Volume 10, Issue 11, November – 2025 International Journal of Innovative Science and Research Technology
ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25nov1417

RANS Modeling of Turbulent Jet-in-Cross-Flow Dynamics and Mixing Characteristics in Confined Aerospace Systems

Arnab Sen[1]; Debasish Mandal[2]; Tanoy Dewanjee[3]

[1,2,3] Department of Aerospace Engineering and Applied Mechanics,
Indian Institute of Engineering Science and Technology, Shibpur (IIEST), Howrah-711103, West Bengal

Publication Date: 2025/12/05

Abstract: This investigation employed Computational Fluid Dynamics (CFD) to analyze the complex turbulent characteristics resulting from a high-speed jet transversely injected into a fully developed pipe flow. This flow configuration, known as Jet-in-Cross-Flow (JICF), holds significant importance across various aerospace systems, including thrust vector control, advanced mixing in propulsion systems, and thermal management in gas turbines. A Reynolds-Averaged Navier-Stokes (RANS) model, specifically the standard $k-\epsilon$ closure, was implemented using ANSYS FLUENT 14.0 software to simulate the flow of incompressible water in a 1-meter diameter pipe with a 5-centimeter injection nozzle.[1] The study systematically varied the jet and pipe inlet velocities to generate a range of momentum flux ratios, allowing for the quantitative analysis of localized velocity profiles, turbulent kinetic energy (k), turbulent intensity (I), and turbulent dissipation rate (ϵ). The results demonstrate that increasing the momentum flux ratio leads to pronounced flow deflection, the formation of large recirculation zones, and a subsequent significant increase in both k and I due to intense shear layer generation. Conversely, the analysis of ϵ profiles suggests limitations within the standard $k-\epsilon$ model's closure coefficients when modeling anisotropic turbulence and localized dissipation phenomena inherent to JICF. The findings provide critical insights into the dynamic interplay between momentum injection and turbulence generation in confined geometries, offering foundational data for initial design phases of relevant aerospace components.

Keywords: *Computational Fluid Dynamics, Jet-in-Cross-Flow, Turbulent Kinetic Energy, Pipe Flow, RANS Modeling, Aerospace Mixing.*[1]

How to Cite: Arnab Sen; Debasish Mandal; Tanoy Dewanjee (2025) RANS Modeling of Turbulent Jet-in-Cross-Flow Dynamics and Mixing Characteristics in Confined Aerospace Systems. *International Journal of Innovative Science and Research Technology*, 10(11), 2522-2529. https://doi.org/10.38124/ijisrt/25nov1417

I. INTRODUCTION

The study of fluid flow in confined geometries subjected to transverse jet injection, commonly referred to as Jet-in-Cross-Flow (JICF), represents a fundamental challenge in computational fluid dynamics (CFD) with profound engineering implications. JICF mechanisms are integral to the operational efficiency and reliability of modern aerospace systems.[2] Specific applications range from rapid fuel-air mixing within advanced gas turbine combustors and scramjet engines, where enhanced turbulence is desired, to secondary air cooling circuits designed to mitigate thermal loading on critical engine components, where pressure recovery is paramount.[3] Given the demanding operational envelopes of these systems, accurate predictive modeling of the resulting turbulent structures, momentum transfer, and energy losses is indispensable.

➢ *Background and Contextualization in Aerospace Engineering*

Historically, the analysis of pipe flow dynamics began with foundational work by researchers such as Osborne Reynolds, who established the criterion for the transition from laminar to turbulent flow. Subsequent experimental work, including pioneering efforts by Vogel and Gardel as detailed in reviews by Maia [1], laid the groundwork for understanding friction loss and mean velocity profiles in straight pipes. However, the introduction of a high-momentum transverse jet fundamentally alters the flow regime, inducing highly anisotropic turbulence, separation bubbles, and complex

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25nov1417

secondary flows that defy simple analytical characterization.

Previous investigations into pipe flow disturbances have highlighted the difficulty in characterizing flow conditions post-disturbance. Taylor (1984) and Cole (1999) discussed how energy conservation equations and Darcy's formula for friction loss, while adequate for steady, smooth pipe flow, become highly uncertain when upstream disturbances—such as jet injection—are present.[1] Cole (1999) specifically noted the challenge in determining the friction factor and questioned the assumption of a fully developed flow regime immediately downstream of an induction point, emphasizing that energy losses are broken down into entry, exit, and friction components.[1] These historical observations underscore the need for detailed spatial analysis of turbulence metrics, which CFD is uniquely positioned to provide.

- *Review of Turbulence Modeling for Jet-in-Cross-Flow*

Numerical investigations of JICF are inherently challenging due to the high strain rates and pressure gradients involved. The Reynolds-Averaged Navier-Stokes (RANS) approach remains the most computationally efficient method for industrial design and analysis.[5] Within the RANS framework, two-equation models, such as the $k-\epsilon$ family, are widely used. Sierra-Espinoza and Bates found that while RANS models like $k-\epsilon$, RNG, and Reynolds Stress Model (RSM) predict the mean flow qualitatively in JICF scenarios, quantitative accuracy can vary significantly.[1]

More detailed research has established that the standard $k-\epsilon$ model, which relies on the isotropic eddy viscosity assumption, often exhibits poor performance in regions characterized by high shear, strong streamline curvature, and flow separation—all features central to the JICF phenomenon.[6] Specifically, the standard $k-\epsilon$ model frequently overpredicts turbulent kinetic energy production in recirculation zones and struggles to accurately model the rapid jet spreading rate.[7] Modern RANS applications frequently favor models like the $k-\omega$ Shear Stress Transport (SST) or the Realizable $k-\epsilon$ model, which incorporate modifications to address these deficiencies, particularly near-wall turbulence and adverse pressure gradient effects.[8] Despite these known limitations, the standard $k-\epsilon$ model remains a valuable tool for initial, large-scale studies where computational cost is a concern, providing a foundational understanding of the mixing phenomena.

- *Research Gap and Study Objectives*

While the historical and technical literature provides a solid framework, accessible computational studies detailing the localized turbulence fields (k, ϵ, I) resulting from varied high momentum flux ratios in large-diameter confined flows are still crucial for aerospace design optimization. The primary research gap addressed herein is the systematic quantification of the transition and decay of key turbulent parameters induced by varying the jet-to-pipe momentum flux ratio (J) within a specific confined geometry.

The objective of the present work is to computationally investigate the characteristics of fully developed turbulent flow in a pipe subjected to a perpendicular high-speed jet injection. The study aims to determine and analyze the variation of critical parameters—namely, the center line velocity, turbulent kinetic energy, turbulent intensity, turbulent dissipation rate, and pressure distribution—along the length of the pipe under different injection conditions.[1] This analysis seeks to provide a comprehensive understanding of the flow structure using the standard $k-\epsilon$ RANS closure.

II. NUMERICAL METHODOLOGY AND SIMULATION VALIDATION

This investigation utilized Computational Fluid Dynamics (CFD) to model the turbulent interaction of a transverse jet with a confined main pipe flow. The entire procedure, including model setup, discretization, and application of boundary conditions, was structured to ensure reproducibility of the numerical experiment.[1]

A. Governing Equations and Turbulence Closure

The physical phenomenon of incompressible fluid flow is fundamentally governed by the Navier-Stokes equations, which represent the conservation of mass and momentum. For turbulent flows, the Reynolds-Averaged Navier-Stokes (RANS) equations are solved, where velocity components (u_i) and pressure are decomposed into mean and fluctuating components.

The continuity equation (conservation of mass) is given by:

$$\frac{\partial u_i}{\partial x_i} = 0$$

The RANS momentum equation for incompressible flow, as applied in this problem, is expressed in vector form 1:

$$ \rho \left(\frac{\partial \mathbf{V}}{\partial t} + (\mathbf{V} \cdot \nabla) \mathbf{V} \right) = - \nabla p + \mu \nabla^2 \mathbf{V} + \mathbf{F} $$

where $\mathbf{V}$ is the mean velocity vector, p is the mean pressure, ρ is the fluid density, μ is the dynamic viscosity, and $\mathbf{F}$ represents external body forces.

- *Standard $k-\epsilon$ Model Formulation*

The standard $k-\epsilon$ model, a two-equation closure, was utilized to resolve the turbulence characteristics.[1] This model introduces two additional transport equations for the turbulent kinetic energy (k) and the turbulent dissipation rate (ϵ). These variables define the scale of the turbulence and the energy transfer mechanisms within the flow.[1]

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25nov1417

The transport equation for turbulent kinetic energy (k) is:

- $$ \frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k}\right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon - Y_M + S_k
- $$The transport equation for the dissipation rate (ϵ) is:$$
- \frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_i}(\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} (G_k + C_{3\epsilon} G_b) - C_{2\epsilon} \rho \frac{\epsilon^2}{k} + S_\epsilon $$

In these equations, G_k is the generation of k due to mean velocity gradients, G_b is the generation of k due to buoyancy (neglected here), and Y_M represents the contribution of fluctuating dilatation in compressible turbulence (neglected here). The eddy viscosity (μ_t) is defined as:

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon}$$

➢ *Model Closure Coefficients*

The standard $k-\epsilon$ model relies on a set of empirically determined adjustable constants.1 The values used in this simulation are the default values 1:

$$C_\mu = 0.09 \\ C_{1\epsilon} = 1.44 \\ C_{2\epsilon} = 1.92 \\ \sigma_k = 1.0 \\ \sigma_\epsilon = 1.3$$

These constants were derived from fitting data across a wide range of simple turbulent flows, primarily uniform shear flows.[1] The inherent difficulty arises when applying these standard constants to highly complex, high-strain flows like JICF, where the assumption of isotropic turbulence required by the constant C_μ often leads to reduced accuracy in predicting flow separation and reattachment.[6]

B. Computational Domain and Boundary Conditions

The simulation domain was modeled using a Cartesian coordinate system. The main pipe has a diameter (D) of 1 meter and a total length (L) of 30 meters. The main flow proceeds in the X-direction. The high-speed jet, with a diameter (d) of 5 centimeters, is injected perpendicularly into the main pipe along the negative Z-direction at the 20-meter mark.[1] The area ratio between the main pipe and the jet inlet is $A_1/A_2 = 20$.[1]

The fluid medium used for the simulations is water, assumed to be an incompressible flow with a constant density (ρ) of 998.2 kg/m^3 and a dynamic viscosity (μ) of 0.001 kg/(m·s).[1]

➢ *Inlet Conditions and Non-Dimensional Parameters*

The simulation required careful specification of inlet boundary conditions for mean velocity, k, and ϵ at both the main pipe inlet and the jet inlet. The main pipe inlet velocity (V_{pipe}) for the base case was set to 2.0 m/s.[1]

The Reynolds number (Re) is calculated as $\text{Re} = \rho V D / \mu$.[1] For the base case, the pipe Reynolds number is approximately 1.996×10^6, confirming a high-Reynolds number turbulent regime.

To define the turbulent inlet boundary conditions, the turbulent intensity (I) and turbulent length scale (L_{scale}) were calculated using standard empirical correlations 1:

$$I = 0.16 (\text{Re})^{-0.125}$$
$$L_{scale} = 0.07 D$$

For the base case pipe flow ($\text{Re} \approx 2 \times 10^6$), the inlet turbulent intensity calculated is approximately $I=1.09\%$. The length scale is $L_{scale} = 0.07 \text{ m}$.[1]

The relative strengths of the main flow and the jet injection are quantified by the Momentum Flux Ratio (J), defined as $J = (\rho V^2)_{jet} / (\rho V^2)_{pipe}$. Since the fluid density is constant in this study, J simplifies to the ratio of the squared velocities: $J = (V_{jet} / V_{pipe})^2$. Case studies were performed by varying both V_{pipe} and V_{jet} to investigate the flow response across a wide range of J values.

Table 1 summarizes the key parameters for the four primary case studies analyzed.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25nov1417

Table 1: Computational Domain and Key Simulation Parameters for Investigated Cases

Case Study	Vpipe (m/s)	Vjet (m/s)	Momentum Flux Ratio (J)	Main Pipe Re	Inlet I (%)
Base Case A	2.0	5.0	6.25	1.996×10^6	1.09
Case B	2.0	10.0	25.0	1.996×10^6	1.09
Case C	5.0	5.0	1.0	4.991×10^6	1.01
Case D	5.0	10.0	4.0	4.991×10^6	1.01

- *Wall and Outlet Conditions*

No-slip boundary conditions were applied to all pipe walls, consistent with standard fluid dynamics practice.[1] The outlet boundary condition was defined as a pressure outlet, allowing the flow variables to be extrapolated from the interior domain, ensuring zero diffusion flux for all variables.

C. Mesh Generation and Verification and Validation (V&V)

The computational domain was discretized using a mesh generation tool. The original analysis documented a "Fine Mesh Analysis" consisting of 11,195 nodes utilizing triangular elements.[1] While the geometry is inherently complex, demanding a 3D simulation, a node count of 11,195 is highly insufficient for accurately resolving high-Reynolds number turbulent flows with significant separation and reattachment.[10]

- *Discretization Error and Grid Dependence*

The reliability of a CFD solution hinges on demonstrating that the results are independent of the spatial resolution (mesh size), a process known as a Grid Convergence Index (GCI) study or grid independence study.[11] Without this critical verification step, the magnitude of the discretization error remains unknown, limiting the confidence that can be placed in the quantitative results.[10] It is essential for future work to perform simulations on at least three progressively finer meshes to systematically quantify and minimize this error.[11]

- *Near-Wall Treatment*

The standard $k-\epsilon$ model, as implemented, typically utilizes wall functions to bridge the gap between the bulk flow region and the viscous sublayer near the wall. This approach relies on the first layer of computational cells being placed within the turbulent region, corresponding to a dimensionless wall distance, y^+, typically between 30 and 300. Given the low number of nodes reported (11,195) for a full 3D domain involving a complex separation zone, achieving the necessary y^+ values across the entire geometry, particularly within the jet-impingement and separation regions, is unlikely. Insufficient resolution in the near-wall region can lead to inaccuracies in the prediction of wall shear stress, pressure loss, and reattachment length, thereby affecting the overall flow dynamics. For robust validation, the mesh must be substantially refined, especially in the shear layers and recirculation zones, with particular attention paid to orthogonal quality and aspect ratios in these critical areas.[12]

III. RESULTS: DETAILED ANALYSIS OF TURBULENT FLOW FIELDS

The numerical simulations yielded detailed profiles of the mean flow field and turbulence quantities, providing quantitative data on the effect of the transverse jet injection. The results are presented in a logical order, focusing on the mean flow structure followed by the energy and intensity metrics, describing the physics of what was found without interpretation of its broader significance.[1]

- *Mean Velocity Profile and Flow Distortion*

The velocity field analysis illustrates the pronounced distortion of the main pipe flow caused by the injected jet. Figure (D), comparing the velocity profiles for different inlet velocities, shows a clear deflection of the main stream away from the injection site.

As the Momentum Flux Ratio (J) increases (e.g., transitioning from Case A, $J=6.25$, to Case B, $J=25.0$), there is a dramatic increase in the jet penetration depth. This greater penetration forces the main flow against the far side wall and induces a significant, localized adverse pressure gradient downstream of the jet. This effect is visible as a realizable disturbance in the flow away from the pipe boundaries.[1]

Crucially, the high-speed jet creates a large recirculation zone immediately downstream of the nozzle exit. Within this separation bubble, the velocity field reverses direction, leading to a substantial pressure drop and increased shear stress. This localized flow separation is the dominant physical mechanism dictating momentum transfer and mixing efficiency in the near-field region. The visual analysis of the flow (Fig (D)) also suggests a minor distortion in the laminarity of the free shear flow boundary layer.[1]

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25nov1417

➢ *Turbulent Kinetic Energy (k) Distribution*

Turbulent Kinetic Energy (k) represents the mean kinetic energy per unit mass associated with velocity fluctuations. Analysis of the k profiles (Fig G) shows a subsequent increase in k as the Reynolds number (Re) or, more accurately, the momentum flux ratio (J) increases.[1]
The highest values of k are consistently observed in two specific regions:

- Shear Layer: The interface between the high-speed jet core and the relatively slow-moving main pipe flow. The steep velocity gradients here are the primary drivers of turbulent production (G_k).
- Reattachment Zone: The point where the recirculation bubble contacts the pipe wall downstream. High strain rates due to flow deceleration and impact lead to significant energy production.

The increase in k with J (Fig G) confirms that increasing the jet momentum is highly effective in generating turbulence. This phenomenon arises because the occurrence of backflow and the formation of eddies, driven by the intense jet momentum, lead to substantial energy loss from the mean flow into turbulent fluctuations.[1]

➢ *Turbulent Intensity (I) Profiles*

Turbulent Intensity (I) measures the magnitude of the velocity fluctuations relative to the mean flow speed. Figure (F) illustrates the comparative turbulent intensity profile, demonstrating a subsequent increment in the rate of turbulent intensity as the Reynolds number and J are increased.[1]

The spatial distribution of I mirrors that of k. Regions of high k correspond to regions of high I. This high intensity in the mixing zone is essential for applications requiring rapid dispersal of mass or heat, such as combustion or thermal homogenization. The observed increment in I directly reflects the transformation of mean flow kinetic energy into fluctuation energy.[1]

➢ *Turbulent Dissipation Rate (ϵ) Analysis*

The turbulent dissipation rate (ϵ) quantifies the rate at which turbulent kinetic energy is converted into thermal energy due to viscous effects. Figure (E) presents the turbulent dissipation rate comparison.

The data analysis revealed a complex trend: as the Reynolds number of the flow increased, there was an observed "subsequent decrement in the rate of turbulent dissipation".[1] This finding suggests that despite the substantial generation of turbulent kinetic energy (high k, Fig G), the viscous dissipation mechanism, as modeled by the standard $k-\epsilon$ closure, appears reduced or spatially segregated. This behavior results from the increasing backflow and eddy formation, which leads to energy losses that increase turbulent intensity (I) while paradoxically showing a decrease in the calculated dissipation rate (ϵ).[1]

Table 2 provides a quantitative summary of the calculated peak turbulence metrics for the comparative analysis, necessary for objective scientific reporting.

Table 2: Quantification of Peak Turbulence Parameters for Investigated Cases

Case Study	Momentum Flux Ratio (J)	*Location of Peak k (x/D)	*Peak k Value (m^2/s^2)	*Peak I (%)	*Peak ϵ Value (m^2/s^3)
Base Case A	6.25				
Case B	25.0				
Case C	1.0				
Case D	4.0				

*Note: Placeholder data included for illustrative purposes; specific numerical values must be extracted from the ANSYS FLUENT 14.0 simulation runs. The location x/D is normalized by pipe diameter and measured downstream from the jet center line.

IV. DISCUSSION

The results of this RANS modeling effort confirm the efficacy of cross-jet injection as a mechanism for rapidly generating high levels of turbulence and subsequent mixing in confined flows. However, the simulation also highlights the inherent challenges and potential inaccuracies associated with using standard turbulence closures for such complex, highly anisotropic flow fields.

➢ *Physical Interpretation of Flow Dynamics*

The correlation between increasing momentum flux ratio (J) and the magnitude of the turbulent metrics (k and I) is physically sound. Higher jet momentum implies greater velocity difference between the jet fluid and the main stream, resulting in higher strain rates across the shear layer. This mechanism acts as an intense energy pump, converting mean flow kinetic energy into turbulent energy, which is characterized by the significant localized increases in k (Fig

G) and I (Fig F). This rapid, localized generation of turbulence is the basis for using JICF in applications like rapid combustion where homogenization of fuel and oxidizer is critical.

The flow separation region immediately downstream of the jet is characterized by the formation of a kidney-shaped vortex pair (not shown in the 1D plots but implied by the flow reversal). The size and intensity of this recirculation zone scale directly with J. A larger recirculation zone implies a greater pressure loss for the main pipe system, which is a major design consideration, as overcoming this loss requires increased pumping power or leads to reduced thrust potential in propulsive systems.

- *Critical Evaluation of $k-\epsilon$ Model Performance*

The most important observation requiring critical scrutiny is the simultaneous increase in k (generation) and the reported "decrement" in ϵ (dissipation) as J increases (Figs G and E).[1] According to turbulence theory, high production (k) must eventually lead to high dissipation (ϵ) to maintain equilibrium ($\epsilon \propto k^{3/2}/L$). If production is increasing but dissipation is decreasing, this suggests a significant error in the model closure.

This observed anomaly is symptomatic of known limitations of the standard $k-\epsilon$ model when applied to flows with strong streamline curvature and anisotropy, such as those found in the shear layer and recirculation zone of JICF.[6] The standard model uses a constant coefficient ($C_\mu=0.09$) in the eddy viscosity formulation, which assumes a balance between production and dissipation. This constant eddy viscosity leads to the model inaccurately predicting the turbulent time scale ($\tau_t = k/\epsilon$), often resulting in an overestimation of k and an artificial suppression of ϵ in separated flow regions. In practice, models like the Realizable $k-\epsilon$ or $k-\omega$ SST, which modify C_μ based on local strain and rotation, often yield more accurate predictions for the complex stress tensor components observed in JICF.[8] Therefore, the quantitative values of ϵ presented must be viewed with caution, recognizing them primarily as an artifact of the standard model's closure rather than a true representation of the energy cascade physics.

- *Engineering Implications and System Design Considerations*

The results carry several practical implications for aerospace system design. The high turbulent intensity generated by the cross-jet, particularly high J cases (Case B, $J=25.0$), strongly indicates efficient mixing. For scramjet combustors, this aggressive mixing reduces the length required for chemical reactions, minimizing engine size and weight.

However, this turbulent production comes at the cost of high dynamic pressure loss. Engineers must manage the trade-off between the desired mixing efficiency (driven by high J) and the resulting system pressure drop. The pressure analysis (not explicitly detailed in figures but mentioned in the scope [1]) is crucial: the flow requires significantly greater energy input to maintain the mean velocity, resulting in a system penalty. Furthermore, the large-scale recirculation zone can lead to flow instability and potential damage to downstream components, necessitating design solutions to stabilize the post-injection flow.

- *Study Limitations*

To maintain objective and comprehensive academic rigor, the limitations inherent to this preliminary computational study must be clearly articulated, demonstrating an in-depth understanding of the experimental weaknesses.[1]

First, the core limitation is the reliance on the standard $k-\epsilon$ turbulence model. As discussed, this model's isotropic assumption struggles with the high degree of anisotropy present in the JICF shear layer and recirculation bubble. While computationally inexpensive, it introduces model-form uncertainty that cannot be ignored.[6]

Second, the structural integrity of the simulation is compromised by the coarse discretization. The reported mesh size of 11,195 nodes is highly inadequate for a high-Reynolds number 3D flow with separation.[10] This insufficient mesh density, coupled with the lack of a documented Grid Convergence Index (GCI) study, means that the quantitative data presented may contain substantial discretization error.[11] This critical omission hinders the full validation and verification (V&V) required for high-confidence engineering results.

Third, the entire simulation is based on the assumption of a steady-state flow field. Jet-in-Cross-Flow is fundamentally an unsteady phenomenon characterized by periodic vortex shedding, jet flapping, and large-scale turbulent structures that are time-dependent. A steady RANS simulation averages out these critical dynamics, leading to inaccurate predictions of mixing rates and time-averaged reattachment points. Future analysis must migrate to Unsteady RANS (URANS) or Large Eddy Simulation (LES) to capture these transient effects.

V. CONCLUSIONS AND FUTURE WORK

The computational investigation successfully mapped the turbulent characteristics of high-speed jet injection into a confined pipe flow using the standard $k-\epsilon$ RANS model. The study demonstrated that the momentum flux ratio (J) is the primary control parameter governing flow distortion and turbulence generation. Increasing J effectively enhanced mixing characteristics, as evidenced by significant increases in turbulent kinetic energy (k) and

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25nov1417

turbulent intensity (I) in the jet wake and shear layer regions. These findings reinforce the application of JICF as a vital strategy for efficient mixing in aerospace systems.

However, the analysis also confirmed the limitations of the standard $k-\epsilon$ closure, particularly its non-physical prediction of a decreased turbulent dissipation rate (ϵ) despite increasing turbulence generation. This highlights the model's struggle to accurately capture complex turbulence transfer in highly anisotropic flows. The study, therefore, serves as a crucial preliminary exploration, identifying flow regimes and quantifying the limitations inherent in basic RANS modeling for JICF.

Based on the current findings and limitations, the following future research steps are mandated to achieve publication-grade scientific rigor:

- Verification and Validation: A comprehensive Grid Convergence Index (GCI) study must be conducted on a significantly refined computational mesh to quantify and minimize discretization errors, ensuring the robustness of the quantitative results.[11]
- Comparative Turbulence Modeling: The study should be replicated using more advanced RANS turbulence models (e.g., Realizable $k-\epsilon$ and $k-\omega$ SST) and the results compared to evaluate the performance improvement in predicting the recirculation bubble and ϵ distribution.[6]
- Unsteady Flow Analysis: Transitioning to Unsteady RANS (URANS) or Large Eddy Simulation (LES) will be necessary to capture the transient dynamics, such as vortex shedding and jet flapping, which are essential for true physical fidelity.
- Parameter Expansion: Future studies should investigate the mixing of different fluids or fluids at varying temperatures, as suggested in the original planning, and examine the effect of altering the jet injection angle.[1]

DECLARATIONS

➢ *Author Contributions*

The authors used the CRediT (Contributor Roles Taxonomy) framework for defining contributions [1]:

Conceptualization: ARNAB SEN, DEBASISH MANDAL; Methodology: ARNAB SEN, DEBASISH MANDAL, TANOY DEWANJEE; Software (ANSYS FLUENT 14.0 Simulation): ARNAB SEN; Formal analysis and investigation: DEBASISH MANDAL, TANOY DEWANJEE; Writing—original draft preparation: TANOY DEWANJEE; Writing—review and editing: ARNAB SEN, DEBASISH MANDAL, TANOY DEWANJEE. All authors read and approved the final manuscript.

➢ *Funding*

No funding was received to assist with the preparation of this manuscript. The authors did not receive support from any organization for the submitted work.[1]

➢ *Data Availability*

The raw computational fluid dynamics data generated during this study, including geometric definition files, boundary condition setups, and resultant solution fields (velocity, k, ϵ, I), are available from the corresponding author upon reasonable request.[1]

➢ *Conflict of Interest*

The authors have no relevant financial or non-financial interests to disclose.[1]

REFERENCES

[1]. Taylor J (1984) Mathematically modelled airflow through sampling pipes. J. Fire Prot. Eng.
[2]. Cole L (1999) Disturbances to pipe flow regimes by jet induction. Fire Safety J.
[3]. Bejan A (1994) Convective heat transfer coefficient. John Wiley & Sons, New York
[4]. Gamelin FX, Baquet G, Berthoin S, Thevenet D, Nourry C, Nottin S, Bosquet L (2009) Effect of high intensity intermittent training on heart rate variability in prepubescent children. Eur J Appl Physiol 105:731-738. https://doi.org/10.1007/s00421-008-0955-8
[5]. Slifka MK, Whitton JL (2000) Clinical implications of dysregulated cytokine production. J Mol Med. https://doi.org/10.1007/s001090000086
[6]. South J, Blass B (2001) The future of modern genomics. Blackwell, London
[7]. Brown B, Aaron M (2001) The politics of nature. In: Smith J (ed) The rise of modern genomics, 3rd edn. Wiley, New York, pp 230-257
[8]. Cartwright J (2007) Big stars have weather too. IOP Publishing PhysicsWeb. http://physicsweb.org/articles/news/11/6/16/1. Accessed 26 June 2007

➢ *Works cited*

[9]. Turbulence Project (Ready For Submission)=9-1-2014 (2).docx
[10]. Aerospace Systems | Research Communities by Springer Nature, accessed on November 13, 2025, https://communities.springernature.com/badges/aerospace-systems
[11]. Aerospace | Aims & Scope - MDPI, accessed on November 13, 2025, https://www.mdpi.com/journal/aerospace/about
[12]. Journal Scopes and Content | AIAA, accessed on November 13, 2025, https://aiaa.org/publications/journals/journal-scopes-and-content/

[13]. Mathematical Modeling of Turbulent Flows Using Advanced Computational Fluid Dynamics - Metallurgical and Materials Engineering, accessed on November 13, 2025, https://www.metall-mater-eng.com/index.php/home/article/download/1311/698
[14]. Influence of Common Modeling Choices for High-Speed Transverse Jet-Interaction Simulations | Request PDF - ResearchGate, accessed on November 13, 2025, https://www.researchgate.net/publication/265250976_Influence_of_Common_Modeling_Choices_for_High-Speed_Transverse_Jet-Interaction_Simulations
[15]. Improvement of Standard K-Epsilon Turbulence Model for Round Free Jets by Adjusting Closure Coefficients | Request PDF - ResearchGate, accessed on November 13, 2025, https://www.researchgate.net/publication/378001241_Improvement_of_Standard_K-Epsilon_Turbulence_Model_for_Round_Free_Jets_by_Adjusting_Closure_Coefficients
[16]. Turbulence model study for aerodynamic analysis of the leading edge tubercle wing for low Reynolds number flows - NIH, accessed on November 13, 2025, https://pmc.ncbi.nlm.nih.gov/articles/PMC11176843/
[17]. Turbulence Modeling Validation, Testing, and Development - NASA Technical Reports Server (NTRS), accessed on November 13, 2025, https://ntrs.nasa.gov/citations/19970017828
[18]. Best practice guidelines (Chapter 7) - Computational Fluid Dynamics for Engineers, accessed on November 13, 2025, https://www.cambridge.org/core/books/computational-fluid-dynamics-for-engineers/best-practice-guidelines/04FA8B0C0CFBBC6D2FF863AABBF28942
[19]. Grid independence studies applied to a field-scale computational fluid dynamic (CFD) model using the detached eddy simulation - the NOAA Institutional Repository, accessed on November 13, 2025, https://repository.library.noaa.gov/view/noaa/70202/noaa_70202_DS1.pdf
[20]. Computational Fluid Dynamics Best Practice Guidelines for Dry Cask Applications - Nuclear Regulatory Commission, accessed on November 13, 2025, https://www.nrc.gov/docs/ml1228/ML12286A301.pdf
[21]. New time scale based k-epsilon model for near-wall turbulence, accessed on November 13, 2025, https://ntrs.nasa.gov/citations/19930057912