ISOMAse International Society of Ocean, Mechanical and Aerospace Scientists and Engineers
Journal of Ocean, Mechanical and Aerospace
-Science and Engineering-
30 July 2026. Vol.70 No.2
DOI: 10.36842/jomase.v70i2.616

Comparative CFD Analysis of Air–Fuel Ratio Effects on the Performance of a Non-Premixed Combustion Burner

Taufik Kurniawan[a*], Muhammad Ikhlas[a] and Mohammad Zaini Ma'ruf[a]

[a)]*Teknologi Rekayasa Pemeliharaan Pesawat Udara, Politeknik Kirana, Indonesia*

*Corresponding author: Taufik.kurniawan@politeknikkirana.ac.id

Paper History

Received: 24-June-2026
Received in revised form: 22-July-2026
Accepted: 30-July-2026

ABSTRACT

Checker bricks in regenerative furnaces gradually accumulate deposits from glass raw materials during long-term operation, reducing heat transfer efficiency and obstructing combustion air and exhaust gas flow. These deposits increase furnace pressure, accelerate checker brick degradation, and shorten furnace service life. This study evaluates the performance of a non-premixed combustion burner for checker brick maintenance. Computational Fluid Dynamics (CFD) simulations were conducted using the non-premixed combustion model in ANSYS Fluent with different air-to-fuel ratios (AFR) and validated experimentally. The optimum performance was achieved at an AFR of 10.6:1, producing a maximum flame temperature of 944°C and a flame length of 1,683 mm, satisfying the required heating temperature and flame penetration for effective deposit removal. Experimental validation showed stable burner operation at air damper openings between 55% and 60%, successfully melting deposits on the checker brick surface. The proposed burner is suitable for regenerative furnace maintenance, while the validated CFD model provides a reliable approach for future burner optimization and heating coverage analysis of checker brick treatment.

KEYWORDS: *Non-Premixed, CFD, Fluent, Burner, Combustion.*

1. INTRODUCTION

Glass manufacturing is an energy intensive industry. Production of glass includes six processes namely: mixing, melting, refining, forming, annealing and finishing. Melting process takes place in the melting chamber at a temperature of 1400-1500°C and it is the most energy-intensive process [1]. The combustion process inside the furnace utilizes fuels such as natural gas or fuel oil mixed with combustion air to generate the thermal energy required for the glass melting process.

During combustion, in addition to generating thermal energy, the furnace produces flue gas containing various components, including nitrogen (N_2), carbon dioxide (CO_2), water vapor (H_2O), residual oxygen (O_2), and small amounts of pollutants such as carbon monoxide (CO), nitrogen oxides (NO_x), sulfur oxides (SO_x), and particulate matter, depending on the type of fuel and combustion conditions. Since the flue gas still contains a significant amount of thermal energy, it can be utilized through a heat recovery system to improve the overall energy efficiency of the melting process.

Flue gas management is an important aspect of glass furnace operation because it directly affects combustion efficiency, fuel consumption, equipment lifetime, and compliance with environmental emission standards. One of the key parameters influencing combustion performance is the regulation of combustion air, including the adjustment of the fresh air damper opening. Changes in the fresh air can damper opening influence the amount of excess air, oxygen concentration in the flue gas, flue gas temperature, and heat transfer efficiency within the furnace. Therefore, analyzing the relationship between the fresh air damper opening, flue gas temperature, and other operating parameters is essential for evaluating the performance of the combustion system. The results of such analyses can serve as the basis for optimizing furnace operation to achieve an appropriate balance between energy efficiency, glass product quality, and emission control.

In regenerative glass furnaces, checker bricks function as thermal energy storage media. As the hot flue gas passes through the regenerator, thermal energy is absorbed by the refractory checker bricks. When the flow direction is reversed, the stored heat is released to preheat the combustion air, thereby increasing the temperature of the air entering the burner. If the flow reversal occurs too frequently, the checker bricks are unable to reach their optimum heat storage capacity, resulting in insufficient preheating of the combustion air. Conversely, a longer regenerator reversal cycle allows more effective heat storage and heat release, thereby reducing fuel consumption. Previous studies have demonstrated that optimizing the regenerator reversal time is a relatively simple operational strategy because it does not require major modifications to the furnace design while providing significant improvements in energy efficiency [2].

The glass melting process also generates flue gases containing volatile compounds such as sodium hydroxide (NaOH), sulfur dioxide (SO_2), and fine particulate matter. As the flue gas cools within the regenerator, these volatile compounds react and condense, forming deposits predominantly composed of sodium sulfate (Na_2SO_4) [3]. The accumulation of deposits on the checker bricks reduces the effective heat transfer surface area and increases gas flow resistance. Consequently, the heat recovery efficiency decreases, the combustion air temperature is reduced, and additional fuel is required to maintain the desired glass melting temperature. Furthermore, partial blockage of the gas passages increases the pressure drop across the regenerator, disrupts gas flow distribution, and accelerates the deterioration of overall furnace performance.

In magnesia–chromite refractories, liquid corrosive species react with the refractory constituents to form low-melting silicate and vanadate phases within the intergranular regions of MgO crystals. During thermal cycling, these phases experience volumetric expansion and contraction, generating mechanical stresses that lead to crack initiation and spalling. Consequently, the corrosion process is accelerated, resulting in progressive degradation of the refractory material [4].

The degradation of regenerator performance caused by deposit accumulation also leads to an increase in the temperature of the flue gas discharged through the chimney. This indicates that a considerable portion of the thermal energy that should have been stored by the checker bricks cannot be recovered, resulting in lower overall energy efficiency and potentially higher exhaust gas emissions. Therefore, monitoring the condition of checker bricks and evaluating the influence of deposits on heat transfer performance are essential aspects of regenerative glass furnace operation. The findings can be used as the basis for determining appropriate cleaning strategies, optimizing combustion conditions, and regulating the combustion air supply to maintain the thermal efficiency of the regenerator and extend the operational lifetime of the furnace.
Several methods are available for cleaning checker bricks, including mechanical cleaning, compressed-air cleaning (air blowing), water jet cleaning, and chemical cleaning. The selection of the cleaning method depends on the type, composition, and hardness of the deposits formed during furnace operation.

One of the most widely adopted industrial methods is burner cleaning. This technique employs a high-capacity burner flame directed toward the checker brick region to generate sufficiently high temperatures that soften the deposits, induce thermal cracking, or detach them from the checker brick surface. Once detached, the deposits can be removed using airflow, vacuum systems, or mechanical cleaning. Compared with complete regenerator dismantling, burner cleaning requires a significantly shorter shutdown period, thereby minimizing production losses.

The effectiveness of burner cleaning depends on several operating parameters, including flame temperature, heat distribution, heating duration, fuel type, and the characteristics of the deposits. These parameters must be carefully controlled to ensure effective deposit removal without damaging the checker bricks or the surrounding refractory materials. In addition, operational safety and temperature control are critical considerations because the cleaning process is performed under extremely high-temperature conditions.

Selecting an appropriate cleaning method is essential for restoring the heat transfer capability of the regenerator, reducing gas flow resistance, and extending the service life of the checker bricks. Furthermore, the effectiveness of the cleaning process can be evaluated by monitoring key operating parameters such as flue gas temperature, combustion air temperature, regenerator pressure drop, and fuel consumption. Consequently, a well-planned cleaning strategy not only enhances the operational reliability of the furnace but also improves energy efficiency and reduces operating costs in the glass manufacturing industry.

2. METHOD

Computational Fluid Dynamics or CFD is a branch of fluid mechanics that involves numerical analytic process and algorithms to simulate fluid flow around an object. These state-of-the-art methods had a vast means of application, due to the availability of numerical codes and the capability of the computational performance, with the possibility to generalize certain algorithms to more intricate physical quandaries [5]. To evaluate the effectiveness of burner performance, a comparative study is required to quantitatively assess the performance of different burner designs or operating conditions. One of the most effective approaches is the use of Computational Fluid Dynamics (CFD) based numerical analysis implemented in ANSYS Fluent. This simulation technique enables detailed investigation of fluid flow behavior, fuel–air mixing, temperature distribution, heat transfer, and the formation of combustion products without the need for costly and time-consuming experimental testing.

The comparative study was conducted by evaluating several burner configurations or operating parameter variations under identical boundary conditions. This approach ensures that any differences observed in the simulation results are solely attributed to the burner design rather than external operating factors. The performance assessment included the analysis of temperature distribution within the furnace, gas flow patterns, heat transfer uniformity, combustion efficiency, and the potential formation of localized high-temperature regions (hot spots) as well as zones of incomplete combustion.

The simulation results obtained from ANSYS Fluent were subsequently analyzed to identify the burner configuration that provided the most uniform temperature distribution, higher heat transfer efficiency, and more stable flow characteristics. Consequently, this numerical approach offers a reliable technical basis for evaluating burner performance and supports informed decision-making in burner design optimization.

Granados et al. [6] used CFD analysis to investigate the combustion phenomenon in a rotary cement, namely recirculation. The remaining combustion gases were used for the combustion process and the combustion air had been mixed with the exhaust gases. The study aimed to determine the combustion temperature, radiation and convection heat transfer, and the flame length. It revealed that the highest temperature results occurred at a concentration of 85%. Similarly, the convection and radiation heat transfer are also reduced. According to this study, the most optimal concentration was at 85% [6].

ISOMAse
International Society of Ocean, Mechanical and Aerospace Scientists and Engineers
Journal of Ocean, Mechanical and Aerospace
-Science and Engineering-
30 July 2026. Vol.70 No.2
DOI: 10.36842/jomase.v70i2.616

2.1 Non-Premixed Combustion

The combustion systems used in the energy sector can generally be divided into three categories: (1) gas turbines for electricity production and industrial purposes, (2) internal combustion engines and aeronautical engines for transport, and (3) industrial burners and furnaces [7]. The mixture fraction concept plays a vital role in No premixed combustion model. Considering certain assumptions, the instantaneous thermo-chemical state of the fluid is related to conserve scalar quantity known as mixture fraction f. The elemental mass fractions remain constant throughout all the reactions. In Non-premixed combustion model, flame is considered as co-flow of fuel and oxidizer. In such mixture, Mixture fraction f for element n at a specific point can be given as:

$$f_n = \frac{Z_n - Z_{n,ox}}{Z_{n.fuel} - Z_{n,ox}} \quad (1)$$

Abbreviation z is the mass fraction of species n. During use of no premixed combustion model, Fluent has to solve two equations. These are mean mixture fraction equation, and mixture fraction variance equation [8].

The test rig was used for the experimental study of the isothermal swirling flow field by Boushaki et al. [9] and for the impact of oxygen enrichment on non-premixed swirling flames by Merlo et al. [10]. The burner is coaxial; it is fed by air in the annular part and by methane in the central tube. From other research when the excess air rate is in the range of 1.15 to 1.4 and the heat load is 24 kW, the CO emissions of the SS burner are higher than of the MF burner. The CO emission of SS burner increases first and then decreases, while that of the MF burner decreases all the time. When other conditions are the same, the CO emission of the two burners increases with the increase of heat load [11].

The CO_2 pollutant mass fraction indicates a reduction in CO_2 emissions with increasing co-firing of empty fruit bunches (EFB). A significant decrease in the CO_2 mass fraction was observed when comparing coal combustion with co-firing containing 10% EFB, followed by a slight reduction as the EFB proportion increased from 20% to 30%. These results indicate that increasing the percentage of EFB in the fuel mixture leads to a reduction in the CO_2 mass fraction [12]. The CO_2 concentration increases as the air–fuel ratio (AFR) approaches the stoichiometric condition, reaching its maximum value at the optimum air–fuel ratio. However, the CO_2 concentration decreases when the AFR is either too low (fuel-rich mixture) or too high (excess-air condition).

These findings are consistent with previous studies on natural gas combustion and premixed burners, including those reported by Shi and Wang (2023) and Liu et al. (2020), which demonstrated that the highest CO_2 concentration is achieved under the optimum excess-air condition. In contrast, excessively low AFR values promote carbon monoxide (CO) formation and reduce the conversion efficiency of carbon into CO_2 [13].

2.2 Geometry for CFD Analysis

To achieve optimal performance from a non-premixed combustion burner, a comprehensive analysis is required to ensure that the generated heat meets the operational requirements. In this study, the initial burner design selected for analysis is presented as the baseline configuration for numerical evaluation. The design and optimization of the checker burner were carried out based on the following engineering constraints and operational requirements:

1. The melting point of the salt cake deposit is 920°C.
2. The maximum allowable temperature of the rider arch is 1000°C.
3. The minimum required flame length is 1,674 mm.

These design criteria were adopted as the primary performance indicators in evaluating the burner configuration. The objective was to produce a stable flame with sufficient length to ensure effective heat transfer while preventing excessive temperatures that could accelerate refractory degradation or cause structural damage to the rider arch. Furthermore, maintaining the wall temperature below the specified limit is essential to minimize the risk of refractory corrosion associated with molten salt deposits.

As shown in the Figure 1, the cross-sectional areas of the gas and air supply inlets used for the combustion process were determined. Based on these data, a series of simulations with different air–fuel ratios (AFRs) was performed to identify the optimum combustion conditions. The objective of the simulation was to determine the appropriate fuel and air consumption required to achieve the target furnace temperature and the desired flame length. The simulation setup parameters for the non-premixed combustion burner can be seen in Figure 2.

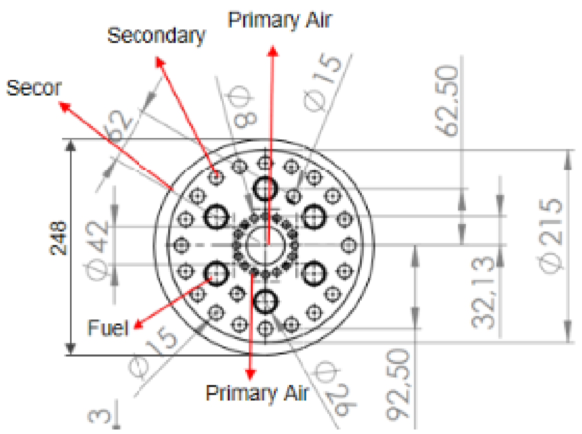

Figure 1: Geometry of the inlet head of the non-premixed combustion burner

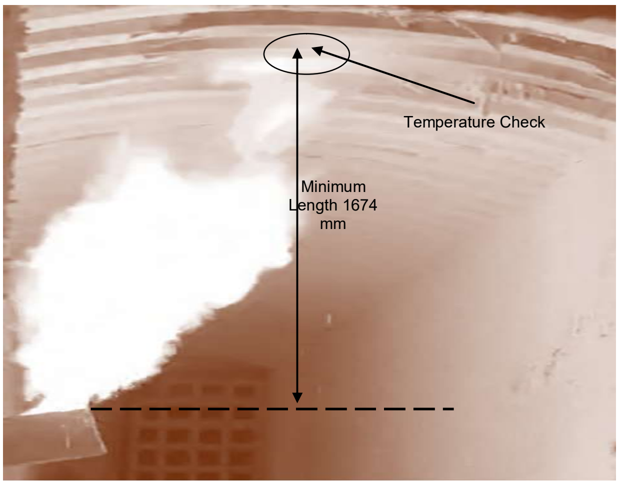

Figure 2: Simulation setup parameters for the non-premixed combustion burner

JOMAse | Received: 24-June-2026 | Accepted: 30-July-2026 | [(70) 2: 118-126]
Published by International Society of Ocean, Mechanical and Aerospace Scientists and Engineers, www.isomase.org., ISSN: 2354-7065 & e-ISSN: 2527-6085

The parameters were used for the comparative analysis in the ANSYS Fluent simulations depicted in Table 1.

Table 1: Comparative air–fuel ratio data for ansys fluent analysis

Parameter	Fuel	Primary Air	Secondary Air
Area (m²)	2.94 x 10⁻³	2.4 x 10⁻³	1.35 x 10⁻²
V AFR 21 : 1 (m/s)	5	18	20
V AFR 15 : 1 (m/s)	7	18	20
V AFR 10.6 : 1 (m/s)	10	18	20

2.3 Experimental and Setup Measurement

Experimental tests were carried out to investigate the combustion characteristics of a non-premixed combustion burner using natural gas as the fuel. In this configuration, the fuel and combustion air are introduced through separate channels and are mixed only inside the combustion chamber. This arrangement allows the mixing process and combustion to occur simultaneously within the flame region, which is a characteristic feature of non-premixed combustion systems.

The combustion air was supplied by a blower with a capacity was 3,600 m³/h, providing a continuous airflow to meet the operational requirements of the burner. The airflow rate was adjusted to achieve the desired air–fuel ratio (AFR), ensuring stable and efficient combustion throughout the experiments. Natural gas was selected as the fuel because of its high heating value and low particulate emissions, making it suitable for high-temperature industrial combustion applications.

The fuel supply system was equipped with a gas train incorporating a manual ignition system and a safety shut-off valve. The manual ignition system was used to initiate flame formation during burner start-up, while the safety shut-off valve automatically interrupted the gas supply under abnormal operating conditions, such as flame failure or combustion system malfunction. This safety arrangement was implemented to enhance operational safety during the experimental tests while ensuring stable and reliable burner operation. Experimental and setup measurement of items and the specification presented in Table 2. Whilst, in Table 3 shows the parameter setting for input natural gas, natural gas setting, input air supply and air supply setting.

The 3D program is required to get the three-dimensional images of the propeller, so that it can be used for meshing. The propeller model needs to mesh to break the measuring object into an infinite element, and make it easy for a computer to calculate the finite form elements. The finite elements are necessary to obtain the results of numerical simulation in the form of Computer Fluid Dynamic (CFD) [14].

Figure 3: Non-premixed combustion burner head assembly

Table 2: Experimental and setup measurement

Item	Specification
Burner	Non premixed Combustion
Air Supply	Blower cap 3600 m³/hr
Gas Material	Natural Gas
Gas train setting	Manual Ignition with safety valve

Table 3: Parameter setting

Parameter	Specification
Input Natural Gas	300 Nm³/hr
Natural Gas Setting	0.9 bar
Input Air Supply	3600 m³/hr
Air Supply setting	Manual opening damper 50%-75%
Temperature Check	Manual check every 20 minutes

During the experiments, the burner was operated under steady-state conditions to ensure that both the temperature field and the combustion process had reached thermal and chemical equilibrium prior to data collection. Different operating conditions were established by varying the combustion air and fuel flow rates to achieve a range of air–fuel ratios (AFRs). The resulting experimental data were then analyzed to assess the burner performance in terms of temperature distribution and flame characteristics.

3. RESULTS AND DISCUSSION

3.1 Fluent Analysis

The air–fuel ratio (AFR) was selected as the primary operating parameter in the ANSYS Fluent simulations because it has a significant influence on combustion efficiency, flame stability, temperature distribution, and pollutant formation. Three AFR conditions, namely 21:1, 15:1, and 10.6:1, were investigated to evaluate their effects on the combustion characteristics and

JOMAse | Received: 24-June-2026 | Accepted: 30-July-2026 | [(70) 2: 118-126]
Published by International Society of Ocean, Mechanical and Aerospace Scientists and Engineers, www.isomase.org., ISSN: 2354-7065 & e-ISSN: 2527-6085

ISOMAse International Society of Ocean, Mechanical and Aerospace Scientists and Engineers
Journal of Ocean, Mechanical and Aerospace
-Science and Engineering-
30 July 2026. Vol.70 No.2
DOI: 10.36842/jomase.v70i2.616

overall burner performance. These operating conditions represent different combustion regimes, ranging from excess-air conditions to a near-optimum air–fuel mixture. For each case, the numerical simulation was performed using the non-premixed combustion model to predict flame temperature, flame propagation, flow behavior, and species distribution. The simulation results were subsequently compared to determine the optimum AFR capable of producing the required thermal performance for checker brick maintenance. In addition, the numerical predictions were validated through experimental testing to assess the reliability of the CFD model and to ensure that the selected operating condition could provide stable combustion and sufficient heating for effective deposit removal under practical operating conditions.

3.1.1 Air–Fuel Ratio (AFR) 21:1

The air–fuel ratio (AFR) was determined from the inlet velocities of natural gas and combustion air. The inlet velocity of natural gas was maintained at 5 m/s, whereas the primary and secondary air velocities were specified as 18 m/s and 20 m/s, respectively. Using these inlet conditions, the corresponding AFR values were calculated and subsequently employed as the operating parameters for the CFD simulations.

a. Fuel flow Capacity 52.94 m³/h
b. Air Flow Capacity 1127.52 m³/h
c. Air Fuel Ratio 21 : 1

Based on these conditions, the simulation results are presented in the Figure 4.

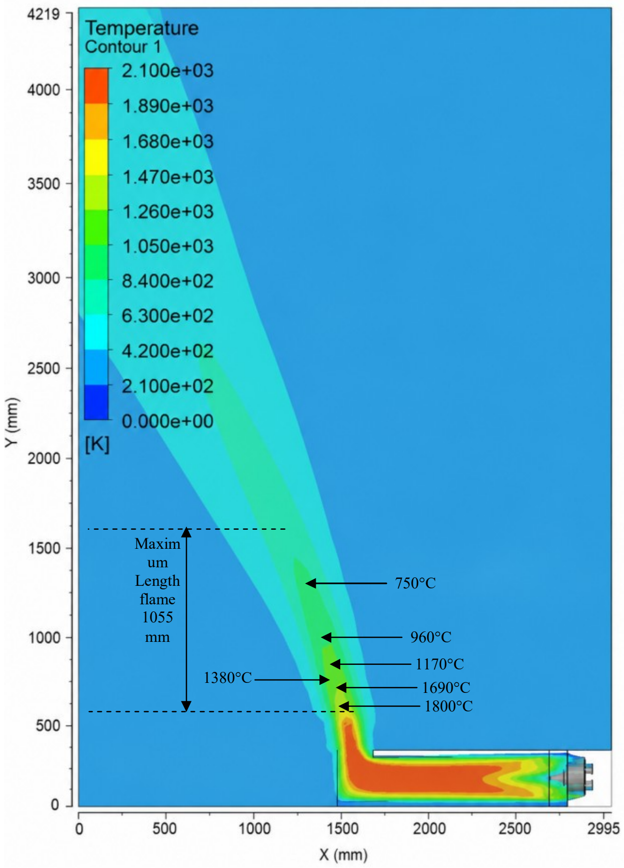

Figure 4: Temperature distribution of the non-premixed combustion burner at an air–fuel ratio (AFR) of 21:1

Based on the simulation results, it can be concluded that an air–fuel ratio (AFR) of 21:1 is not suitable for the checker burner application. The maximum temperature achieved was only 750°C, whereas a minimum temperature of 920°C is required to melt the salt cake deposits. Furthermore, the flame length reached only 1,055 mm, which is significantly shorter than the minimum required flame length of 1,674 mm.Flame length is one of the most important parameters in industrial burner design because it directly influences heat distribution, combustion efficiency, and heat transfer. In oxy-fuel non-premixed combustion systems, flame characteristics differ from those of air-fired combustion due to the higher oxygen concentration, which produces higher flame temperatures and faster chemical reaction rates [15].

However, the combustion performance in terms of carbon dioxide (CO_2) formation was satisfactory, with a maximum CO_2 mass fraction of 0.1394. This result indicates that the combustion process was relatively complete under this operating condition, despite the burner failing to satisfy the required thermal performance in terms of flame temperature and flame length.

3.1.2 Air–Fuel Ratio (AFR) 15: 1

The air–fuel ratio (AFR) was determined from the inlet velocities of natural gas and combustion air. The inlet velocity of natural gas was maintained at 7 m/s, whereas the primary and secondary air velocities were specified as 18 m/s and 20 m/s, respectively. Using these inlet conditions, the corresponding AFR values were calculated and subsequently employed as the operating parameters for the CFD simulations.

a. Fuel flow Capacity 74.08 m³/h
b. Air Flow Capacity 1127.52 m³/h
c. Air Fuel Ratio 15 : 1

Based on these conditions, the simulation results are presented in the Figure 5.

Based on the simulation results, it can be concluded that an air–fuel ratio (AFR) of 15:1 is not suitable for the checker burner application. The maximum temperature achieved was only 755°C, whereas a minimum temperature of 920°C is required to melt the salt cake deposits. In addition, the flame length reached only 1,447 mm, which is below the minimum required flame length of 1,674 mm.

However, the combustion process still exhibited satisfactory CO_2 formation, although the maximum CO_2 mass fraction decreased to 0.1214 compared with the AFR of 21:1. This reduction in CO_2 mass fraction indicates a slight decrease in combustion completeness under the simulated operating conditions. Nevertheless, the burner performance remained insufficient to satisfy the thermal requirements of the checker burner, as both the flame temperature and flame length failed to meet the specified design criteria.

3.1.3 Air–Fuel Ratio (AFR) 10.6:1

The air–fuel ratio (AFR) was determined from the inlet velocities of natural gas and combustion air. The inlet velocity of natural gas was maintained at 10 m/s, whereas the primary and secondary air velocities were specified as 18 m/s and 20 m/s, respectively. Using these inlet conditions, the corresponding AFR values were calculated and subsequently employed as the operating parameters for the CFD simulations.

a. Fuel flow Capacity 105.84 m³/h
b. Air Flow Capacity 1127.52 m³/h

Journal of Ocean, Mechanical and Aerospace
-Science and Engineering-
30 July 2026. Vol.70 No.2
DOI:
10.36842/jomase.v70i2.616

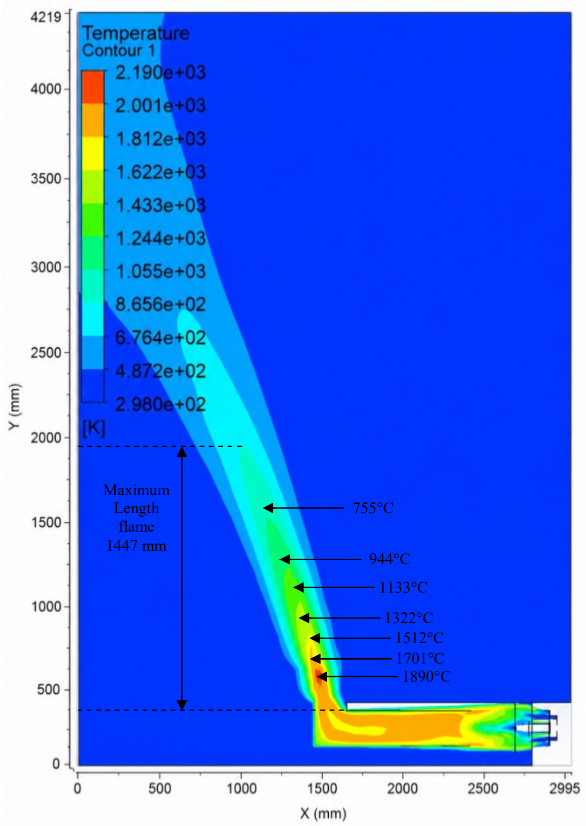

Figure 5: Temperature distribution of the non-premixed combustion burner at an air–fuel ratio (AFR) of 15:1

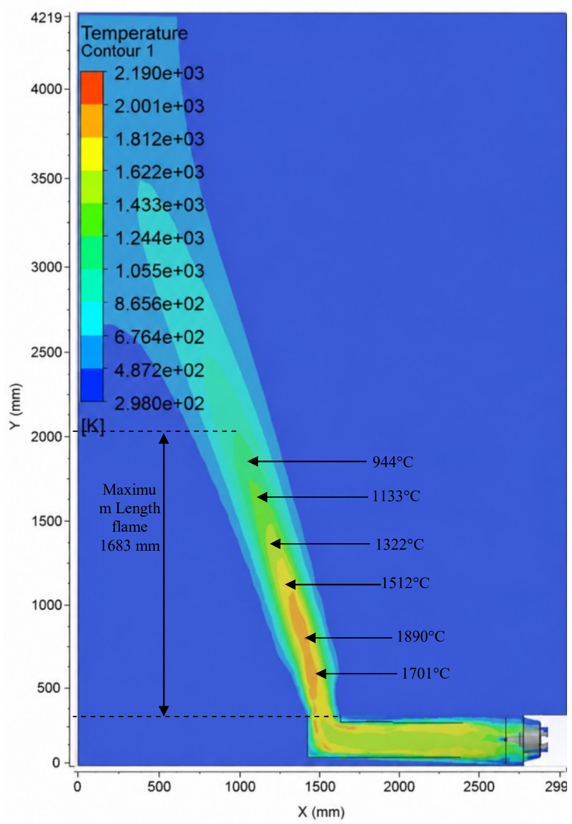

Figure 6: Temperature distribution of the non-premixed combustion burner at an air–fuel ratio (AFR) of 10.6:1

c. Air Fuel Ratio 10.6 : 1

Based on these conditions, the simulation results are presented in the Figure 6.

Based on the simulation results, it can be concluded that an air–fuel ratio (AFR) of 10.6:1 is suitable for the checker burner application. The maximum temperature achieved was 944°C, exceeding the minimum required temperature of 920°C needed to melt the salt cake deposits. In addition, the flame length reached 1,683 mm, which satisfies the minimum design requirement of 1,674 mm. The combustion performance in terms of carbon dioxide (CO_2) formation was also satisfactory, with a maximum CO_2 mass fraction of 0.1256, representing a slight increase compared with the AFR of 15:1. Very low concentrations of CO and C_3H_8 were observed at the combustor outlet, indicating a nearly complete combustion process [16].

This result indicates that the combustion process remained efficient while simultaneously meeting the required thermal performance in terms of both flame temperature and flame length. Therefore, among the AFR conditions investigated, an AFR of 10.6:1 provided the most favorable operating condition for the checker burner. The variation of the air–fuel flow has a significant influence on the combustion characteristics and overall burner performance. Changes in the air flow supplied to the burner alter the air–fuel mixing process, resulting in differences in flame stability, temperature distribution, flame propagation, and combustion efficiency. In the present study, reducing the air–fuel ratio (AFR) toward the optimum value improved the combustion performance by producing a higher flame temperature and a longer flame, indicating more effective heat transfer to the checker bricks.

These findings are consistent with previous CFD studies conducted using OpenFOAM, which demonstrated that variations in the confluence angle (90°, 70°, 50°, 30°, and 10°) significantly affected flow velocity and the formation of secondary flow regions, including contraction, separation, stagnation, and flow deflection zones [17]. The comparative results of air–fuel ratio (AFR) simulations using ansys fluent presents in Table 4.

Table 4: Comparative results of air–fuel ratio (AFR) simulations using ansys fluent

Parameter	Flame Height (mm)	Max Temp (°C) at max height
V AFR 21 : 1 (m/s)	1055	750
V AFR 15 : 1 (m/s)	1447	755
V AFR 10.6 : 1 (m/s)	1683	944

3.2 Experimental Result

Figure 7 shows the experimental results of the checker brick temperature during burner operation together with the opening of the air damper. The dashed horizontal line represents the minimum target temperature of 920°C, which were the minimum temperatures required to melt the salt cake deposits on the checker bricks.

JOMAse | Received: 24-June-2026 | Accepted: 30-July-2026 | [(70) 2: 118-126]
Published by International Society of Ocean, Mechanical and Aerospace Scientists and Engineers, www.isomase.org., ISSN: 2354-7065 & e-ISSN: 2527-6085

ISOMAse International Society of Ocean, Mechanical and Aerospace Scientists and Engineers
Journal of Ocean, Mechanical and Aerospace
-Science and Engineering-
30 July 2026. Vol.70 No.2
DOI: 10.36842/jomase.v70i2.616

At the beginning of the experiment (07:15–15:00), the checker brick temperature exhibited noticeable fluctuations, ranging from approximately 840°C to 960°C. During this period, several measurements fell below the minimum target temperature of 920°C, indicating that the combustion process had not yet reached stable operating conditions. These temperature fluctuations are attributed to variations in the combustion process and the air–fuel mixing characteristics during burner operation.

After approximately 15:30, the combustion system became significantly more stable. The checker brick temperature was consistently maintained within the range of 920–980°C, satisfying the minimum operational requirement for melting the deposits. During this period, the air damper opening was adjusted and maintained at approximately 55–75%, providing sufficient combustion air to sustain stable flame conditions and uniform heat transfer.

The highlighted region in Figure 7 indicates the operational damper target, where the burner achieved its optimum operating condition. Within this operating range, the checker brick temperature remained above the required minimum temperature while exhibiting relatively small fluctuations. This result demonstrates that appropriate control of the air damper plays a crucial role in maintaining stable combustion and achieving the desired thermal performance.

Overall, the experimental results confirm that operating the burner with an air damper opening of approximately 55–60% is capable of maintaining the checker brick temperature above 920°C, thereby satisfying the thermal requirement for melting salt cake deposits. The stable temperature distribution observed under these conditions also indicates effective combustion performance and supports the suitability of the selected burner operating parameters for checker burner applications.

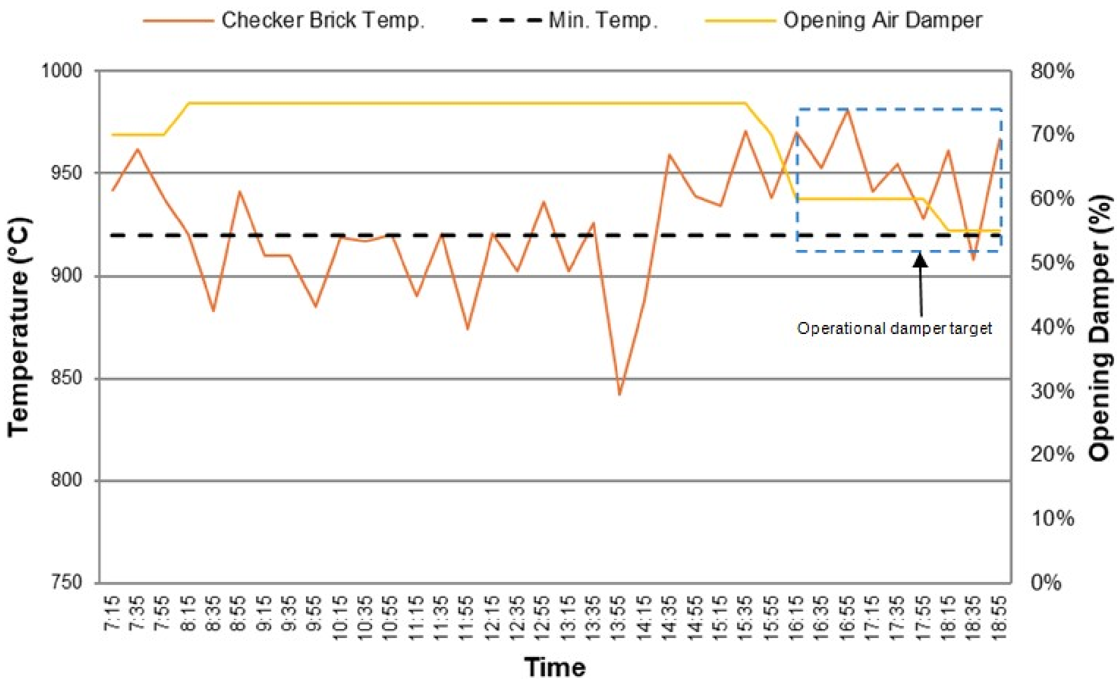

Figure 7: Experimental results of the non-premixed combustion burner.

3.3 Comparison of Simulation and Experimental Results

Figure 8 compares the flame shape obtained from the ANSYS Fluent simulation with that observed in the experimental investigation of the non-premixed combustion burner. Overall, the simulation results show good agreement with the experimental observations in terms of flame shape, temperature distribution, and flame propagation direction. This agreement indicates that the numerical model is capable of accurately representing the combustion phenomena occurring under actual operating conditions.

In the CFD simulation, the flame originates at the burner exit and propagates upward with a slight inclination following the gas flow direction. The highest temperature region is located within the flame core, as indicated by the red and yellow temperature contours, whereas the lowest temperature region is observed near the burner lip and in the recirculation zone, represented by the blue and green contours. The recirculation zone serves as a flame anchoring region, promoting stable combustion by continuously supplying hot combustion products to the flame base.

The experimental results exhibit similar combustion characteristics. The flame emerges from the burner nozzle and extends upward with a flame shape that closely resembles the CFD prediction. The flame core appears bright white to yellow, indicating a high-temperature combustion zone, while the lower region near the burner outlet exhibits a blue flame, corresponding to a relatively lower-temperature region. The similarity in the locations of the highest and lowest temperature zones between the simulation and the experiment demonstrates that the temperature distribution predicted by ANSYS Fluent is in good agreement with the actual combustion behavior.

Furthermore, the flame length and propagation direction predicted by the CFD simulation are consistent with those observed experimentally. Both results indicate that the flame remains stable throughout the combustion process without evidence of flame lift-off or flame extinction. This behavior suggests that the selected operating conditions provide effective mixing between natural gas and combustion air, resulting in stable and efficient combustion.

Overall, the comparison confirms that the ANSYS Fluent simulation accurately predicts the combustion characteristics of the non-premixed burner. The close agreement in flame shape, temperature distribution, flame length, and flame propagation between the numerical and experimental results validates the reliability of the CFD model. Therefore, the developed numerical model can be considered a reliable tool for analyzing combustion behavior and supporting the optimization of burner design and operating parameters in future studies.

The study concluded that burner diameter, flame temperature, fuel dilution level, and oxidizer composition are the primary factors influencing flame length in non-premixed combustion. Furthermore, the CFD simulations were shown to accurately predict flame length, with results that were in good agreement with the experimental measurements [18]. The study concluded that the flow characteristics within the furnace have a significant influence on the performance of non-premixed combustion. The distributions of flow velocity, pressure, and the thermal boundary layer affect the flame temperature, flame length, and the formation of combustion emissions [19].

A study on the recirculation zone and ignition location of a bluff-body burner under non-premixed MILD combustion conditions was conducted using Computational Fluid Dynamics (CFD). The results showed that the recirculation zone generated by the bluff body enhanced the turbulence intensity of the fuel and air flows. This increased turbulence promoted more homogeneous and effective fuel–air mixing compared with non-turbulent flow conditions, thereby improving combustion stability and overall combustion efficiency [20].

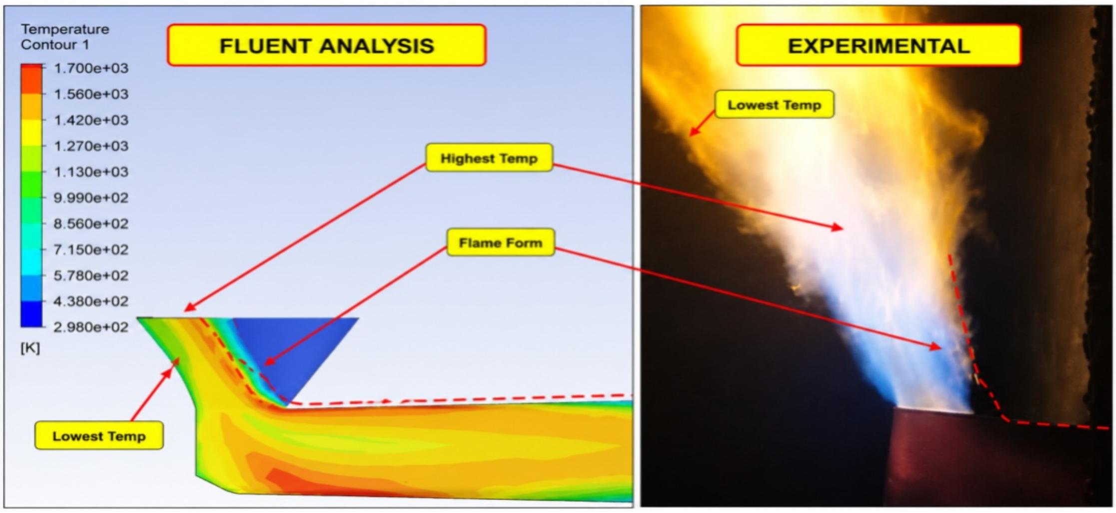

Figure 8: Comparison of flame characteristics between ansys fluent simulation and experimental results

4. CONCLUSION

This study demonstrated that the performance of the non-premixed combustion burner is strongly influenced by the air–fuel ratio (AFR). Among the investigated operating conditions, an AFR of 10.6:1 provided the optimum combustion performance, achieving a maximum flame temperature of 944°C, a flame length of 1,683 mm, and a CO_2 mass fraction of 0.1256. These results satisfied the thermal and geometric requirements for melting salt cake deposits on checker bricks and indicate efficient combustion performance. The observed relationship between AFR and combustion characteristics is consistent with previous studies, which reported that optimum excess-air conditions improve combustion efficiency, maximize CO_2 formation, and minimize incomplete combustion products [11–13]. Experimental validation confirmed the CFD predictions by maintaining checker brick temperatures within 920–980°C under stable burner operation with air damper openings of 55–60%, demonstrating the burner's capability for effective checker brick maintenance. The close agreement between numerical and experimental results verifies the reliability of the ANSYS Fluent CFD model as a predictive tool for burner performance evaluation and operating parameter optimization. Future work should focus on optimizing burner geometry and evaluating the heating coverage area to further improve the efficiency of checker brick maintenance in industrial regenerative furnaces.

REFERENCES

[1] Worrell, E. (2008). Energy Efficiency Improvement and Cost Saving Opportunities for the Glass Industry. An ENERGY STAR Guide for Energy and Plant Managers. *Lawrence Berkeley National Laboratory*. doi:10.2172/927883.

[2] Arican, T. F. & Hortaçsu, Ö. (1992). *Energy conservation in glass manufacture*. Energy, 17(6), 617–624. doi: 10.1016/0360-5442(92)90097-J.

[3] Kumpa, J., Ineure, P., Chakartnarodom, P., Cherdhirunkorn, B., Laitila, E. A. & Kongkajun, N.

JOMAse | Received: 24-June-2026 | Accepted: 30-July-2026 | [(70) 2: 118-126]
Published by International Society of Ocean, Mechanical and Aerospace Scientists and Engineers, www.isomase.org., ISSN: 2354-7065 & e-ISSN: 2527-6085

(2022). Recycling of exhausted dust from regenerator of glass furnace in glass batch melting. *Journal of Metals, Materials and Minerals,32*(2), 50–55. doi:10.55713/jmmm.v32i2.1263.

[4] An, Y., Liu, H., Wang, Z., Ma, Y. & Wang, X. (2023). Research on Corrosion Resistance of Checker Bricks in Regenerators of Glass Furnaces. *China's Refractories*, *32*(1), 20.

[5] Aksenov, A. A., Pechenyuk, A. V. & Vučinić, D. (2015). Ship hull form design and optimization based on CFD. In *Proceedings of the Towards Green Marine Technology and Transport Conference*. London, United Kingdom.

[6] Granados, D. A., Chejne, F., Mejía, J. M., Gómez, C. A., Berrío, A. & Jurado, W. J. (2014). Effect of flue gas recirculation during oxy-fuel combustion in a rotary cement kiln. *Energy, 64*, 615–625. https://doi.org/10.1016/j.energy.2013.11.032.

[7] Mansouri, Z. & Boushaki, T. (2018). Experimental and numerical investigation of turbulent isothermal and reacting flows in a non-premixed swirl burner. *International Journal of Heat and Fluid Flow, 72*, 200–213. Doi:10.1016/j.ijheatfluidflow.2018.06.007.

[8] Dabade, T. M., Li, X., Chen, D., Lou, H. H., Martin, C., Li, K. & Singh, K. D. (2012). *CFD simulation of confined non-premixed flames*. In *Proceedings of the ASME 2012 Summer Heat Transfer Conference (HT2012)*. ASME. doi:10.1115/HT2012-58497.

[9] Boushaki, T., Merlo, N., Chauveau, C. & Gökalp, I. (2017). Study of pollutant emissions and dynamics of non-premixed turbulent oxygen enriched flames from a swirl burner. *Proceedings of the Combustion Institute*, *36*(3), 3959-3968.

[10] Merlo, N., Boushaki, T., Chauveau, C., de Persis, S., Pillier, L, Sarh, B., Gökalp, I., (2013).Experimental study of oxygen enrichment effects on turbulent non-premixed swirling flames. *Energy Fuel* 27, 6191–6197.

[11] Liu, F., Zheng, L. & Zhang, R. (2020). *Emissions and thermal efficiency for premixed burners in a condensing gas boiler*. *Energy*, 202**,** 117449. https://doi.org/10.1016/j.energy.2020.117449.

[12] Nuryadi, A. P., Helios, M. P., Fathoni, A. M., Chairunnisa, C., Fitrianto, F., Pujowidodo, H., Sumarah, K. P., Komara, R. J., Kuswa, F. M. &Soewono, R. T. (2023). CFD simulation of CO_2 reduction in co-firing of coal and palm oil empty fruit bunches using a non-premixed combustion model. *Jurnal Teknologi Lingkungan, 24*(2), 283–291. https://doi.org/10.55981/jtl.2023.725.

[13] Shi, H.-W. & Wang, H.-P. (2023). Research on full premixed combustion and emission characteristics of non-electric gas boiler. *Energies, 16*(21), 7409. https://doi.org/10.3390/en16217409.

[14] Asrowibowo, N., Indiaryanto, M. & Rina (2016). Optimization design and hydrodynamic test on propeller mini submarine. *Journal of Ocean, Mechanical and Aerospace – Science and Engineering (JOMAse), 27(1), 8–12.* https://doi.org/10.36842/jomase.v27i1.425

[15] Sautet, J. C., Salentey, L., Ditaranto, M. & Samaniego, J. M. (2001). Length of natural gas-oxygen non-premixed flames. *Combustion Science and Technology, 166(1), 131–150.* https://doi.org/10.1080/00102200108907823

[16] Kumar, A. & Bharti, R. P. (2026). Performance evaluation of rans-based turbulence models in predicting turbulent non-premixed swirling combustion within a realistic can combustor. *arXiv* .https://doi.org/10.48550/arXiv.2605.19994

[17] Pratiwi, T. M., Putra, Y. S., Sampurno, J., Muhardi & Sutanto, Y. (2024). Study of the secondary flow formed on two-channel junction using computational fluid dynamics. *Journal of Ocean, Mechanical and Aerospace - Science and Engineering, 68(3), 106–117.* https://doi.org/10.36842/jomase.v68i3.372.

[18] Mishra, A., Sharma, P., Gebreyesus, B. Y., Kumar, M., & Ray, A. (2022). A Comprehensive Study on Flame Length under Oxygen Enhanced Laminar Non-premixed Combustion. *Combustion Science and Technology, 196(2), 223–244.* https://doi.org/10.1080/00102202.2022.2071608.

[19] Ariwibowo, T. H., Safitra, A. G., Fakri, M. H. & Fitrony, M. A. (2022). Numerical study of non-premixed combustion characteristics on supplementary firing of hrsg. *Eksergi, 18(2), 77–84.* https://doi.org/10.32497/eksergi.v18i2.3546.

[20] Noor, M. M., Wandel, A. P. & Yusaf, T. F. (2013). The analysis of recirculation zone and ignition position of non-premixed bluff-body for biogas MILD combustion. *International Journal of Automotive and Mechanical Engineering, 7, 984–995*. https://doi.org/10.15282/ijame.7.2013.15.0082.