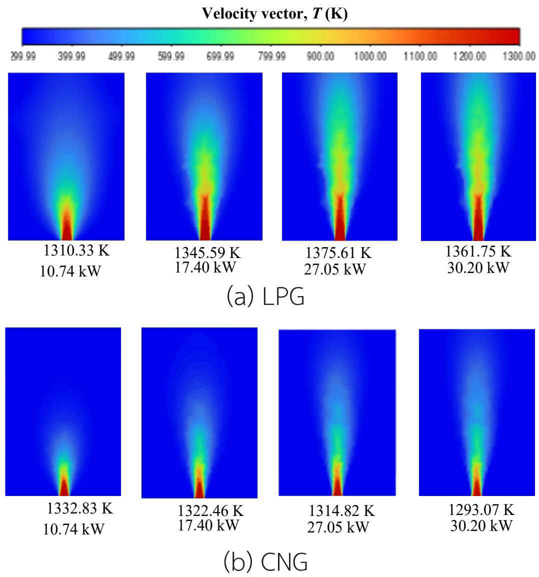
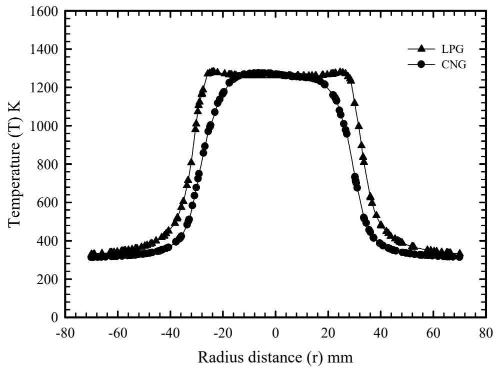


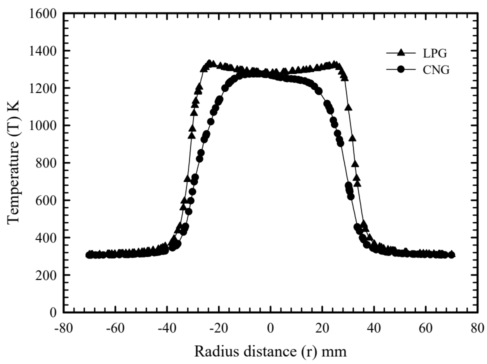
Figure 8 Comparison of temperature contour for Shower burners at mid-plane.

CNG's narrower flame reflects its lower heating value and reactivity. This suggests LPG provides better flame spread and thermal consistency for uniform heating applications.

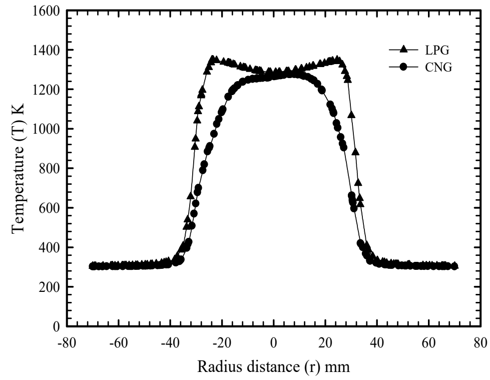
The simulation results comparing temperature distributions at thermal power levels of 10.74, 17.40, 24.05, and 30.20 kW are presented in Figures 9(a)–(d). These results illustrate the radial temperature profiles within the combustion chamber (domain condition) for both LPG and CNG fuels. At all power levels, the maximum temperature, approximately 1330 K, is observed at the center of the burner, indicating this location as the peak combustion zone [24]. As the radial distance increases from the burner center, the temperature progressively decreases, reflecting the dissipation of thermal energy outward from the combustion core. The LPG combustion model reveals a broader high-temperature zone, with a distinct thermal region extending up to a radius



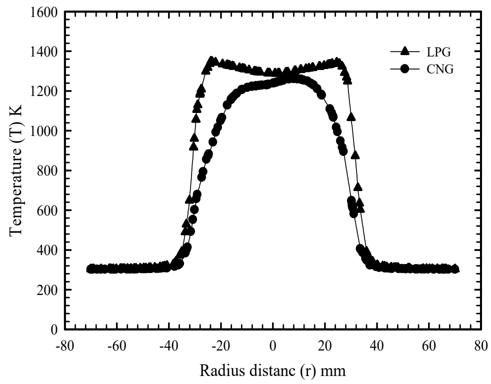
(a) 10.74 kW



(b) 17.40 kW



(c) 27.05 kW



(d) 30.20 kW

Figure 9 Temperature distribution from the model at different power input pressures.

of approximately 40 mm from the burner center. In contrast, the temperature distribution in the CNG combustion model demonstrates a narrower flame radius, with a notable drop in flame temperature beyond 20 mm. This discrepancy is attributed to the higher combustion intensity of LPG, which promotes more effective heat release and broader flame spread [17]. This discrepancy is attributed to the higher combustion intensity of LPG, which promotes more effective heat release and broader flame spread, resulting in superior temperature distribution compared to CNG. Across all tested power levels, the LPG model consistently yields higher flame temperatures than the CNG model. The average temperature deviations between LPG and CNG were calculated to be 14.49%, 12.78%, 12.87%, and 14.63% for 10.74, 17.40, 24.05, and 30.20 kW, respectively. These findings underscore the superior thermal performance of LPG in this burner configuration and highlight its effectiveness in achieving more uniform and intense combustion across the radial domain [19], [22]. Figures 9(a)–(d) show that LPG produces a broader, more uniform flame than CNG. On average, LPG's peak temperature is 14–15% higher, indicating greater combustion intensity and heat distribution. Its wider thermal spread supports more consistent heating, while CNG's narrower flame reflects its lower energy content.

4. Conclusion

This study addresses the need for cleaner, more efficient combustion in Thailand's ceramic industry, where LPG remains prevalent but CNG offers lower emissions and cost advantages. CFD simulations using ANSYS FLUENT 2021 R2 were conducted in two stages: cold-flow analysis for fuel–air mixing and hot-flow analysis for combustion behavior. A tetrahedral mesh with the RNG k–ε model was applied, and validation was performed at 4 psi using a K-type thermocouple.

Simulation and experimental results showed strong agreement, with deviations of 6.52% for velocity and 5.30% for temperature. LPG combustion achieved a higher peak temperature (1,375.61 K) and more uniform temperature distribution, while CNG exhibited higher flow velocities due to its lower density. These results confirm LPG's superior combustion characteristics and demonstrate the viability of CNG with appropriate burner modifications. Unlike high-pressure industrial systems, this study focuses on simple, cost-effective designs tailored for small ceramic kilns highlighting the novelty and practical value for resource-limited applications. Future work should explore hydrogen-enriched fuel blends, which offer potential for significant CO_2 emission reductions.

5. Acknowledgement

The author acknowledges financial support from the 2021 Government Personnel Development Scholarship (Ministry of Higher Education, Science, Research, and Innovation) and the Research, Innovation, and Academic Services Fund of Ubon Ratchathani University (Science and Technology Research Grant, Fiscal Year 2025).

References

[1] Energy Policy and Planning Office (EPPO). General Energy Information. [Online]. Available: https://www.eppo.go.th/epposite/templates/eppo_v15_mixed/eppo_oil_gen.php.

[2] W. Puttapoun, J. Moran, P. Aggarangsi, and A. Bunkham, "Powering shuttle kilns with compressed biomethane gas for the Thai ceramic industry," *Energy for Sustainable Development*, vol. 28, pp. 95–101, 2015.

[3] Y. Juntarawijit and C. Juntarawijit, "Cooking smoke exposure and respiratory symptoms among those responsible for household cooking: A study in Phitsanulok, Thailand," *Heliyon*, vol. 5, no. 6, pp. e01706, 2019.

[4] M. Wae-hayee, K. Yeranee, W. Suksuwan, and C. Nuntadusit, "Effect of burner-to-plate distance on heat transfer rate in a domestic stove using LPG," *Case Studies in Thermal Engineering*, vol. 28, pp. 101418, 2021

[5] N. Sritrakul and T. Hudakorn, "The economic value and satisfaction of substituting LPG in households by a biogas network: A case study of Bo Rae subdistrict in Chai Nat province Thailand," *Energy Reports*, vol. 6, pp. 565–571, 2020.

[6] K. Chuenwong, S. Chiarakorn, and B. Sajjakulnukit, "Specific energy consumption and carbon intensity of ceramic tablewares: Small Enterprises (SEs) in Thailand," *Journal of Cleaner Production*, vol. 147, pp. 395–405, 2017.

[7] M. S. A. Momin, M. Dutta, M. G. Kader, and S. M. Iftakher, "Study of LPG (Liquefied

Petroleum Gas) and CNG (Compressed Natural Gas) vehicles and its future aspects," in *Proceedings of the International Conference on Mechanical, Industrial and Energy Engineering (ICMIEE)*, Khulna, Bangladesh, 2016.

[8] N. I. Masuk, K. Mostakim, and S. D. Kanka, "Performance and emission characteristic analysis of a gasoline engine utilizing different types of alternative fuels: A comprehensive review," *Energy & Fuels*, vol. 35, no. 6, pp. 4644–4669, 2021.

[9] M. I. Khan, T. Yasmin, and A. Shakoor, "Technical overview of Compressed Natural Gas (CNG) as a transportation fuel," *Renewable and Sustainable Energy Reviews*, vol. 51, pp. 785–797, 2015.

[10] S. Jugjai and S. Sanitjai, "Parametric studies of thermal efficiency in a Proposed Porous Radiant Recirculated Burner (PRRB): A design concept for the future burner," *International Energy Journal*, vol. 18, no. 2, 1996.

[11] P. Aroonjarattham, "The parametric studied of high pressure gas burner affect thermal efficiency," *Engineering Journal*, vol. 20, no. 3, pp. 33–48, 2016.

[12] P. Boggavarapu, B. Ray, and R. V. Ravikrishna, "Thermal efficiency of LPG and PNG-fired burners: Experimental and numerical studies," *Fuel*, vol. 116, pp. 709–715, 2014.

[13] F. J. Cadavid, Y. Cadavid, A. A. Amell, A. E. Arrieta, and J. D. Echavarría, "Numerical and experimental methodology to measure the thermal efficiency of pots on electrical stoves," *Energy*, 2014.

[14] W. Li, M. Al-Khishali, and A. Kewlani, "Pollutant emission validation of a heavy-duty gas turbine burner: A 3D CFD study," *International Journal of Thermal Sciences*, vol. 191, pp. 107990, 2023.

[15] H. Liu, Z. Wang, and J. Zhao, "A review of employed flame stabilization techniques in development of micro/mesoscale combustion," *Renewable and Sustainable Energy Reviews*, vol. 158, pp. 112111, 2022.

[16] A. Salehi, M. Mahmoudi, and M. Saffar-Avval, "A Bayesian optimization framework for the control of combustion stability in a bluff-body combustor," *Energy*, vol. 236, pp. 121413, 2021.

[17] M. Shehata, I. A. Ibrahim, and H. M. Gad, "Combustion characteristics of natural gas/air flat premixed laminar flames in a developed matrix burner," *Scientific African*, vol. 20, pp. e01659, 2023.

[18] D. S. Yadav and B. Paul, "Conversion of the domestic LPG cook stove to use biogas fuel: An experimental approach," *Transactions of the Indian National Academy of Engineering*, vol. 7, pp. 1213–1222, 2022.

[19] F. J. Rojas, F. Jiménez, and J. Soto, "Design and experimental analysis of an improved burner with natural gas," *Energy Efficiency*, vol. 14, no. 43, pp. 1–13, 2021.

[20] Z. F. A. Gani, "Experimental investigation on lift-off, blowout, and drop-back in partially premixed LPG open flames in tubular burner," *Thermal Science*, vol. 26, no. 6A, pp. 4607–4615, 2022.

[21] R. A. Pradana, T. Prabowo, and A. Saifudin, "Pengaruh variasi lubang udara terhadap

efisiensi kompor gas LPG,” *J-Proteksion*, vol. 10, no. 2, pp. 23–28, 2020.

[22] H. O. Gómez, M. C. Calleja, L. A. Fernández, A. Kiedrzynska, and R. Lewtak, “Application of the CFD simulation to the evaluation of natural gas replacement by syngas in burners of the ceramic sector,” *Energy*, vol. 185, pp. 15–27, 2019.

[23] M. Wichangarm, A. Matthujak, T. Sriveerakul, S. Sucharitpwatskul, and S. Phongthanapanich, “Investigation on thermal efficiency of LPG cooking burner using computational fluid dynamics,” *Energy*, vol. 203, pp. 117849, 2020

[24] A. Matthujak, M. Wichangarm, T. Sriveerakul, S. Sucharitpwatskul, and S. Phongthanapanich, “Numerical and experimental study for a modified LPG cooking burner,” *Journal of King Saud University – Science*, vol. 35, pp. 102752, 2023.