Gautam Choubey
Assistant Professor
Department of Mechanical Engineering
National Institute of Technology Silchar
Assam, 788010
India

Gurkreetkaur Brar
Department of Mechanical and Aerospace Engineering, IITRAM Ahmedabad, 380026
Gujarat
India

Mrinal Kaushik
Associate Professor
Department of Aerospace Engineering
Indian Institute of Technology Kharagpu
Kharagpur, West Bengal, 721302
India

Numerical Investigation of Strut-Dual Cavity Flame Holders for Enhanced Combustion Performance in Scramjet Engines

Scramjet engines hold significant promise for hypersonic propulsion, yet the complexities of supersonic combustion pose considerable challenges. Computational Fluid Dynamics (CFD) offers a cost-effective approach to studying turbulent reactive flows in such systems, particularly addressing the critical turbulence-chemistry interactions that differ from conventional combustion. This study employs two-dimensional Reynolds-averaged Navier–Stokes (RANS) simulations to evaluate a novel strut-dual cavity flame holder in scramjet combustors. Among various configurations analyzed, the double-step cavity demonstrated a superior balance, achieving high combustion efficiency with moderate total pressure loss. The numerical model was validated against experimental data, confirming its reliability and predictive accuracy. The findings highlight the effectiveness of integrating cavity-based geometries with strut injectors in enhancing fuel-air mixing and stabilizing combustion. This research underscores the strut-dual cavity flame holder's potential as a practical solution for advancing scramjet performance in hypersonic applications.

Keywords: *Supersonic Combustion, Strut-Dual Cavity Flame Holder, CFD, Scramjet Engine, Combustion Enhancement*

1. INTRODUCTION

The scramjet engine is regarded as a highly promising technology for achieving sustained hypersonic propulsion. Significant efforts have been dedicated to its development since the 1960s, driven by the demand for hypersonic vehicles in both civilian and military sectors. These engines have garnered substantial global interest due to their potential applications in defense systems, such as advanced weaponry, and their capability to enable hypersonic travel for commercial transportation. Additionally, scramjets are regarded as a pillar of technology for the development of reusable space-access vehicles, offering the possibility of reduced costs for space exploration and transport missions. Despite these promising applications and decades of research progress, the practical implementation of scramjet engines remains a challenge. One of the primary hurdles lies in their unique supersonic combustion mechanism, which often results in engine blow-off during startup, impeding reliable operation [1-4].

Among the various scramjet configurations, the strut-based combustor has emerged as a widely adopted design [5]. In this configuration, fuel is injected into the airflow through strategically placed injection holes on the strut. This setup enhances fuel penetration depth and promotes efficient mixing and combustion. Over time, numerical simulations have proven to be a powerful tool for analyzing and optimizing the performance of strut-based scramjet combustors. Notable studies, such as those by Oevermann et al. [6], Fureby et al. [7], and Génin et al. [8], have leveraged computational methods to refine strut designs and assess their impact on the combustion process. For example, Huang et al. [9] investigated the influence of strut positioning and fuel properties on combustion behavior. Their results revealed that moving the strut further downstream from the combustor inlet improved combustion efficiency. They also noted that increasing the fuel injection pressure and temperature shifted combustion disturbances closer to the strut base, thereby enhancing overall performance [10].

Building on this foundation, Chen et al. [11] examined the auxiliary strut's impact on mixing performance by varying the injection hole diameter and angle. Their findings indicated that while hole diameter had minimal influence, increasing the injection angle significantly improved hydrogen-air mixing efficiency in the ignition zone by approximately 15%. Similarly, Choubey et al. explored double-strut [12-15] combustors, demonstrating that an attack angle of $\alpha = 0°$ minimized hydrogen ignition time and maximized combustion efficiency at around 83%.

In recent years, cavity-based scramjet designs have gained attention for their ability to enhance fuel-air mixing and stabilize combustion. Cavities act as flame holders [16], generating recirculation zones that support ignition and flame stabilization, even in supersonic flow conditions. Combining strut and cavity configurations integrates their respective advantages: the strut ensures effective fuel injection and penetration, while the cavity enhances residence time and mixing, leading to superior

Received: January 2025, Accepted: April 2025
Correspondence to: Dr Gautam Choubey
Assistant Professor, Department of Mechanical Engineering, NIT Silchar, 788010, Assam, India
E-mail: gautamchoubey@mech.nits.ac.in
doi: 10.5937/fme2502289C

combustion performance [17]. Dual-cavity designs, in particular, have shown promise for optimizing scramjet functionality in aerospace applications. For instance, cavity flame holders create low-drag recirculation zones, while strategically positioned small struts near the cavity's leading edge facilitate effective fuel injec–tion and maintain the combustion zone's stability away from the cavity walls.

Recent research has increasingly focused on opti–mizing strut-cavity configurations. For example, Kuang-Yu Hsu [18, 19] explored flame propagation and ignition efficiency with various strut designs, observing successful combustion across different setups. Gu Hon–gbin [20] compared wall-injection methods for small struts using optical measurements, while Nathan R. Grady [21] employed experiments and large eddy simu–lations (LES) to investigate cavities with and without upstream struts, highlighting their effect on enhancing the recirculation zone. In this context, B. Rašuo and his research group [22-24] have made significant contri–butions to the field of high-speed aerodynamics. Their work has addressed critical challenges such as flow separation in overexpanded nozzles and jet interactions in thrust vector control systems, both of which are directly relevant to shock-induced separation and mixing in air-breathing propulsion systems like scram–jets. Moreover, their studies [25] on projectile aero–dynamics and missile propulsion under high-speed con–ditions provide valuable insights into compressible flow behavior - a core aspect of scramjet operation. Impor–tantly, their [26] emphasis on wind tunnel experi–mentation to validate supersonic flow models under–scores the importance of experimental benchmarking in scramjet research, offering a robust foundation for CFD validation and hybrid modeling approaches.

Despite these advancements, numerical studies on strut-cavity combustor configurations remain limited, particularly regarding the effects of dual-cavity designs. Computational fluid dynamics (CFD) has emerged as an indispensable tool for reducing the high costs associated with experimental and flight tests. It also supports theoretical analyses by providing detailed insights into flow-field behavior within airbreathing propulsion systems. This study addresses this gap by analyzing four distinct strut-cavity combinations: (i) strut + double circular cavity, (ii) strut + double step cavity, (iii) strut + circular + step cavity, and (iv) strut + rectangular + step cavity. The investigation thoroughly examines their ef–fects on the combustion flow field, contributing valu–able insights for the engineering design of combined-cycle engines for space vehicles.

2. COMPUTATIONAL MODEL AND GRID GENERATION

The scramjet combustor geometry used in this study is shown in Fig. 1 and is based on the well-established DLR scramjet combustor configuration. It employs a two-dimensional planar model to simulate key flow features while maintaining geometric fidelity to expe–rimental setups. The inlet height is 50 mm, with a uniform supersonic air inflow at Mach 2.0. The overall length of the computational domain is 340 mm, and the outlet height increases to 62 mm, resulting in a 3° divergence along the upper wall only, beginning at 100 mm downstream from the inlet. This asymmetric divergence (mirroring the original DLR design) is intended to support controlled expansion and better simulate realistic combustor pressure gradients. The lower wall remains straight, and all walls are treated as no-slip adiabatic boundaries. A strut-based flame holder is positioned 77 mm from the inlet, having a 6° half-angle wedge shape [27] and a total length of 32 mm from its tip to the base. Hydrogen fuel is injected at Mach 1.0, parallel to the main airflow, through an orifice located at the base of the strut, which functions simultaneously as a flame holder and injector. To further promote mixing and flame stabilization, four different cavity configurations were introduced into the model :(i) strut with double circular cavities, (ii) strut with double step cavities, (iii) strut with a circular and step cavity combination, and (iv) strut with a rectangular and step cavity combination. The corres–ponding detailed dimensions are provided below. These cavities are mounted on the lower wall at a fixed distance of 120 mm from the inlet, downstream of the strut fuel injector to interact with both the boundary layer and the shock-induced shear layers. The inter-cavity spacing is 5 mm.

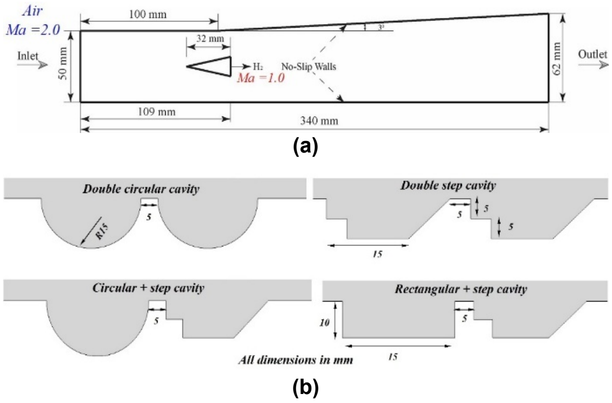
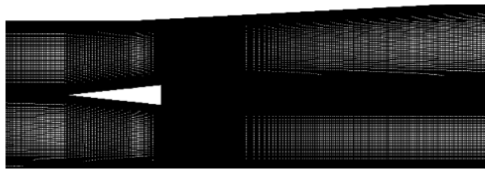
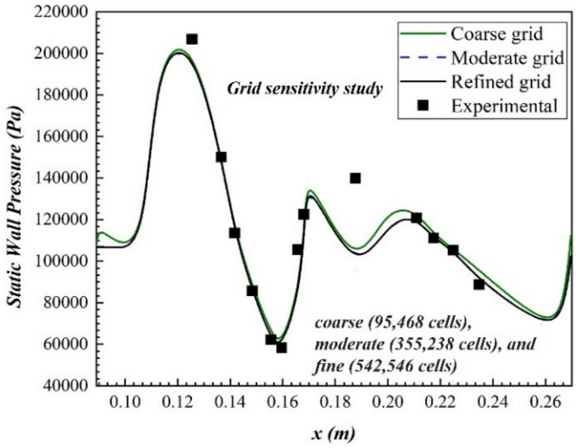


Figure 1. Schematic representation of the (a) DLR scramjet model (b) proposed cavities in strut-based scramjet

Structured computational meshes were generated using ANSYS Meshing [28], with a high mesh density concentrated near the strut and wall regions [Fig. 2(a)]. This meticulous grid arrangement ensures accurate resolution of shock waves and detailed flow structures within the combustor. To achieve precise predictions in the wall-adjacent region, special attention was given to the boundary layer mesh, incorporating a first layer thickness of 0.001 mm for enhanced resolution near the walls ($y^+ \approx 1.25$). Fig. 2 (b) illustrates the variation in pressure along the combustor's bottom wall for three different grid resolutions. Simulations are conducted using meshes comprising 95,468, 355,238, and 542,546 cells respectively.



(a)



(b)

Figure 2. (a) Grid generation and (b) grid sensitivity study (variation in pressure along the combustor's bottom wall)

The numerical results demonstrate convergence and stability as the grid density reaches a certain threshold. For this study, refined grids (542,546 cells) were se–lected to ensure reliable outcomes. The computational data closely matches experimental results, both in the intensity and spatial locations of the shocks and expan–sion fans.

3. COMPUTATIONAL SIMULATION METHOD

3.1 Numerical approaches

The Reynolds-averaged Navier-Stokes (RANS) appro–ach, utilizing Reynolds decomposition, is a well-estab–lished method for modeling turbulence across various scales. It remains one of the most computationally effi–cient and widely applied techniques for addressing real-world flow challenges. In this study, the RANS frame–work is utilized to model the supersonic combustion flow field. A steady-state, density-based solver opera–ting in double precision is employed to solve the governing equations [29-34]. Numerical discretization is carried out using the finite volume method, with a second-order upwind scheme ensuring accuracy. Convective fluxes are computed using the Advection Upstream Splitting Method (AUSM) to capture shock waves effectively.

The choice of turbulence model is crucial in accu–rately simulating chemically reactive flows. For this investigation, the k-ω Shear Stress Transport (SST) mo–del [35] was selected due to its ability to handle boun–dary layer interactions and free shear flows. This hybrid approach applies the *k-ω* formulation within boundary layers and transitions to the *k-*□ formulation in regions outside solid boundaries, ensuring robust performance across diverse flow conditions. The mathematical formulation of the turbulence model is provided below.

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left(\Gamma_k \frac{\partial k}{\partial x_j}\right)+\tilde{G}_k-Y_k+S_k \quad (1)$$

$$\frac{\partial}{\partial t}(\rho \omega)+\frac{\partial}{\partial x_i}(\rho \omega u_i)=\frac{\partial}{\partial x_j}\left(\Gamma_\omega \frac{\partial \omega}{\partial x_j}\right)+G_\omega-Y_\omega+D_\omega+S_\omega \quad (2)$$

Compressibility contributes to dissipation terms Y_k and Y_ω and plays a role in the generation of turbulent kinetic energy outlined below [35]:

$$Y_k=\rho\beta^* k\omega \quad (3)$$

$$Y_\omega=\rho\beta\omega^2 \quad (4)$$

Herein, β^* and β are functions of $F(M^t)$.

$$\beta^*=\beta_i^*\left[1+\zeta^* F\left(M_t\right)\right] \quad (5)$$

$$\beta=\beta_i\left[1-\frac{\beta_i^*}{\beta_i}\zeta^* F\left(M_t\right)\right] \quad (6)$$

where $\zeta^* = 1.5$.
Moreover, the representation of compressibility function $F(M_t)$ *is denoted as*

$$F\left(M_t\right)=\begin{cases}0 & M_t \le M_{t0}\\ M_t^2-M_{t0}^2 & M_t > M_{t0}\end{cases} \quad (7)$$

$$M_t^2=\frac{2k}{a^2} \quad (8)$$

where

$$M_{t0}=0.25 \qquad \alpha=\sqrt{\gamma RT} \quad (9)$$

Depending on compressibility, the term $\tilde{G}_k$ indi–cates the generation of turbulent kinetic energy, which is expressed as:

$$\tilde{G}_k=\min\left(G_K, 10\rho\beta^* k\omega\right) \quad (10)$$

In high-speed combustion scenarios, turbulence-chemistry interaction models like the finite rate/eddy dissipation approach are commonly chosen for their efficiency and reliability. This method assumes that reaction rates are significantly faster than the transport processes within the flow field. To further enhance computational efficiency while maintaining predictive accuracy, a hydrogen-oxygen single-step reaction mec–hanism is implemented in this study. This mechanism has proven effective for modeling scramjet combustor performance with minimal computational overhead. The adopted reaction mechanism is detailed below.

$$2H_2+O_2 \rightarrow 2H_2O \quad (11)$$

Table 1. The initial flow conditions for air and hydrogen (H_2) fuel [27].

	Fluid	
	Air	*Hydrogen*
P (pa)	101325	101325
Ma	2	1
T (K)	340	250
Y_{N_2}	0.736	0
Y_{O_2}	0.232	0
Y_{H_2O}	0.032	0
Y_{H_2}	0	1

At the combustor inlet, the flow remains supersonic. Boundary conditions are defined by specifying the static pressure, temperature, and mass fractions of individual species [Table 1]. At the combustor outlet, the flow also exits in a supersonic state, with all flow variables extrapolated from the computational domain's interior. The upper and lower walls of the combustor are modeled with no-slip and adiabatic conditions to reflect realistic thermal and viscous interactions.

4. VALIDATION OF THE MODEL

To validate the proposed numerical methodology, the simulation results for the strut injector were compared with experimental data obtained by Waidmann et al. at the German Aerospace Center (DLR) [27]. This experimental dataset served as a reliable benchmark for advancing CFD models tailored for strut-based injectors. Fig. 3 (a) presents a comparison between the numerical density contours for non-reacting flow and the experimental Schlieren images [27]. The analysis reveals distinct shock wave structures at the trailing edge of the strut, including: (1) an oblique shock wave initiated by the strut's leading edge that reflects off the combustor wall to form reflected shock wave 1, (2) expansion wave 2 generated at the trailing edge of the strut as the flow expands, and (3) a collision shock wave 3 formed where the upper and lower streams converge after being deflected towards the flow field center. These shock waves propagate downstream, influencing the boundary layer along the combustor walls. Compression shock wave 4 arises as the flow exiting the separation zone undergoes compression.

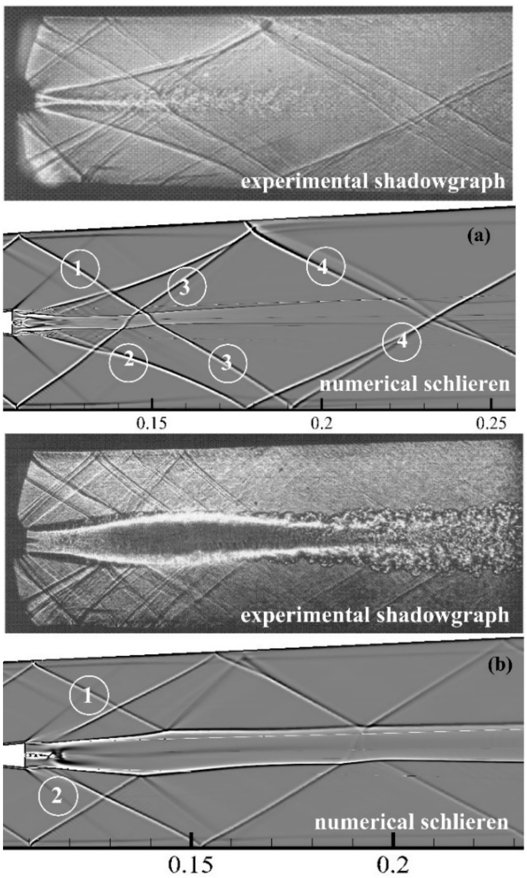


Figure 3. Experimental schlieren [27] and computational schlieren for (a) non-reacting (b) reacting case

Fig. 3(b) further highlights the comparison between experimental shadowgraphs and numerical density contours for reacting flow. The results reveal that the combustor's upper wall, which expands at a 3° angle, alters the interaction dynamics between reflected shock wave 1 and the hydrogen jet. The abrupt channel expansion at the strut's trailing edge leads to the formation of expansion wave 2, which weakens reflected shock wave 1 through successive interactions with expansion wave 2, recirculation zone shear layers, and the supersonic flow. The numerical study effectively captures variations in the combustion region's width and shape at different axial locations, demonstrating good agreement with experimental observations [27].

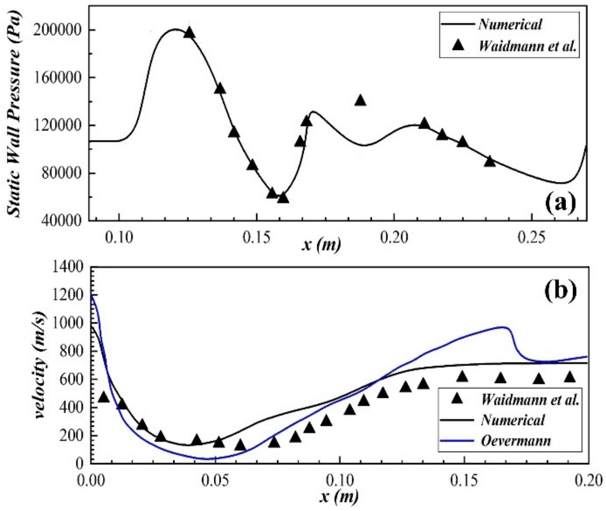


Figure 4 Experimental and computational (a) pressure (non-reacting) and (b)velocity distribution (reacting) at the lower wall

Figure 4(a) illustrates the static pressure distribution for the non-reacting case along the lower wall of the combustor. The changes in pressure and velocity downstream of the injector are influenced by the interaction between the leading shock wave, which is formed when the supersonic airflow hits the sharp edge of the strut, and the wall boundary layer. This interaction becomes stronger along the wall and leads to pressure and velocity variations further downstream. Figure 4(b) presents the axial velocity profile at approximately y = 25 mm for the reacting case. It shows an initial drop in velocity, followed by an increase near $x \approx 0.1$ m, and then a decrease around $x \approx 0.18$ m due to the formation of a weak shock wave.

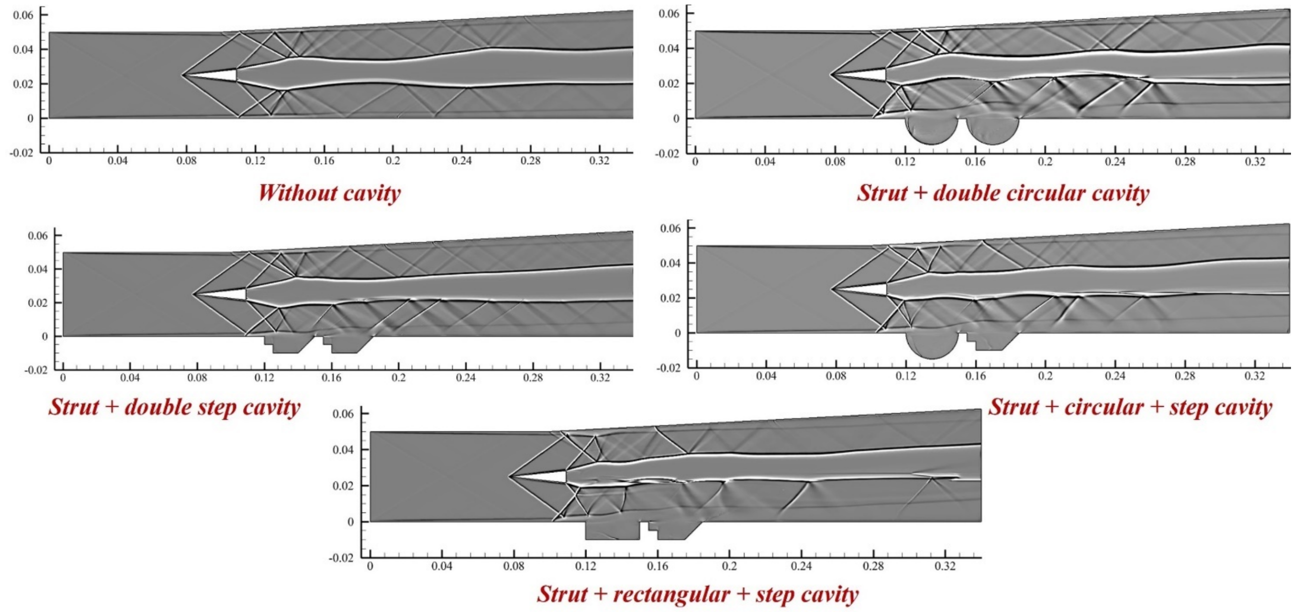
Overall, the numerical simulations exhibit a strong correlation with experimental findings [27], effectively validating the computational framework and its capability to predict key flow phenomena in scramjet combustors.

5. RESULTS AND DISCUSSION

The recirculation zone in supersonic scramjet combustors plays a pivotal role in stabilizing the flame and enhancing the mixing and combustion of air and fuel. In a supersonic flow regime, where the airflow through the combustion chamber has an exceptionally short residence time on the order of 1 ms, the challenges of efficient air-fuel mixing, atomization, vaporization, ignition, and combustion are significant. Inefficiencies in these processes often lead to the wastage of unburned fuel, flushed out by the high-speed airstream. A potential solution lies in the strategic integration of flame-holder cavities, which induce recirculation zones to extend the effective residence time for combustion processes. In this study, four innovative double-cavity configurations such as a double circular cavity, double step cavity, circular and step cavity, and rectangular and step cavity were numerically simulated to investigate their effects on mixing enhancement and flow dynamics.

The schlieren contours obtained from numerical simulations [Fig. 5] of different cavity configurations in a supersonic scramjet combustor reveal the intricate interactions of shock waves and their influence on recirculation zones, critical for air-fuel mixing. In the baseline configuration without cavities, oblique shock waves form at the leading edge of the strut, reflecting multiple times between the combustion chamber walls and fuel stream, creating a shock train with limited mixing enhancement, while expansion shocks at the trailing edge dissipate with minimal contribution to turbulence. The introduction of cavities significantly alters the flow dynamics; in the strut + circular and step cavity configuration, the smoother circular geometry generates weak shock waves and minimal flow separation, resulting in limited flow disturbances and a clean shock structure forming approximately 0.12 m downstream. In contrast, the strut + rectangular and step cavity produces sharp and strong oblique shocks due to abrupt flow separation at the rectangular edges, with shock reflections initiating at 0.14 m downstream and leading to pronounced flow disturbances and larger recirculation zones. The strut + double circular cavity configuration demonstrates complex shock interactions, with overlapping circular cavities creating chaotic flow patterns and higher levels of turbulence, while shocks form farther downstream at around 0.16 m, amplifying mixing but introducing significant distortion. The strut + double step cavity exhibits intense and controlled oblique shocks forming at 0.13 m downstream, with each step generating localized recirculation zones that enhance residence time and promote effective mixing through controlled turbulence. Across all cavity-based configurations, the recirculation zones and vortices formed by shock wave interactions are crucial in entraining fuel into the supersonic air stream, highlighting the role of cavity geometry in optimizing combustor performance.

The Mach number contours [Fig. 6] for various cavity configurations in the scramjet combustor illustrate distinct flow acceleration, deceleration, and shock interaction patterns. In the baseline case without cavities, the Mach number distribution is relatively uniform with oblique shocks originating from the leading edge of the strut, followed by a gradual reduction in Mach number near the trailing edge due to weak shock interactions. In the double-step cavity, the contours reveal a significant reduction in Mach number within the recirculation zones created at each step, demonstrating effective flow deceleration and enhanced residence time. For the double circular cavity, overlapping cavities generate intricate low-Mach regions downstream of the circular shapes, highlighting stronger interactions with the supersonic flow but with relatively smooth transitions in the cavity flow structure. The circular + step cavity displays noticeable boundary layer (B.L.) separation just downstream of the step, where the Mach number sharply decreases, leading to localized flow stagnation that promotes enhanced mixing but also increases drag. A similar boundary layer separation is observed in the rectangular + step cavity, where the sharp edges and abrupt geometric transitions cause flow detachment, resulting in distinct low-Mach regions and turbulent recirculation zones. This configuration also shows the strongest Mach number gradients, indicating sharper shock interactions and higher flow deceleration compared to other configurations.



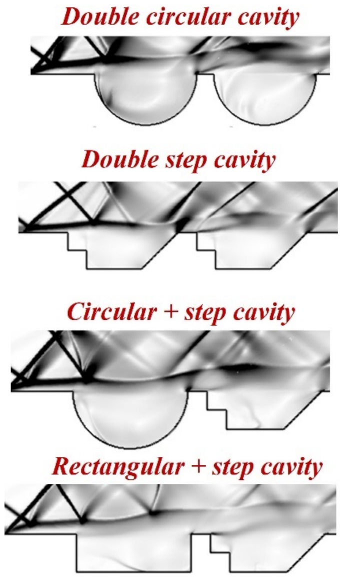


Figure 5. Numerical Schlieren for different flame holder combinations: without cavity and with strut + dual cavity

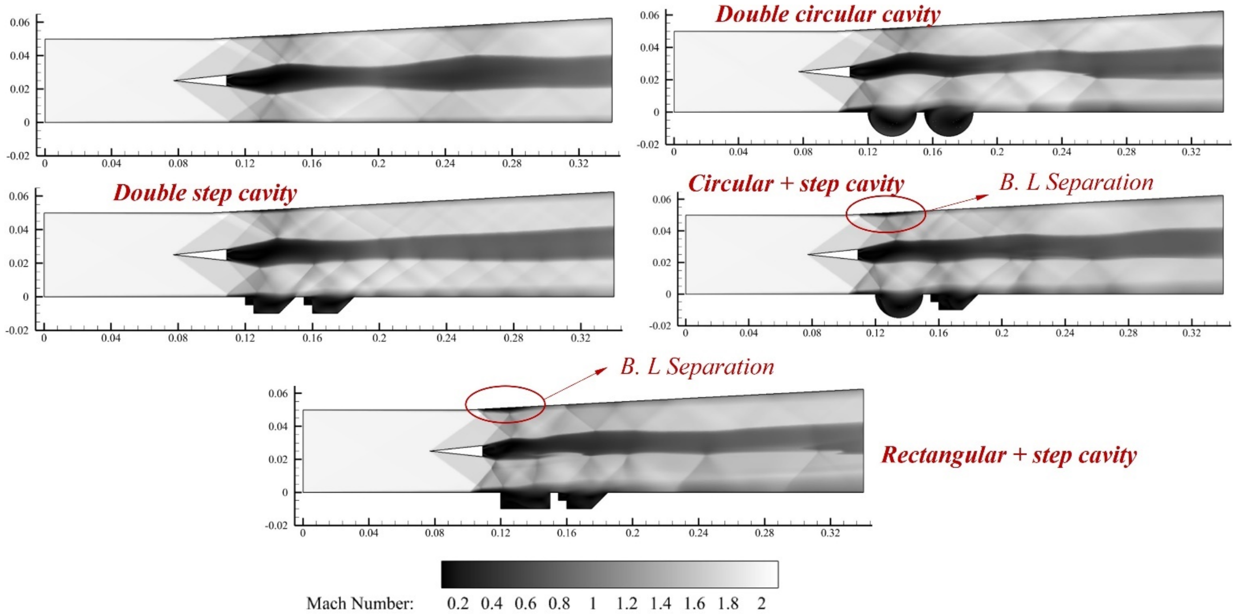


Figure 6. Computed Mach no contour for different flame holder combinations: without cavity and with strut + dual cavity

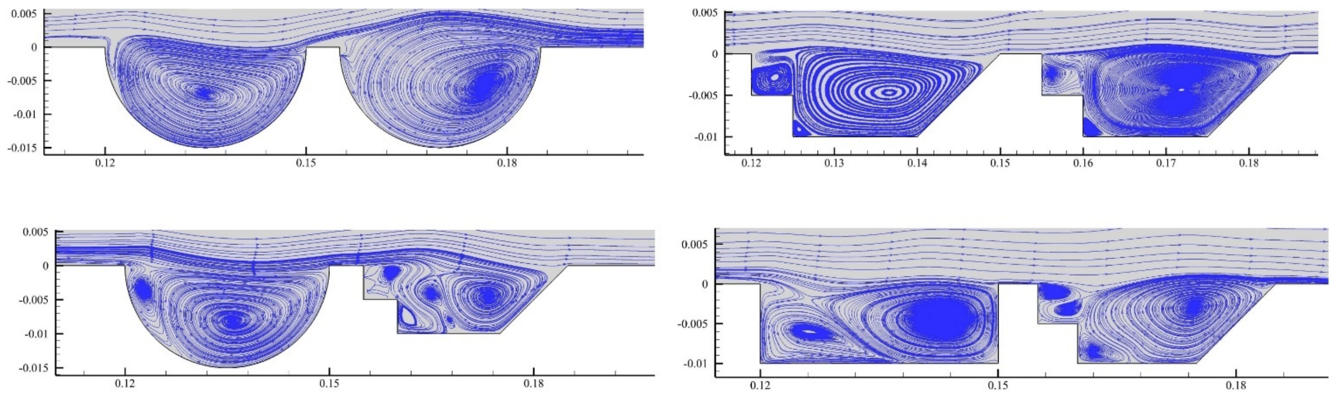


Figure 7. Computed velocity streamlines for different flame holder combination

The streamlined contours [Fig. 7] for different cavity configurations provide insights into the flow recir–culation patterns and shear layer interactions. In the double-step cavity, the presence of two sequential steps creates distinct and well-defined recirculation zones with strong vortices between the steps. These coherent vortices enhance turbulence and residence time, which is critical for effective mixing. The double circular cavity, by contrast, shows more diffused and over–lapping recirculation zones with weaker vortex strength compared to the step cavity, indicating lower shear layer interaction and reduced mixing intensity. For the circular + step cavity, the streamline patterns highlight asymmetric vortex formation downstream of the circular cavity, compounded by boundary layer separation near the step. This leads to irregular flow reattachment, compromising mixing uniformity. The rectangular + step cavity, although showing a more structured vortex formation compared to the circular configurations, exhibits weaker recirculation strength than the double step cavity, with less effective vortex confinement near the step edges. These differences suggest that the geometric configuration of the cavity significantly influences vortex formation and flow stabilization, with the double-step cavity exhibiting the most robust and beneficial flow recirculation.

The temperature contours [Fig.8] illustrate the effectiveness of various cavity configurations in stabi–lizing combustion and generating heat in scramjet com–bustors. In the double-step cavity, combustion is well-stabilized through cavity-stabilized mechanisms, with controlled heat generation visible immediately down–stream of the cavity. The temperature rises efficiently, with combustion starting earlier and intensifying around 0.1 m from the inlet, achieving temperatures exceeding 2200 K within a short length, which indicates optimal thermal energy release. In contrast, the double circular cavity generates intense heat closer to the cavity, with higher temperatures forming quickly, but this configu–ration exhibits less control over the heat release rate, pot–entially leading to hotspots and non-uniform combustion.

For the circular + step cavity, combustion initiation is slightly delayed, occurring prominently beyond 0.12 m, with notable boundary layer separation impacting tempe–rature distribution. The rise in temperature is less uni–form, and maximum heat generation is localized near the step. The rectangular + step cavity shows an inter–medi–ate behavior, achieving higher temperatures over a longer distance, though not as efficiently as the double step cavity. Here, combustion begins slightly earlier than in the circular configurations but with lower intensity in the initial region. The absence of cavities in the standard DLR scramjet results in poor jet-wake stabilized combustion, demonstrating that cavity-induced recircu–lation zones play a critical role in anchoring the flame and ensuring efficient mixing. Among these, the double-step cavity provides the most efficient heat generation and fla–me stabilization by ensuring uniform and controlled tem–perature rise, critical for enhanced scramjet performance.

The combustion efficiency and total pressure loss are critical parameters for evaluating the performance of a scramjet combustor. Combustion is a rapid exothermic chemical reaction that occurs under high turbulence conditions, resulting in the release of significant thermal energy. The equation used to evaluate both combustion efficiency and total pressure loss is well-established and has been comprehensively analyzed in previous research [14, 29, 30].

The combustion efficiency graph [Fig. 9(a)] illus–trates the performance of different strut + dual cavity configurations in stabilizing combustion within the scramjet combustor. The double-step cavity clearly outperforms all other configurations, reaching a com–bustion efficiency of approximately 80% within a shor–ter axial distance (~0.28 m). This superior performance is attributed to its ability to generate strong recirculation zones and enhance turbulent mixing through successive shear layer interactions. The double circular cavity, while showing significant improvement over configu–rations with fewer or no steps, achieves lower combus–tion efficiency (~75% at 0.30 m) compared to the double-step cavity. This is due to the less effective interaction between recirculation zones and the limited flow reattachment capabilities, leading to delayed mixing. The circular + step cavity and rectangular + step cavity configurations perform moderately, with efficiencies reaching 65-70% over similar distances. These designs, though capable of creating shear layers and vortices, lack the sequential flow reattachment mechanism provided by the double-step cavity, resulting in less effective mixing and lower combustion stabilization.

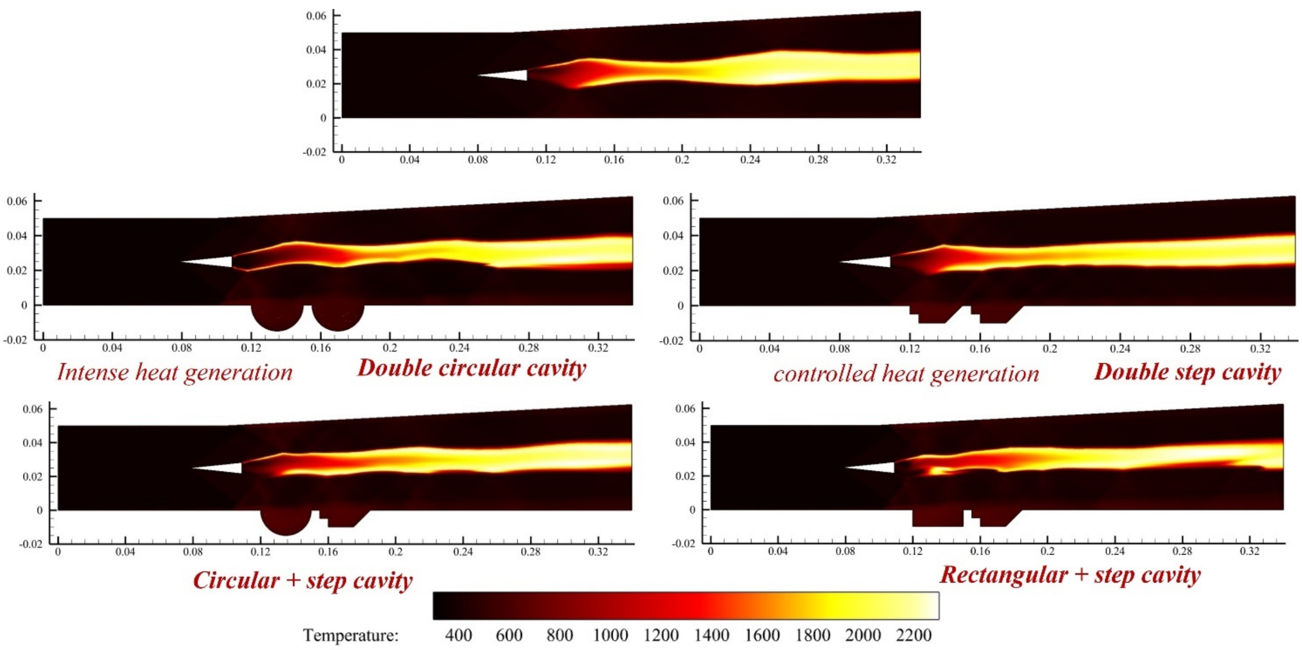


Figure 8. Computed temperature contour for different flame holder combinations

In comparison, the strut without cavity demon–strates the lowest combustion efficiency, struggling to exceed 62% even at the farthest distance measured (~0.30 m). This highlights the importance of cavity-induced recirculation zones and enhanced residence time for achieving effective fuel-air mixing in high-speed flows. In summary, the double-step cavity design ensures optimal combustion efficiency by leveraging enhanced turbulence, multiple recirculation zones, and increased residence time, making it the most effective configuration for high-speed scramjet applications.

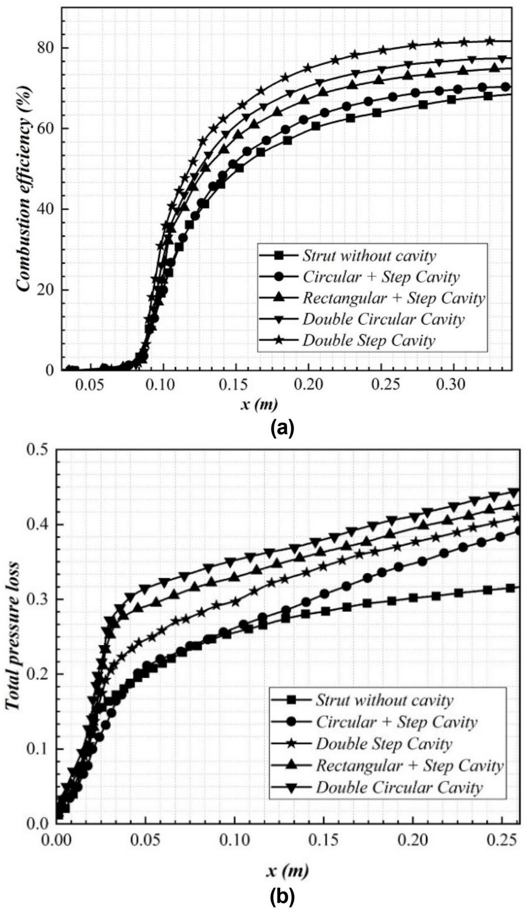


Figure 9 (a) Simulated combustion efficiency (b) total pres–sure loss variation for different flame holder combination

The total pressure loss [Fig. 9(b)] varies signifi–cantly across strut + dual cavity configurations in the scramjet combustor, highlighting the balance between efficient mixing and aerodynamic performance. The double-step cavity achieves a moderate pressure loss (~0.38 at 0.25 m), as its sequential recirculation zones and smoother flow reattachment minimize abrupt energy dissipation while maintaining effective mixing. The double circular cavity, with the highest pressure loss (~0.42 at 0.25 m), generates strong turbulence and overlapping vortices, enhancing mixing but at the cost of greater energy dissipation. Similarly, the rectangular + step cavity incurs a high-pressure loss (~0.40 at 0.25 m) due to sharp edges and abrupt flow separations, which disrupt the flow significantly. In contrast, the circular + step cavity exhibits the lowest pressure loss (~0.33 at 0.25 m) by leveraging smoother contours to reduce turbulence, though it may compromise slightly on mixing efficiency. The baseline strut without cavity shows minimal pressure loss (~0.28 at 0.25 m) but lacks the enhanced mixing benefits of cavity configurations. Among these, the double-step cavity stands out for its balance between moderate pressure loss and superior combustion performance.

6. CONCLUSION

The numerical investigation of a strut-based scramjet combustor equipped with four different cavity confi–gurations as flame holders has been conducted using Reynolds-Averaged Navier-Stokes (RANS) equations, the SST k-ω turbulence model, the finite rate/eddy dissipation chemistry model, and a global one-step reaction mechanism. The study emphasized the critical role of cavity design in influencing the combustor's performance, measured through combustion efficiency and total pressure loss.

Among the configurations analyzed, the double-step cavity demonstrated a balanced performance by achie–ving high combustion efficiency with moderate total pressure loss, making it an optimal choice for scramjet applications. The circular + step cavity, while exhibiting the lowest total pressure loss, showed reduced com–bustion efficiency. In contrast, the double circular cavity delivered better combustion efficiency due to intense mixing but incurred significant pressure losses. The rectangular + step cavity offered intermediate results, with effective mixing but higher pressure loss than the double step cavity.

The results confirmed the effectiveness of passive techniques, specifically cavity-based geometric modifi–cations, in enhancing fuel-air mixing and stabilizing combustion. Validation of the numerical model against experimental data ensured the reliability of the findings. This study underscores the potential of cavity confi–gurations as practical and efficient design strategies for improving scramjet combustor performance. Future research can explore hybrid cavity designs and advan–ced turbulence models to further optimize scramjet technologies.

ACKNOWLEDGMENT

The author extends heartfelt gratitude to the organizers of ICME2024, Serbia, with special acknowledgment to Prof. Ivana Atanasovska and Prof. Tatjana Lazovic, for their gracious invitation and the opportunity to contribute the extended version of our work to FME Transactions.

REFERENCES

[1] Urzay J. Supersonic combustion in air-breathing propulsion systems for hypersonic flight, Annu. Rev. Fluid Mech. Vol. 50, pp. 593–627, 2018

[2] Billig FS. Research on supersonic combustion, J. Propuls Power, Vol 9, No.4, pp. 449-514, 1993

[3] Choubey, G., Tiwari, M., Scramjet Combustion: Fundamentals and Advances, Butterworth-Heine–mann., 2022

[4] Pranaykumar S, Mengu D, Prakash A. Effect of crossflow impinging type throttling mechanism on

combustion characteristics of a strut-based scramjet combustor. Aerospace Science and Technology. Vol.142, p.108648, 2023

[5] Chang J, Zhang J, Bao W, Yu D. Research progress on strut-equipped supersonic combustors for scramjet application, Prog. Aerosp. Sci. Vol. 103, pp. 1–30, 2018

[6] Oevermann M. Numerical investigation of turbulent hydrogen combustion in a SCRAMJET using flamelet modeling, Aero. Sci. Technol. Vol. 4, No. 7, pp. 463–480, 2000

[7] Fureby C, Fedina E, Tegn´er J. A computational study of supersonic combustion behind a wedge-shaped flameholder. Shock Waves Vol.24, pp. 41–50, 2014

[8] G´enin F, Menon S. Simulation of turbulent mixing behind a strut injector in supersonic flow. AIAA J. Vol.48, No.3, pp. 526–539, 2010

[9] Huang W. Investigation on the effect of strut configurations and locations on the combustion performance of a typical scramjet combustor, J. Mech. Sci. Technol. Vol. 29, pp.5485–5496, 2015

[10] Huang W, Wang Z, Luo S, Liu J. Parametric effects on the combustion flow field of a typical strut-based scramjet combustor. Chin. Sci. Bull. Vol. 56, pp.3871–3877, 2011

[11] Pu C, Guo G, Han J, Chen H, Xu P, Zhang C. Influence of flow control schemes of the three-strut injector on the mixing efficiency and total pressure loss for a scramjet combustor, Int. J. Hydrogen Energy Vol.48, No. 94, pp. 36972–36986, 2023

[12] Choubey, G., Pandey, K.M. Effect of variation of angle of attack on the performance of two-strut scramjet combustor, Int J Hydrogen Energy., Vol. 41, No. 26, pp. 11455-11470, 2016

[13] Choubey G, Pandey KM. Investigation on the effects of operating variables on the performance of two-strut scramjet combustor. Int J Hydrogen Energy, Vol.41, No.45, pp.20753-70, 2016

[14] Choubey, G., Pandey, K.M., Effect of parametric variation of strut layout and position on the performance of a typical two-strut based scramjet combustor, Int J Hydrogen Energy., Vol. 42, No. 15, pp. 10485-10500, 2017.

[15] Choubey, G., Pandey, K.M., Effect of different wall injection schemes on the flow-field of hydrogen fuelled strut-based scramjet combustor, Acta Astronautica., Vol. 145, pp. 93–104, 2018.

[16] Choubey, G., Devarajan, Y., Huang, W., Mehar, K., Tiwari, M., Pandey, K.M., Recent advances in cavity-based scramjet engine a brief review, Int J Hydrogen Energy., Vol. 44, No. 26, pp. 13895–13909, 2019

[17] Zong Y, Bao W, Chang J, Hu J, Yang Q, Song J, Wu M. Effect of fuel injection allocation on the combustion characteristics of a cavity-strut model scramjet. Journal of Aerospace Engineering. 2015 Vol.28, No.1, p. 04014050, 2015

[18] Hsu K Y, Carter C D, Gruber M R, Barhorst T and Smith S. Experimental Study of Cavity-Strut Combustion in Supersonic Flow. Journal of Propulsion and Power, Vol. 26, No.6, pp.1237-1246, 2010

[19] Hsu K Y, Campbell D. Carter, Mark R. Gruber and Chung-Jen. Tam. Mixing Study of Strut Injectors in Supersonic Flows. AIAA 2009-5226, 45th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, Denver, Colorado, August 2-5, 2009.

[20] Hongbin G, Lihong C, Dan W, Chang Xin yu, Li Zhi, Xilong Y. Experimental Investigation on Coupling Characteristics of Cavity Flameholder and Strut Jet. AIAA 2012-5961, 18th AIAA/3AF International Space Planes and Hypersonic Systems and Technologies Conference, Tours, France, September 24-28, 2012.

[21] Nathan R. Grady, Robert W. Pitz, Campbell D. Carter, Kuang-Yu Hsu, Chaitanya Ghodke and Suresh Menon. Supersonic Flow over a Ramped-Wall Cavity Flame Holder with an Upstream Strut. Journal of Propulsion and Power, Vol.28, No.5, pp. 982-989, 2012

[22] Zmijanović, V., Rašuo, B., Chpoun, A.: Flow separation modes and side phenomena in an overexpanded nozzle, FME Transactions, Vol. 40, No. 3, pp.111-118, 2012

[23] Marković, M., Rašuo, B., Milinović, M., Amine, BM.: Engagement areas of missiles in the proportional navigated flight powered by air breathing engines, FME Transactions, Vol. 42, No. 3, pp.181-188, 2014

[24] Jeremić, O., Milinović, M., Marković, M., Rašuo, B.: Analytical and Numerical Method of Velocity Fields for the Explosively Formed Projectiles, FME Transactions, Vol. 45, No. 1, pp. 38-44, 2017

[25] Ocokoljić, G., Rašuo, B., Damljanović, D., Živković, S.: Experimental and Numerical Research of the Influence of Thrust Vector Control on the Missile Aerodynamics by Cold and Hot Jet Simulations, FME Transactions, Vol. 48, No. 4, pp. 770-778, 2020

[26] Damljanović, D., Rašuo, B.: Testing of calibration models in order to certify the overall reliability of the trisonic blowdown wind tunnel of VTI, FME Transactions, Vol. 38, No.4, pp. 167-172, 2010

[27] Waidmann W, Alff F, Bohm M, Brummund U, Clauss W, Oschwald M. Supersonic combustion of hydrogen/air in a scramjet combustion chamber, Space Technol, Vol.15, No. 6, pp. 421-9, 1995 https://doi.org/10.1016/0892-9270(95) 00017-8

[28] ANSYS, Inc. ANSYS Fluent User's Guide, Release 17.2, 2016.

[29] Choubey G, Solanki M, Patel O, Devarajan Y, Huang W, Effect of different strut design on the mixing performance of H2 fueled two-strut based scramjet combustor, Fuel, Vol 351, pp.128972, 2023

[30] Choubey G, Patel O, Solanki M, Ingenito A, Devarajan Y, Tripathi S. Influence of cavity floor injection strategy on mixing improvement study of a splitter plate-assisted supersonic combustor. Engineering Analysis with Boundary Elements, Vol 155, pp.995-1012, 2023

[31] Wang SY, Du ZB, Huang W, Choubey G. Numerical study on a novel device for hydrogen mixing enhancement in a scramjet engine: Coaxial injector. Aerospace Science and Technology, Vol.127, p.107680, 2022

[32] Choubey G, Solanki M, Bhatt T, Kshitij G, Yuvarajan D, Huang W. Numerical investigation on a typical scramjet combustor using cavity floor H2 fuel injection strategy. Acta Astronautica. Vol. 202, pp.373-85, 2023

[33] Huang W, Wu H, Du ZB, Yan L, Choubey G. Design exploration on the mixing augmentation induced by the oblique shock wave and a novel step in a supersonic flow. Acta Astronautica. Vol.180, pp. 622-9, 2021

[34] Liu X, Moradi R, Manh TD, Choubey G, Li Z, Bach QV. Computational study of the multi hydrogen jets in presence of the upstream step in a Ma= 4 supersonic flow. International Journal of Hydrogen Energy, Vol. 45, No.55, pp.31118-29, 2020

[35] Menter, F.R., Two-equation eddy-viscosity turbulence models for engineering applications. AIAA journal., Vol. 32, No. 8, pp.1598-1605, 1994

NOMENCLATURE

Ma	Mach Number
k	turbulent kinetic energy [m^2/s^2]
ω	specific turbulent dissipation rate [1/s]
S_k , S_ω	user-defined terms
Γ_k	effective diffusion term for k
Γ_ω	effective diffusion term for ω
D_ω	cross-diffusion term
T	temperature [K]
P	pressure [Pa]

Abbreviations

CFD	Computational Fluid Dynamics
RANS	Reynolds-Averaged Navier-Stokes
LES	Large eddy simulation
DLR	Deutsches Zentrum für Luft- und Raumfahrt
AUSM	Advection Upstream Splitting Method
SST	Shear Stress Transport
B.L	Boundary Layer

НУМЕРИЧКО ИСПИТИВАЊЕ ДРЖАЧА ПЛАМЕНА СА ДВОСТРУКОМ ШУПЉИНОМ ЗА ПОБОЉШАНЕ ПЕРФОРМАНСЕ САГОРЕВАЊА У СКРЕМЏЕТ МОТОРИМА

Г. Шоуби, Г. Брар, М. Каушик

Скремџет мотори имају значајно обећање за хиперсонични погон, али сложеност надзвучног сагоревања представља значајне изазове. Рачунарска динамика флуида (ЦФД) нуди исплатив приступ проучавању турбулентних реактивних токова у таквим системима, посебно се бави критичним интеракцијама турбуленције и хемије које се разликују од конвенционалног сагоревања. Ова студија користи дводимензионалне Рејнолдс-ове просечне Навиер–Стоукс (РАНС) симулације за процену новог држача пламена са двоструком шуп–љином у сцрамјет ложиштима. Међу различитим анализираним конфигурацијама, дуплостепена шуп–љина је показала супериорну равнотежу, постижући високу ефикасност сагоревања са умереним укуп–ним губитком притиска. Нумерички модел је вали–диран у односу на експерименталне податке, чиме је потврђена његова поузданост и тачност предвиђања. Налази истичу ефикасност интеграције геометрија заснованих на шупљини са ињекторима са опругом у побољшању мешања горива и ваздуха и стаби–лизацији сагоревања. Ово истраживање наглашава потенцијал држача пламена са двоструком шуп–љином као практичног решења за унапређење перформанси сцрамјет у хиперсоничним применама.