UDK 621.43.056 **doi: 10.20998/2078-774X.2023.04.04**

MASOUD HAJIVAND

EXPLORING THE IMPACT OF FLARE EXPANSION ANGLE ON COMBUSTION STABILITY IN AEROENGINE COMBUSTION CHAMBER

In this research, a comprehensive numerical analysis was employed to anticipate the total temperature characteristics, NO emissions, and pattern factor within an annular combustor liner. The investigation focused on the impact of various flare angles (25°, 30°, 35° and 40°) of a double axial swirler across four distinct cases, utilizing computational fluid dynamics (CFD) techniques. The simulations were conducted using ANSYS CFX, incorporating finite-rate chemistry and the eddy dissipation model to simulate the combustion of liquid kerosene (C12H23) with air, specifically accounting for fuel droplet evaporation. Spray modeling, inclusive of a Rosin-Rammler droplet distribution, was implemented to capture the intricate dynamics of liquid kerosene combustion. Thermal and prompt nitrogen oxide (NOx) formation processes were conducted, employing a K-epsilon model for turbulence within a realistic annular combustion chamber. The research further delved into presenting the characteristic features and flame structure, showcasing contour plots of total temperature and NO concentration at the combustor liner outlet. Additionally, cross-sectional analyses along the X-axis from the injector center of the combustor were performed, accompanied by charts illustrating relevant trends along the liner from the injector center. This study introduces two-step kinetic schemes for the combustion of kerosene with air, shedding light on the intricate mechanisms underlying the process. The findings indicate that the optimal outcome, characterized by a lower NO concentration, is associated with the case featuring a reduced flare angle compared to the other configurations.

Key words: combustion chamber- emission- flame structure- nitrogen oxide- eddy dissipation- flare angle- CFD.

МАСУД ХАДЖІВАНД

ВИВЧЕННЯ ВПЛИВУ КУТА РОЗШИРЕННЯ ФАКЕЛА НА СТАБІЛЬНІСТЬ ГОРІННЯ У КАМЕРІ ЗГОРАННЯ АВІАЦІЙНИХ ДВИГУНІВ

В роботі було використано всебічний числовий аналіз, щоб передбачити загальні температурні характеристики, викиди NO та коефіцієнт структури в кільцевій камері згоряння. Проведено дослідження вплив різних кутів спалаху (25°, 30°, 35°, 40°) подвійного осьового завихрювача в чотирьох окремих випадках із застосуванням методів обчислювальної гідродинаміки (*CFD*). Моделювання було проведено за допомогою *ANSYS CFX*, що включало хімічні процеси кінцевої швидкості та модель розсіювання вихрів для імітації горіння рідкого гасу (C12H23) з повітрям, зокрема з урахуванням випаровування крапель палива. Моделювання розпилювання, включно з розподілом крапель Розіна-Раммлера, було реалізовано, щоб зафіксувати складну динаміку горіння рідкого гасу. Процеси термічного та швидкого утворення оксиду азоту (NOx) проводилися з використанням K-епсилонної моделі для турбулентності в реалістичній кільцевій камері згоряння. Далі дослідження було спрямовано на представлення характерних особливостей і структури полум'я, демонструючи контурні графіки загальної температури та концентрації NO на виході з камери згоряння. Крім того, було проведено аналіз поперечного перерізу вздовж осі *X* від центру інжектора камери згоряння, що супроводжувався діаграмами, які ілюструють відповідні тенденції вздовж вкладиша від центру інжектора. Це дослідження представляє двоетапну кінетичну схему згоряння гасу з повітрям, проливаючи світло на складні механізми, які лежать в основі процесу. Отримані дані вказують на те, що при зменшенні кута спалаху зменшується концентрація NO з відпрацьованими газами.

Ключові слова: камера згоряння – емісія – структура полум'я – оксид азоту – вихрове розсіювання – кут спалаху – *CFD*.

Introduction

The meticulous design of a combustion chamber becomes even more crucial when considering the environmental impact. Engineers and researchers are not only focused on ensuring self-sustaining flames within the prescribed temperature limits but also on mitigating the formation of environmentally harmful combustion products, thermos-acoustic characteristics, and temperature uniformity at the outflow of the combustor. The combustion process inevitably leads to the generation of pollutants such as soot, nitrogen oxides (NOx), unburned hydrocarbons (UHC), and other noxious substances at the exit of the combustors.

In response to the environmental challenges associated with combustion, ongoing efforts include modifications to combustion chamber configurations to minimize the production of NOx and UHC. Strategies involve the implementation of advanced emission control technologies, adjustments to fuel-air ratios, and the exploration of alternative fuels, including air swirler modification and fuel spray refinements. Engineers are actively engaged in developing combustion systems that not only meet performance criteria but also adhere to stringent environmental regulations, contributing to cleaner and more sustainable gas turbine operations. This dual emphasis on combustion efficiency and environmental responsibility underscores the evolving landscape of combustion chamber design in the pursuit of a greener and more sustainable future.

Evaluating the fulfillment of these objectives and enhancing combustor design requires a crucial blend of numerical and experimental analyses. This process is both resource-intensive and time-consuming, as highlighted in previous studies [1]. Moreover, it is essential to recognize the meticulous nature of CFD numerical simulations, demanding a profound grasp of the assumptions inherent in the mathematical model and a comprehensive understanding of the relevant physical aspects associated with the phenomena being investigated.

Combustion control can be approached through three main strategies: diminishing peak temperatures in the combustion zone, curtailing the gas residence time in the high-temperature zone, and reducing oxygen concentrations in the combustion zone [2]. These alterations to the combustion process can be implemented either by modifying operational conditions and geometric parameters or through procedural adjustments. This study specifically explores the design of axial swirler flare angle to achieve these objectives. In recent decades, various technologies have been introduced to mitigate NOx emissions in aircraft gas turbines. Prominent examples include Water-Steam Injection, Rich burn Quick Quench-Lean Burn (RQL), Lean Premix Pre-vaporized (LPP), and the Lean-Direct Injection (LDI) engine.

Tuijin Jishu et al. (2017) conducted experiments to improve the combustion organization of small-scale triple swirlers [3]. They compared the combustion performance of single and multi-sector combustors under different conditions. The results showed that the multi-sector combustor's performance deteriorated, with a 25% increase in ignition FAR and a 100% increase in lean blowout FAR compared to the single sector combustor. The main issue was an unsuitable flare angle causing interference in adjacent airflow, especially at low FAR. They determined that a 37.5-degree flare angle was optimal for the multi-sector combustor, highlighting the crucial role of flare angle in combustion efficiency.

In their 2019 study, Kumar et al. investigated the impact of flare angle on the non-reactive flow behavior in gas turbine combustors using a counter-rotating high shear injector [4]. They explored seven flare angles (40 to 70 degree) and observed that changes in flare angle affected the size of the Counter-Rotating Toroidal Recirculation Zone (CTRZ), mean velocity, and turbulence. The study, conducted with constant air and liquid mass flow, revealed that a flare angle of 60 degrees initially increased and then decreased the CTRZ size. This research highlights the significant influence of flare angle on the spray flow-field.

Ayşe Bay et al. (2022) conducted an experimental investigation to assess the impact of flare geometry on the mean flow field generated by radial-radial swirlers in a non-reacting planar combustor test section [5]. Two-dimensional two-component PIV measurements were performed on the mid-plane, complemented by three-dimensional numerical simulations. The research expanded on a previous study involving counter-rotating radial-radial swirlers without flare extension.

The study considered four different swirlers, including a baseline swirler, each investigated with three flare geometries (rounded flare with a radius of 4 mm, chamfered flares at angles of 27.5° and 45°). The rounded flare had both co- and counter-rotating configurations. Analysis of time-averaged flow fields revealed increased radial velocity and decreased axial velocity with the introduction of flare geometry, causing a sudden expansion of the swirling jet. Different flare geometries showed almost identical flow fields, and no Counter-Rotating Zone (CRZ) formation was observed with any flare geometry. Although maximum negative axial velocity values decreased with flare geometries, the recirculating mass flow rate was higher than the baseline swirler due to an increased recirculation radius.

Co-rotating swirlers exhibited a higher recirculating mass flow rate than counter-rotating ones due to a stronger adverse pressure gradient along the central axis of the jet. The study also identified coherent flow structures using the snapshot Proper Orthogonal Decomposition (POD) method, reporting mode shapes and energy contents for swirlers with and without flare geometry. Interestingly, the change in rotation sense and flare geometry did not lead to differences in POD modes and their energy contents under given swirl number and confinement conditions.

The refinement of aero engine combustion chamber, including the modification of swirler details, is a multidisciplinary effort that combines scientific expertise and engineering innovation. The aim is to enhance combustion efficiency, reduce emissions, and ultimately improve the overall performance of aero engines.

The purpose of this study

This study aims to analyze the combustion dynamics in an annular combustor, specifically focusing on the impact of various flare angles (25°, 30°, 35° and 40°) of a double axial swirler for the first to fourth case of study. Using computational fluid dynamics (CFD) techniques, the research investigates total temperature characteristics, NO emissions, and pattern factor in four distinct cases.

Geometric Representation of Annular Combustion Chamber and Boundary Condition

The annular combustor assembly in an aero-engine encompasses various components and intricate design features, including injecting devices, liner, swirlers, diffuser, and cooling air holes on the liner. In this study, a simulation was conducted on a 20-degree sector of a realistic annular combustor from an aeroengine. Fig. 1 illustrates the CAD 3D model of the combustor, featuring a double radial swirler and a fuel injector, along with air admission holes on the liner. The combustor domain receives inputs of liquid kerosene (C10H23) and air separately. To enable simulation, the annular combustor sector underwent meshing using an unstructured tetrahedral meshing method, resulting in a total of approximately 7,985,951 elements. Prismatic layers were included around the walls of the annular combustor. The 2D cross-sectional representation of the mesh is depicted in Fig. 2.

At the combustor inlet, the incoming air has a temperature of 750 K, a mass flow rate of 0.8 Kg/s, and an average static pressure of 15 atm at the outlet. The fuel, injected from the surface of the injector, has a temperature of 320 K, a velocity of 50 m/s, and a mass flow rate of 0.016 Kg/s. The atomization of the liquid fuel follows the Rosin-Rammler law, characterized by an average droplet diameter of 30 microns and a non-uniform exponent of 2. The injection plane has a radius of 3mm and a hollow cone angle of 40 degrees.

The Cases of Investigation

In the conducted numerical experiment, four different cases were investigated, each featuring distinct axial swirler flare angles of 25, 30, 35, and 40 degrees. Fig. 3 provides a visual representation of the liner structure and the intricate details of the axial double swirler, specifically highlighting the flare associated with each angle.

Computational Modeling

The simulation undertaken employs a steady-state calculation method using the ANSYS CFX code. For a thorough grasp of the governing equations, encompassing the continuity equation, conservation equation of momentum, energy equation, and species equation, readers are advised to consult the ANSYS CFX help documents [6]. It is pertinent to note that the model parameters for all turbulence models adhere to the default settings of the ANSYS CFX code.

The mathematical formulations describing fuel combustion are rooted in the equations of conservation of mass, momentum, and energy, along with additional supplementary equations for turbulence and combustion. In this study, the standard k-ε turbulence model is employed. The equations governing turbulent kinetic energy (k) and the dissipation rate of turbulent kinetic energy (ε) are solved. Various turbulence models have been proposed by different authors, differing in complexity and applicability. They entail solving different numbers of differential equations. The turbulence model incorporated in this investigation is the high Reynolds number k- ε two-equation model. This model necessitates solving two differential equations for two turbulence properties: the kinetic energy of turbulence (k) and its dissipation rate (epsilon). The model strikes a balance between complexity and accuracy, having been widely employed in numerous studies, demonstrating adequacy across a broad spectrum of flow scenarios. The governing differential equations are outlined by Launder and Spalding (1974).

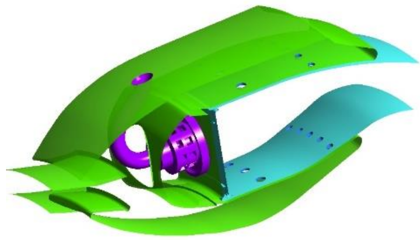

Fig. 1 – 3D CAD model of combustor

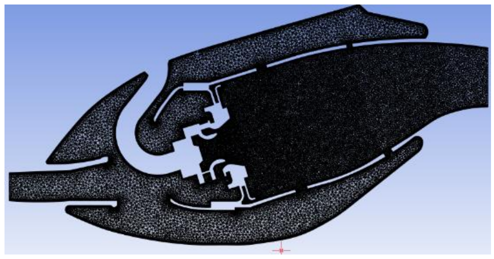

Fig. 2 – 2D view of the cross-section meshes

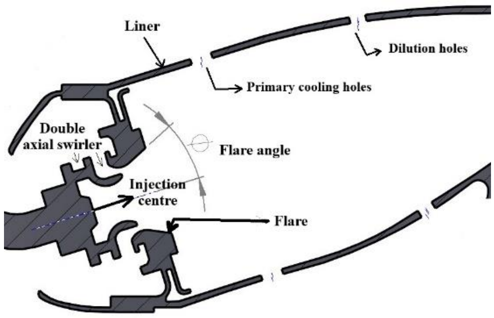

Fig. 3 – Cross section of structure of the double axial swirler including the liner schematic

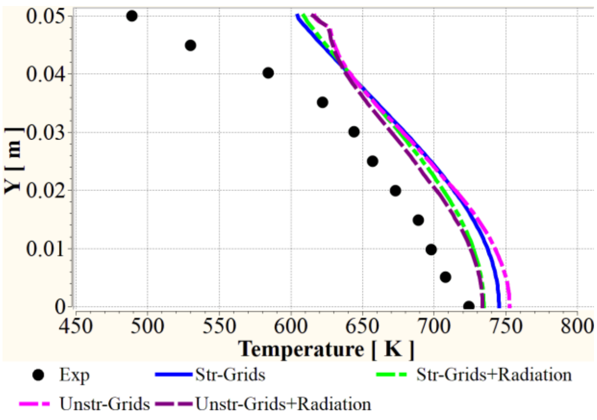

Fig. 4 – Exit gas temperature distribution obtained at the outlet of combustor in real experiment done by Ghose P. et al. [8] and numerical experiments done by Masoud Hajivand (2021) [9], including structed and unstructured grids, with and without radiation effect

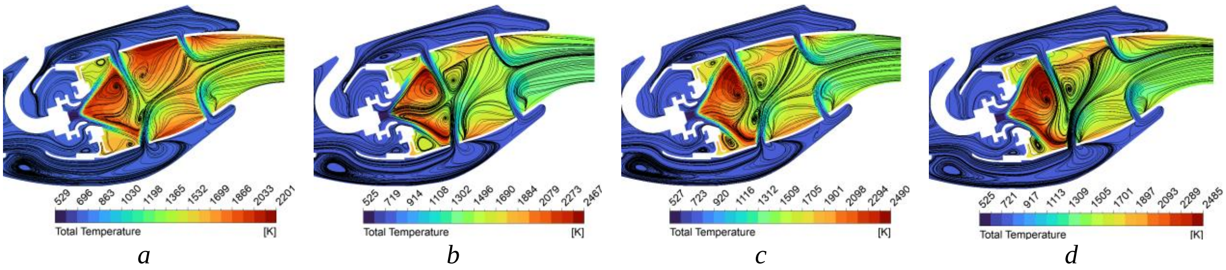

Fig. 5 – Cross section of total temperature distribution contours including the streamlines in various flare angle: *a* – 25°; *b* – 30°; *c* – 35°; *d*– 40°

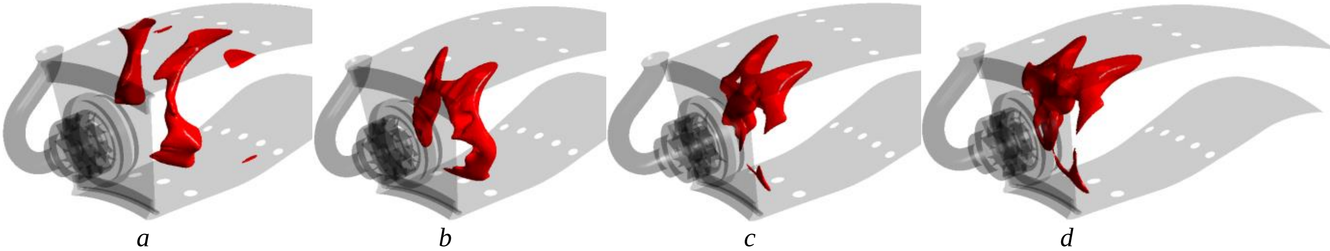

Fig. 6 – Iso surfaces of maximum total temperature in flame tube in various flare angle: *a* – 25°; *b* – 30°; *c* – 35°; *d*– 40°

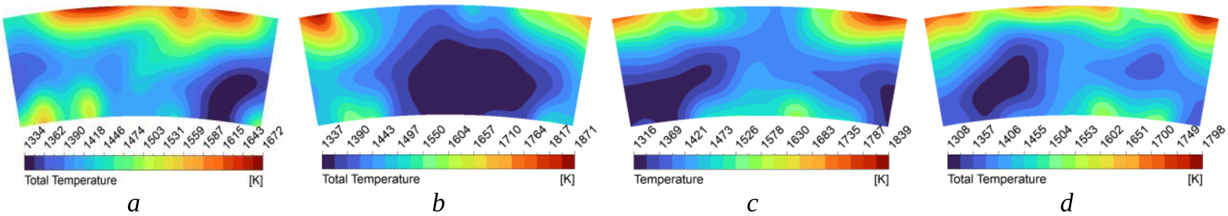

Fig. 7 – Contour of total temperature distribution at the outlet of combustor in various flare angle: *a* – 25°; *b* – 30°; *c* – 35°; *d*– 40°

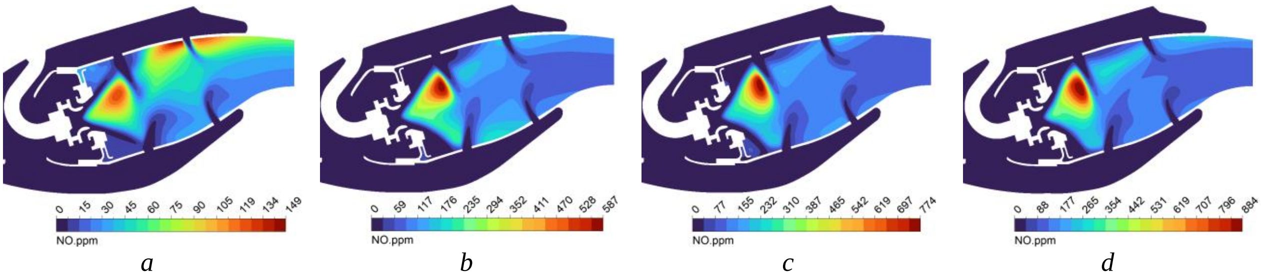

Fig. 8 – Cross section of NO distribution contours along the combustion chamber in various flare angle: *a* – 25°; *b* – 30°; *c* – 35°; *d*– 40°

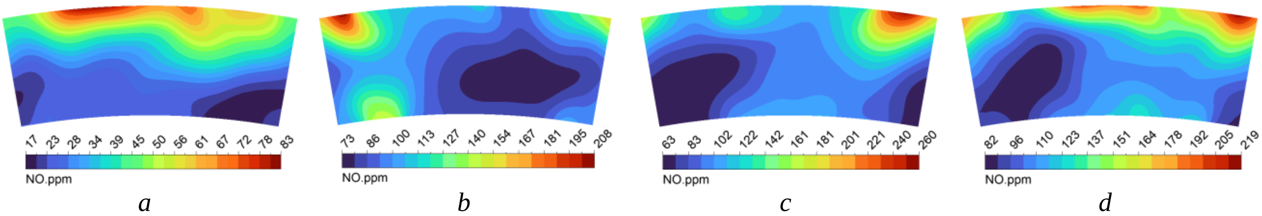

Fig. 9 – Contour of NO distribution at the outlet of combustor in various flare angle: *a* – 25°; *b* – 30°; *c* – 35°; *d*– 40°

Combustion Modeling

The investigation incorporated a hybrid approach involving the combination of finite rate chemistry and the eddy dissipation model. In the context of the combined Finite Rate Chemistry/Eddy Dissipation Model, reaction rates are initially computed independently for each model, and subsequently, the minimum of the two rates is utilized. This process is applied individually for each reaction step. Consequently, while the rate for one step may be constrained by chemical kinetics, another step may concurrently be limited by turbulent mixing at the same physical location. Moreover, the approach allows for the application of different combustion models to each step within a multi-step scheme. Certain predefined schemes take advantage of this flexibility, irrespective of the overall model selection.

The combined model is applicable across a broad spectrum of configurations, especially in turbulent flows. It remains valid for reactions with Damköhler numbers spanning from low to high, signifying a range where chemistry is either slow or fast compared to the turbulent time scale. This model is particularly recommended when reaction rates are constrained by turbulent mixing in one region of the domain and kinetics in another. However, it is noteworthy that the Eddy Dissipation model can, in certain cases, exhibit greater robustness compared to Finite Rate Chemistry or the combined model.

The combustion reaction of JetA with air is represented through a simplified single-species surrogate. Utilizing a two-step global mechanism for JetA (C12H23), the process involves a primary step for fuel oxidation into CO and H2O, followed by a secondary step for CO oxidation into CO2:

$$C_{12}H_{23} + 11.75O_2 \rightarrow 12CO_2 + 11.5H_2O, \quad (1)$$

$$CO + 0.5O_2 \rightarrow CO_2. \quad (2)$$

The Arrhenius coefficients governing this mechanism can be referenced in [7].

In gas flames with temperatures surpassing 1800K, the primary source of NOx is the abundance of N and O radicals. This phenomenon is chiefly influenced by the Zeldovich mechanism proposed in 1947, as denoted by the reactions:

$$O_2 + N_2 \rightarrow NO + N, \quad (3)$$

$$N + O_2 \rightarrow NO + O. \quad (4)$$

Furthermore, the combination of reaction (3) with either of reactions (4) or the overall reaction results in the formation of two NO molecules:

$$N_2 + O_2 \rightarrow 2NO. \quad (5)$$

These reactions collectively provide a simplified representation of the combustion dynamics of JetA with air, emphasizing key steps in fuel oxidation and NOx formation.

Validation of the Simulation

As a benchmark for validation, a simplified combustion chamber, consistent with the experimental setup conducted by Prakash Ghose et al. [8], was simulated. The experimental study by Prakash Ghose et al. utilized a can combustor featuring kerosene as the liquid fuel, mixed with air in an atmospheric condition.

The focus of the validation was on comparing the outlet temperature obtained from the simulation with the corresponding experimental results of Prakash Ghose et al. specifically, the outlet temperature in the simulated combustion chamber was compared to the experimental data (Fig. 4) obtained from Prakash Ghose et al.'s study.

The mean deviation between the simulated and experimental outlet temperatures was calculated, yielding a result of approximately 7 %. This low mean deviation suggests a close agreement between the simulated and experimental data, indicating that the simulation model accurately captures the thermal behavior within the combustion chamber.

This validation provides confidence in the predictive capabilities of the simulation model. The consistency with the experimental results of Prakash Ghose et al. reinforces the credibility of the simulation results, demonstrating their suitability for predicting temperature profiles and NOx formation in similar combustion chambers.

The validation results highlight the capability of the ANSYS CFX simulation model to accurately predict the outlet temperature in a combustion chamber, as evidenced by the close agreement with experimental data from Prakash Ghose et al. This validation process strengthens the reliability of the simulation results and underscores the potential of the model for analyzing and optimizing combustion processes in aero engine applications.

Additionally, the related simulations were conducted under various conditions, including scenarios with and without radiation effects and employing both structured and unstructured grids. For further details, please refer to the relevant publication authored by Masoud Hajivand [9].

Results and discussion

This investigation provides a detailed analysis of temperature distribution behavior and NO formation within the combustor, including the outlet, as observed through Computational Fluid Dynamics (CFD) simulations. The focus is on understanding the impact of flare expansion angle on combustion stability in an aero-engine combustion chamber. Within this context, a pattern factor is introduced to assess outlet temperature uniformity, a crucial consideration for effective combustion chamber design. Moreover, the study goes beyond theoretical frameworks by presenting the iso-surface of the maximum temperature distribution along the flame tube. These insights are derived from CFD simulations, specifically conducted to evaluate the effects of flare expansion angle on combustion stability. Such a comprehensive exploration contributes valuable information for optimizing aero engine combustion chamber performance and enhancing overall combustion stability.

Temperature Distribution and pattern factors

The CFD simulation results provide valuable insights into the impact of varying flare expansion angles (25, 30, 35, and 40 degrees) on combustion stability and characteristics within the annular aeroengine combustion chamber. Analyzing the cross-sectional temperature distribution (shown in Fig. 5) reveals that as the flare angle increases, the maximum temperature within the combustor rises progressively from 2200K for a flare angle of 25 degrees to 2466K, 2490K, and 2486K for flare angles of 30, 35, and 40 degrees, respectively. The observed trend indicates that an increased flare angle results in elevated temperatures within the primary combustion zone. This phenomenon is attributed to the presence of a broader central toroidal recirculation zone (CTRZ) and an enhanced mixing rate, which influences the residence time of hot gases in this specific area.

To elaborate further, the flare angle refers to the angle formed by the flare's structure, and as this angle widens, it leads to the expansion of the central toroidal recirculation zone. This expanded zone plays a crucial role in the combustion process, affecting the level of mixing and, consequently, the residence time of hot gases. The longer these gases remain in the primary combustion zone, the higher the temperatures achieved. The relationship between the flare angle and temperature in the primary combustion zone is influenced by the geometry of the central toroidal recirculation zone and the associated mixing rate, ultimately impacting the residence time of hot gases within this critical area. For a clearer comprehension of the maximum temperature distribution along the flame tube, please refer to Fig. 6. This figure illustrates iso-surfaces representing the maximum temperature. These iso-surfaces visually depict regions within the flame tube where the temperature reaches its peak values.

Examining the temperatures at the outlet of the combustor (shown in Fig. 7), crucial for assessing combustion efficiency and stability, a similar trend emerges. With increasing flare angle, the maximum outlet temperatures rise from 1673K to 1871K, 1838K, and 1801K, corresponding to flare angles of 25, 30, 35, and 40 degrees, respectively. Simultaneously, the minimum outlet temperatures exhibit a similar pattern, increasing from 1333K to 1336K, 1317K, and 1307K for the same flare angles.

The pattern factors, indicative of combustion uniformity, mirror the trends observed in temperature distributions. As the flare angle increases, the pattern factor shows an increasing trend. This is evident in the values of 0.34, 0.63, 0.58, and 0.53 for flare angles of 25, 30, 35, and 40 degrees, respectively. A lower pattern factor indicates more temperature uniformity, suggesting that wider flare angles may lead to increased temperature gradients and non-uniform combustion within the chamber.

The observed trends in temperature and pattern factors can be attributed to the alteration of the recirculation zone within the combustion chamber as the flare angle changes. A wider flare angle results in a larger recirculation zone, influencing the mixing and combustion characteristics. The decrease in the swirler flare angle enhances the recirculation, leading to increased hot gas residence time and more efficient mixing, ultimately influencing the observed temperature patterns and combustion uniformity.

CFD simulation results demonstrate a clear correlation between flare expansion angles, temperature distributions, and combustion stability. The widening of the flare angle contributes to higher temperatures within the combustor, impacting both the cross-sectional and outlet temperature profiles. The observed changes in pattern factors suggest a shift towards less uniform combustion. The role of the recirculation zone emerges as a key factor, influencing residence time and mixing efficiency in response to alterations in the swirler flare angle.

NO Distribution

The CFD simulations conducted in Ansys CFX to explore the impact of flare expansion angle on combustion stability revealed significant variations in NO formation. Across the combustor's cross-section, the maximum NO concentrations (shown in Fig. 8) exhibited an upward trend from 67 ppm (flare angle of 25 degrees) to 106 ppm (30 degrees), 110 ppm (35 degrees), and 187 ppm (40 degrees). Similarly, the level of NO concentration at the outlet (shown in Fig. 9), maximum NO concentrations followed a similar pattern with flare angles: 83 ppm (25 degrees), 208 ppm (30 degrees), 260 ppm (35 degrees), and 219 ppm (40 degrees).

The consistent increase in maximum NO concentrations with higher flare angles strongly suggests a correlation between flare angle and NO production, likely attributed to alterations in fuel-air mixing dynamics. The observed variations at the outlet underscore the intricate relationship between flare angles and combustion characteristics, influencing both maximum and minimum NO concentrations.

As the swirl flare angle increases, the expansion of the Central Toroidal Recirculation Zone (CTRZ) occurs. Consequently, this expanded zone interacts with the outflow from the primary cooling holes in a high-speed region, facilitating more thorough combustion at elevated temperatures. This intensified combustion process results in an augmented formation of nitrogen oxides (NO) within the primary combustion zone.

Conversely, a lower swirl flare angle leads to a narrower CTRZ. This reduced zone interacts with the outflow from the cooling holes surrounding the primary combustion zone, characterized by a lower flow velocity. In this scenario, a lower temperature is established within that region, leading to a diminished formation of NO compared to flames with higher angles. The significance of the recirculation zone, influenced by the swirler, emerges as a critical factor. As flare angles increase, a larger recirculation zone may contribute to prolonged residence time of combustion products, promoting NO formation. This underscores the necessity for optimizing aerodynamics to strike a balance between combustion stability and minimizing NO emissions.

Conclusion

In conclusion, this comprehensive research delves into the intricate dynamics of combustion stability within an aeroengine combustion chamber, with a specific focus on the impact of flare expansion angles of a double axial swirler. Employing advanced computational fluid dynamics (CFD) techniques, the study investigates temperature characteristics, NO emissions, and pattern factors across four distinct flare angles (25°, 30°, 35°, and 40°).

The findings emphasize a clear correlation between flare expansion angles and combustion stability. As the flare angle increases, both cross-sectional and outlet temperatures within the combustor rise, reflecting the influence of a broader central toroidal recirculation zone (CTRZ) and enhanced mixing rates. The widening flare angle contributes to higher temperatures and, consequently, less uniform combustion, as indicated by increasing pattern factors which are shown in Fig. 10. The study underscores the critical role of the recirculation zone in shaping residence time and mixing efficiency. Furthermore, the research sheds light on the intricate relationship between flare angles and nitrogen oxide (NO) formation. Higher flare angles result in an augmented formation of NO, attributed to intensified combustion processes within the expanded CTRZ. Conversely, lower flare angles lead to reduce NO formation due to a narrower recirculation zone with lower temperatures. In addition to the observed trends in temperature distributions and NO formation, it is crucial to highlight the average temperatures and NO concentrations across the entire volume of the combustion chamber for each flare angle, as illustrated in Fig. 11, Fig. 12, and Fig. 13.

The average temperatures throughout the combustion chamber volume reveal distinctive patterns for each flare angle: 1130 K for a flare angle of 25 degrees, 1151 K for 30 degrees, 1160 K for 35 degrees, and 1162 K for 40 degrees. These values provide a comprehensive understanding of the thermal characteristics within the combustor, showcasing the influence of flare angles on temperature profiles.

Simultaneously, the average NO concentrations in the combustion chamber volume display clear trends, emphasizing the impact of flare angles on NO formation. For a flare angle of 25 degrees, the average NO concentration is 18 ppm, increasing to 53 ppm for 30 degrees, 57 ppm for 35 degrees, and 68 ppm for 40 degrees. These results underscore the intricate relationship between flare angles and NO production, providing valuable insights into the combustion dynamics.

Furthermore, the study assesses the average NO formation specifically at the outlet of the combustion chamber for each flare angle. The average NO concentrations at the outlet follow a similar trend, with values of 35 ppm for 25 degrees, 97 ppm for 30 degrees, 100 ppm for 35 degrees, and 114 ppm for 40 degrees. These findings highlight the variations in NO concentrations at the outlet, further emphasizing the role of flare angles in influencing combustion efficiency and emission characteristics.

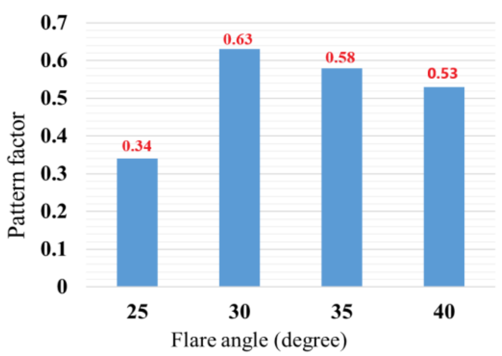

Fig. 10 – Pattern factor for the various flare angle

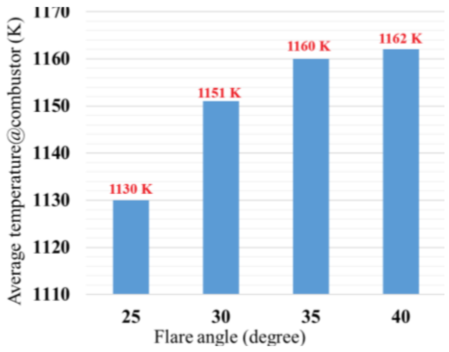

Fig. 11 – Average temperature distribution in the whole combustor

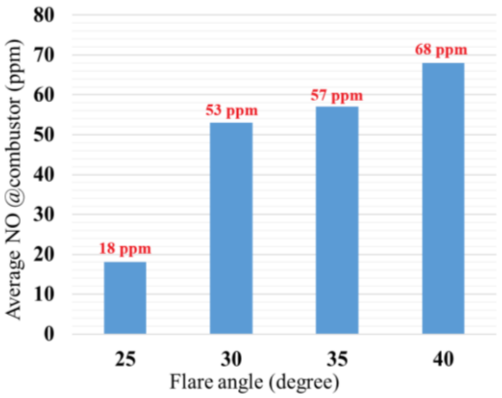

Fig. 12 – Average NO distribution in the whole combustor

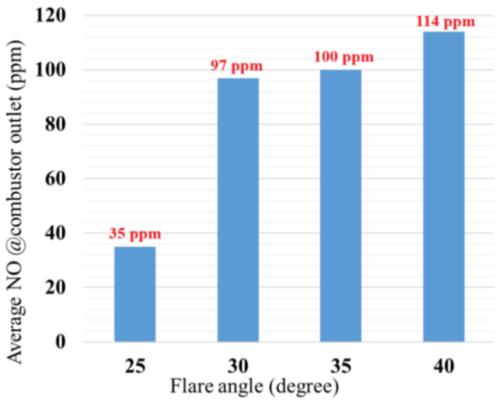

Fig. 13 – Average NO distribution at the outflow combustor

It is noteworthy that, across all cases of study, the average temperature at the outlet of the combustor remains consistent at 1435 K. This uniformity in outlet temperatures suggests a stable and optimized combustion process, reinforcing the potential for achieving both enhanced efficiency and reduced emissions in aeroengine applications. In conclusion, the integration of average temperature and NO concentration data enhances the comprehensiveness of this study, providing a holistic view of combustion dynamics within the aeroengine combustion chamber.

The observed trends underscore the significance of optimizing flare angles to achieve the desired balance between combustion stability, temperature uniformity, and NO emissions, contributing to the ongoing efforts for cleaner and more sustainable gas turbine operations.

The combustion chamber with a 25-degree flare angle demonstrates superior performance in multiple aspects. It exhibits the lowest average NO concentrations, ensuring compliance with strict environmental regulations and minimizing nitrogen oxide (NOx) emissions.

Additionally, this configuration showcases a more uniform temperature distribution, contributing to efficient combustion and mitigating temperature-related issues. Despite maintaining a consistent outlet temperature of 1435 K, the 25-degree flare angle case stands out for its lower NO concentrations, indicating stable combustion.

This optimized balance between combustion stability, temperature uniformity, and NO emissions underscores its suitability for achieving optimal combustion efficiency in aeroengine applications. Furthermore, the validation of simulation results against experimental data in the future enhances the reliability of the model, instilling confidence in its predictive capabilities for similar combustion chambers.

References (transliterated)

1. Fuligno, L., Micheli D., Poloni C. (2009), “An Integrated Approach for Optimal Design of Micro Gas Turbine Combustors”, *Journal of Thermal Science*, vol. 18, pp. 173–184, https://doi.org/10.1007/s11630-009-0173-7.
2. World Bank Group (1998), “Nitrogen Oxides: Pollution Prevention and Control*”, Pollution Prevention and Abatement Handbook: Toward Cleaner Production*, no. 19128, pp. 245–249, ISBN 978-0-8213-3638-0, https://doi.org/10.1596/0-8213-3638-X.
3. Ding Guo-yu, He Xiao-min, JIANG Ping, Zhu Yi-xiao, Jin Yi (2017), “Effects of Flare Angle on Combustion Performance of a Triple Swirler Combustor”, *Journal of Propulsion Technology*, no. 02, pp. 376–385, https://doi.org/10.13675/j.cnki.tjjs.2017.02.017.
4. Kumar, S., Chaudhuri, S., Basu, S. (2019), “On Effect of the Flare Angle on the Behaviour of the Flow Field of Twin-Radial Swirlers/High Shear Injector”, *The American Society of Mechanical Engineers (ASME) : Gas Turbine India Conference*, GTINDIA2019-2537, V002T04A014, 6 p, https://doi.org/10.1115/gtindia2019-2537.
5. Ayşe Bay, Fırat Kıyıcı, Mustafa Perçin (2022), “Effect of Flare Geometry on the Flow Field of Radial-Radial Swirlers”, *The American Society of Mechanical Engineers (ASME) : Turbo Expo: Power for Land, Sea, and Air*, GT2022-83234, V03BT04A047, 12 p, https://doi.org/10.1115/gt2022-83234.
6. ANSYS, Inc. (2015), *ANSYS CFX-Solver Theory Guide*, United states, Access mode: https://www.ansys.com/ (accessed 12 December 2023).
7. Valachovic T. G. (1993), “Numerical predictions of idle power emissions from gas turbine combustors”, *The American Society of Mechanical Engineers (ASME) : Turbo Expo: Power for Land, Sea, and Air*, no. 93-GT-175, V03AT15A026, 7 p, https://doi.org/10.1115/93-GT-175.
8. Ghose, P., Datta, A., Ganguly, R., Mukhopadhyay, A., & Sen, S. (2017, “Modelling of Soot Formation in a Kerosene Spray Flame”, *Modeling and Simulation of Turbulent Combustion : Energy, Environment, and Sustainability*, pp. 363–394, ISBN 978-981-10-7409-7, https://doi.org/10.1007/978-981-10-7410-3_12.
9. Hajivand, M., & Dolmatov, D. A. (2021), “Numerical Analysis of Kerosene Spray Combustion in Various Injection Preparations and Emission Behaviour Assessment”, *Collection of Scientific Papers of Admiral Makarov National University of Shipbuilding*, no. 1(484), pp. 125–141, ISSN 2311-3405, https://doi.org/10.15589/znp2021.1(484).18.

Список літератури

1. Fuligno, L. An Integrated Approach for Optimal Design of Micro Gas Turbine Combustors / L. Fuligno, D. Micheli, C. Poloni // Journal of Thermal Science. – 2009. – Vol. 18. – P. 173–184. – DOI: https://doi.org/10.1007/s11630-009-0173-7.
2. Nitrogen Oxides: Pollution Prevention and Control / World Bank Group // Pollution Prevention and Abatement Handbook: Toward Cleaner Production. – 1998. – No. 19128. – P. 245–

249. – ISBN 978-0-8213-3638-0. – DOI: https://doi.org/10.1596/0-8213-3638-X.

3. Ding Guo-yu Effects of Flare Angle on Combustion Performance of a Triple Swirler Combustor / Ding Guo-yu, He Xiaomin, JIANG Ping, Zhu Yi-xiao, Jin Yi // Journal of Propulsion Technology. – 2017. – No. 02. – P. 376–385. – DOI: https://doi.org/10.13675/j.cnki.tjjs.2017.02.017.
4. Kumar, S. On Effect of the Flare Angle on the Behavior of the Flow Field of Twin-Radial Swirlers/High Shear Injector / S. Kumar, S. Chaudhuri, S. Basu // The American Society of Mechanical Engineers (ASME) : Gas Turbine India Conference. – 2019. – GTINDIA2019-2537, V002T04A014. – 6 p. – DOI: https://doi.org/10.1115/gtindia2019-2537.
5. Bay, A. Effect of Flare Geometry on the Flow Field of Radial-Radial Swirlers / Ayşe Bay, Fırat Kıyıcı, Mustafa Perçin // The American Society of Mechanical Engineers (ASME) : Turbo Expo: Power for Land, Sea, and Air. – 2022. – GT2022-83234, V03BT04A047. – 12 p. – DOI: https://doi.org/10.1115/gt2022-83234.
6. ANSYS, Inc. ANSYS CFX-Solver Theory Guide / ANSYS, Inc. – United states, 2015. – URL: https://www.ansys.com/ (дата звернення: 12.12.2023)
7. Valachovic T. G. Numerical predictions of idle power emissions from gas turbine combustors / T. G. Valachovic // The American Society of Mechanical Engineers (ASME) : Turbo Expo: Power for Land, Sea, and Air. – 1993. – No. 93-GT-175, V03AT15A026. – 7 p. – DOI: https://doi.org/10.1115/93-GT-175.
8. Ghose, P. Modelling of Soot Formation in a Kerosene Spray Flame / P. Ghose, A. Datta, R. Ganguly, A. Mukhopadhyay, S. Sen // Modeling and Simulation of Turbulent Combustion : Energy, Environment, and Sustainability. – 2017. – P. 363–394. – ISBN 978-981-10-7409-7. – DOI: https://doi.org/10.1007/978-981-10-7410-3_12.
9. Hajivand, M. Numerical Analysis of Kerosene Spray Combustion in Various Injection Preparations and Emission Behaviour Assessment / M. Hajivand, D. A. Dolmatov // Collection of Scientific Papers of Admiral Makarov National University of Shipbuilding. – 2021. – No. 1(484). – P. 125–141. – ISSN 2311-3405. – DOI: https://doi.org/10.15589/znp2021.1(484).18.

Received (надійшла) 22.10.2023

Відомості про авторів / About the Authors

Хаджіванд Масуд (Hajivand Masoud) – здобувач кафедри конструкції авіаційних двигунів Національного аерокосмічного університету ім. М. Е. Жуковського «ХАІ», м. Харків, Україна; e-mail: m.hajivand82@gmail.com; ORCID: https://orcid.org/0000-0002-9990-9761, Scopus 57225272766.