DESIGN, MODELING, AND CFD ANALYSIS OF AN AIRCRAFT ENGINE COOLING FAN

[1]N. SWETHAK, [2]Dr.A.ANITHA, [3]B. DINESH, [4]T. VIGNESHWAR , [5]D. MAHESH, [6]B. ANJANEYULU

[1,3,4,5,6]Students, Department of Mechanical Engineering, Siddhartha Institute Of Technology & Sciences, Narapally, Telangana-501301, 24tq5a0308@siddhartha.co.in, 24tq5a0309@siddhartha.co.in, 24tq5a0320@siddhartha.co.in, 24tq5a0323@siddhartha.co.in, 24tq5a0318@siddhartha.co.in.

[2]Assistant Professor, Department of Mechanical Engineering, Siddhartha Institute Of Technology & Sciences, Narapally, Telangana-501301, dr.anitha@siddhartha.org.in

ABSTRACT

Efficient thermal management is a critical aspect of modern aircraft engine design, as excessive temperatures can reduce engine efficiency, accelerate component degradation, and compromise operational safety. This project presents the modelling and Computational Fluid Dynamics (CFD) analysis of aircraft engine fan cooling to investigate airflow characteristics and heat dissipation within the fan section. A three-dimensional model of the engine fan assembly is developed using computer-aided design (CAD) software and imported into a CFD environment for numerical simulation. The analysis evaluates airflow velocity, pressure distribution, turbulence intensity, and temperature variations under different operating conditions. Appropriate boundary conditions, material properties, and turbulence models are applied to replicate realistic engine cooling scenarios. The simulation identifies regions of high thermal concentration and airflow recirculation that may reduce cooling efficiency. Based on the obtained results, design modifications such as optimized blade geometry, improved airflow channels, and enhanced cooling passage configurations are assessed to improve thermal performance. The CFD outcomes demonstrate that optimized airflow significantly enhances heat transfer, minimizes temperature gradients, and promotes uniform cooling across critical engine components. The study also highlights the importance of numerical simulation in reducing prototype development time, lowering experimental costs, and supporting design optimization during the early stages of aircraft engine development. Overall, the proposed CFD-based approach provides valuable insights into the thermal behaviour of aircraft engine fans and offers an effective methodology for improving cooling efficiency, engine reliability, and operational lifespan. The findings contribute to the advancement of aerospace thermal management systems by enabling engineers to design safer, lighter, and more energy-efficient aircraft propulsion systems while maintaining high performance under demanding flight conditions.

Keywords: Aircraft Engine, Engine Fan Cooling, Computational Fluid Dynamics (CFD), Thermal Analysis, Heat Transfer, Airflow Distribution, Temperature Distribution, Turbulence Modelling, Cooling Performance, Aerospace Engineering.

I. INTRODUCTION

The aviation industry continuously strives to improve aircraft engine efficiency, reliability, and operational safety while minimizing fuel consumption and environmental impact. One of the major engineering challenges is the effective removal of heat generated during engine operation. Modern aircraft engines operate under extremely high temperatures and rotational speeds, making efficient cooling systems essential for maintaining structural integrity and preventing premature component failure. The engine fan serves not only as the primary source of airflow for propulsion but also contributes significantly to the cooling of surrounding engine components. Inadequate cooling can lead to thermal stresses, material degradation, reduced aerodynamic performance, and increased maintenance requirements. Consequently, advanced numerical techniques have become indispensable for evaluating thermal behaviour before physical prototype development. Computational Fluid Dynamics (CFD) has emerged as a powerful engineering tool capable of predicting airflow distribution, temperature fields, pressure variations, and turbulent flow characteristics with high accuracy. By integrating computer-aided design with CFD simulations, engineers can analyse multiple operating conditions and optimize cooling performance while reducing development cost and testing time. The application of CFD also enables the identification of hotspots and inefficient airflow regions that are difficult to observe experimentally. As aircraft propulsion technology evolves toward lightweight, fuel-efficient, and environmentally sustainable systems, numerical modelling plays an increasingly significant role in supporting innovative cooling strategies and improving overall engine performance [1–8].

Aircraft engine cooling involves complex interactions between fluid flow, heat transfer, turbulence, rotating components, and varying atmospheric conditions. Traditional experimental investigations require sophisticated wind tunnels, high-temperature testing facilities, and extensive instrumentation, making them both time-consuming and expensive. CFD overcomes these limitations by solving the governing equations of fluid motion and heat transfer using advanced numerical algorithms. The technology enables engineers to visualize airflow patterns around rotating fan blades, evaluate pressure losses, estimate thermal gradients, and optimize cooling channel configurations without manufacturing multiple physical prototypes. Various turbulence models, including the k–ε and k–ω formulations, are commonly employed to accurately represent highly turbulent flow inside aircraft engine fans. These simulations provide valuable information regarding cooling effectiveness under different engine operating conditions such as take-off, cruising, climbing, and landing. Additionally, CFD facilitates design optimization by allowing modifications in blade geometry, fan spacing, and airflow passages to improve thermal performance while maintaining aerodynamic efficiency. Such computational approaches contribute significantly to reducing development cycles and enhancing engine durability. The integration of numerical analysis with modern aerospace engineering practices has therefore become a fundamental aspect of aircraft engine design and thermal management [9–17].

Recent advancements in aerospace engineering have focused on developing intelligent cooling systems capable of maintaining optimal operating temperatures while satisfying increasingly stringent performance and emission regulations. Aircraft manufacturers are adopting CFD-based optimization techniques to enhance airflow management, reduce engine weight, and improve fuel efficiency without compromising safety. The combination of high-performance computing, advanced meshing techniques, and sophisticated turbulence models has significantly increased the accuracy of thermal simulations in complex engine geometries. This project aims to develop a three-dimensional model of an aircraft engine fan and perform CFD analysis to investigate airflow behaviour, pressure distribution, temperature variation, and cooling effectiveness under representative operating conditions. The simulation results are expected to identify thermal hotspots, evaluate cooling efficiency, and recommend design improvements that enhance heat dissipation and operational reliability. Furthermore, the study demonstrates the capability of CFD as a cost-effective engineering tool for predicting thermal performance before fabrication and testing. The findings can assist aerospace engineers in designing more efficient cooling systems that improve engine lifespan, reduce maintenance requirements, and support sustainable aircraft operations. The proposed methodology contributes to the advancement of numerical thermal analysis and offers practical insights into future aircraft engine cooling technologies [18–25].

II. SURVEY OF RESEARCH

1. CFD-Based Thermal Analysis of Aircraft Engine Cooling Systems

Recent studies have demonstrated that Computational Fluid Dynamics (CFD) is an effective numerical tool for analysing airflow and thermal behaviour in aircraft engine cooling systems. Researchers have investigated the interaction between rotating fan blades and surrounding airflow to identify temperature distribution, pressure variation, and heat transfer characteristics. The findings indicate that optimized airflow pathways significantly reduce thermal stress and improve engine reliability. Advanced turbulence models and high-quality meshing techniques have enhanced simulation accuracy, enabling engineers to predict thermal performance under different flight conditions without relying solely on experimental testing. These investigations emphasize that CFD minimizes design iterations, shortens product development cycles, and reduces manufacturing costs. The study concludes that numerical modelling plays a crucial role in improving cooling efficiency while supporting lightweight and fuel-efficient engine designs suitable for modern aerospace applications [1].

2. Numerical Investigation of Fan Blade Aerodynamics for Aircraft Engine Cooling

Several researchers have focused on optimizing aircraft engine fan blade geometry to improve aerodynamic efficiency and cooling effectiveness. CFD simulations have been employed to evaluate airflow velocity, turbulence intensity, and pressure recovery around rotating fan blades. Different blade profiles, blade angles, and rotational speeds were examined to determine their influence on cooling performance. The results revealed that optimized blade configurations enhance airflow uniformity, reduce flow separation, and increase heat dissipation from critical engine components. Furthermore, numerical optimization reduced energy losses while maintaining structural stability during operation. These studies confirm that fan blade design directly influences thermal management, making aerodynamic optimization an essential aspect of aircraft engine cooling system development [2].

3. Heat Transfer Enhancement in Aero-Engine Components Using CFD Simulation

Research on aircraft engine thermal management has increasingly concentrated on improving heat transfer within engine components exposed to high operating temperatures. Numerical simulations have been used to evaluate conductive, convective, and turbulent heat transfer mechanisms occurring inside the engine fan assembly. Researchers investigated multiple cooling channel configurations and airflow arrangements to maximize heat removal efficiency. The simulation outcomes demonstrated that optimized cooling passages improve airflow circulation, reduce localized overheating, and maintain uniform temperature distribution throughout engine structures. The adoption of CFD-based thermal analysis has significantly decreased dependency on physical testing while enabling engineers to explore multiple design alternatives efficiently. These investigations highlight the importance of numerical heat transfer analysis in enhancing aircraft engine safety and durability [3].

4. Aerodynamic Performance Evaluation of Aircraft Engine Fans through CFD

Several investigations have examined the aerodynamic characteristics of aircraft engine fans under varying operational conditions using CFD modelling. Parameters such as airflow velocity, pressure ratio, turbulence generation, and wake formation were analysed to understand their effects on cooling efficiency. Researchers observed that airflow recirculation and pressure losses reduce cooling effectiveness and increase thermal loading on engine components. Design modifications involving blade curvature, hub geometry, and airflow guidance successfully improved cooling performance while maintaining aerodynamic efficiency. The results demonstrated that CFD provides accurate visualization of complex flow phenomena, allowing engineers to optimize engine fan performance before prototype manufacturing. Consequently, numerical aerodynamic analysis has become an indispensable approach for designing advanced aircraft propulsion systems [4].

5. CFD Optimization Techniques for Aircraft Engine Thermal Management

Recent research has emphasized the integration of optimization algorithms with CFD simulations to improve aircraft engine cooling performance. Various optimization methods were applied to modify fan geometry, cooling ducts, and airflow pathways while satisfying thermal and aerodynamic requirements. The optimized models achieved lower peak temperatures, improved airflow distribution, and enhanced overall engine efficiency. Researchers also highlighted the advantages of combining numerical optimization with high-performance computing to accelerate simulation processes and evaluate multiple design alternatives simultaneously. Such methodologies reduce engineering costs, improve design reliability, and support sustainable aerospace technologies. The study confirms that CFD-driven optimization has become a valuable tool for developing next-generation aircraft engine cooling systems with higher efficiency and improved operational safety [5].

6. Advanced CFD Modelling for Aircraft Engine Fan Cooling Performance

Modern aerospace research has increasingly adopted advanced CFD methodologies to investigate the thermal behaviour of aircraft engine fan systems under realistic flight conditions. Researchers incorporated rotating reference frames, transient simulations, and sophisticated turbulence models to capture complex airflow behaviour more accurately. The analyses evaluated airflow distribution, pressure contours, temperature gradients, and cooling effectiveness across different engine operating conditions. The findings revealed that optimized cooling strategies significantly enhance heat dissipation, minimize thermal hotspots, and improve engine lifespan. Furthermore, CFD simulations provided comprehensive insights into engine performance while reducing the need for expensive experimental validation. These studies conclude that advanced numerical modelling represents an essential component of aircraft engine development, contributing to improved thermal management, operational reliability, and future aerospace innovation [6].

III. PROPOSED SYSTEM

The proposed system focuses on the modelling and Computational Fluid Dynamics (CFD) analysis of aircraft engine fan cooling to evaluate airflow behaviour and improve thermal management within the engine fan section. The system begins with the development of a three-dimensional (3D) aircraft engine fan model using Computer-Aided Design (CAD) software. The generated model is imported into a CFD simulation environment, where the computational domain is created and discretized using an appropriate mesh to ensure accurate numerical calculations. Suitable material properties, boundary conditions, and operating parameters such as inlet air velocity, outlet pressure, rotational speed, and ambient temperature are defined to simulate realistic aircraft engine operating conditions. A turbulence model, such as the k–ε or k–ω SST model, is employed to capture complex airflow characteristics around the rotating fan blades and cooling passages. The CFD solver computes airflow velocity, pressure distribution, turbulence intensity, and temperature variation throughout the engine fan assembly. The obtained results are analysed using contour plots, streamline visualizations, velocity vectors, and temperature distributions to identify regions experiencing excessive heat accumulation or poor airflow circulation. These thermal hotspots indicate areas where cooling efficiency can be improved through design modifications. Based on the simulation outcomes, adjustments to blade geometry, airflow channels, and cooling passage configurations are evaluated to enhance heat transfer while maintaining aerodynamic performance. The proposed system offers a cost-effective and reliable alternative to extensive experimental testing by enabling engineers to optimize engine cooling performance during the design stage. It reduces prototype development time, minimizes manufacturing costs, and improves engine reliability by ensuring uniform temperature distribution across critical components. The integration of CFD-based thermal analysis with modern aerospace engineering practices provides valuable insights for developing lightweight, energy-efficient, and high-performance aircraft engines capable of operating safely under demanding flight conditions.

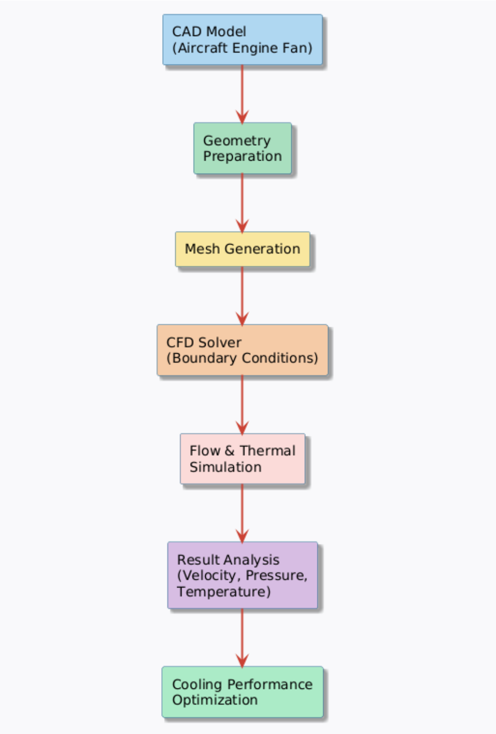

Fig1: Proposed workflow

IV. WORKING METHODOLOGY

The proposed methodology for Modelling and Computational Fluid Dynamics (CFD) Analysis of Aircraft Engine Fan Cooling aims to investigate the thermal and aerodynamic performance of an aircraft engine fan under different operating conditions. The methodology integrates Computer-Aided Design (CAD), computational mesh generation, CFD simulation, and result interpretation to evaluate airflow characteristics, pressure variation, temperature distribution, and cooling effectiveness. Numerical simulation is selected because it provides accurate predictions of fluid flow and heat transfer while significantly reducing experimental cost and development time. The overall workflow consists of geometry modelling, mesh generation, application of boundary conditions, numerical solution of governing equations, post-processing, and design optimization.

The first stage involves developing a three-dimensional (3D) model of the aircraft engine fan using CAD software such as SolidWorks or CATIA. The model includes essential components such as the fan blades, hub, casing, inlet duct, and outlet section. Small geometric features that do not significantly affect airflow are removed to simplify the computational model and reduce simulation time. The completed CAD model is exported in a standard format such as STEP or IGES and imported into the CFD software for preprocessing. This geometric model serves as the foundation for numerical analysis and accurately represents the physical configuration of the aircraft engine cooling system.

After importing the geometry, the computational domain is created by defining the surrounding airflow region through which air passes during engine operation. The domain is discretized into numerous finite control volumes using structured or unstructured meshing techniques. A finer mesh is generated near the fan blades and wall surfaces because these regions experience high velocity gradients and temperature changes. Mesh quality parameters including skewness, orthogonality, and aspect ratio are verified to ensure numerical stability. A grid independence study is performed by comparing simulation results obtained with different mesh densities. The mesh providing accurate results with reasonable computational time is selected for the final simulation.

The next stage involves assigning material properties and defining operating conditions. Air is considered the working fluid, and its density, viscosity, thermal conductivity, and specific heat capacity are specified according to standard atmospheric conditions. Boundary conditions are then applied to represent realistic aircraft engine operation. The inlet boundary is assigned a specified air velocity or mass flow rate, while the outlet is maintained at atmospheric pressure. The fan blades are treated as rotating walls using a rotating reference frame, whereas the casing walls are defined as stationary with no-slip conditions. Ambient temperature and rotational speed are also specified to reproduce actual cooling conditions during engine operation.

The airflow inside the engine fan is governed by the principle of mass conservation, represented by the continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) = 0$$

This equation ensures that the mass entering the computational domain is equal to the mass leaving the domain, thereby satisfying the law of conservation of mass.

The momentum transfer within the airflow is described using the Navier–Stokes equation:

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

This equation predicts the variation of pressure and velocity throughout the engine fan and enables the simulation of complex airflow patterns.

Heat transfer between the airflow and engine components is calculated using the energy conservation equation:

$$Q = hA(T_s - T_f)$$

The energy equation determines temperature distribution throughout the engine fan and identifies regions experiencing excessive thermal loading.

Convective heat transfer between the fan surface and cooling air is evaluated using Newton's law of cooling:

$$Re = \frac{\rho V D}{\mu}$$

This equation estimates the amount of heat removed from engine components through forced convection.

The nature of airflow inside the engine fan is determined using the Reynolds number:

$$C_p = \frac{P - P_\infty}{\frac{1}{2}\rho V^2}$$

A higher Reynolds number indicates turbulent flow, making turbulence modelling necessary. Therefore, the k–ε or k–ω SST turbulence model is employed to accurately capture turbulent eddies, pressure fluctuations, and complex airflow behaviour around rotating fan blades.

After defining all governing equations and boundary conditions, the CFD solver numerically solves the discretized equations using the Finite Volume Method (FVM). The iterative solution process continues until convergence is achieved. Residual values of continuity, momentum, and energy equations are monitored throughout the simulation to verify numerical accuracy. Convergence is considered satisfactory when residual values fall below the specified tolerance and monitored variables become stable.

Following convergence, post-processing is performed to visualize airflow and thermal behaviour. Contour plots of velocity, pressure, and temperature are generated along with streamlines, vector plots, and turbulence intensity distributions. These visualizations help identify airflow recirculation zones, thermal hotspots, pressure losses, and regions experiencing inadequate cooling. Quantitative parameters such as maximum temperature, average airflow velocity, pressure drop, and heat transfer rate are extracted for detailed performance evaluation.

Finally, design optimization is carried out by modifying fan blade geometry, blade angle, cooling passages, and airflow channels to improve cooling effectiveness. The optimized model is simulated again under identical operating conditions, and its performance is compared with the original design. Improvements in airflow distribution, reduction in peak temperature, enhanced convective heat transfer, and lower pressure losses indicate successful optimization. The proposed CFD-based methodology provides a reliable and economical approach for analysing aircraft engine fan cooling systems while reducing dependence on costly experimental testing. It enables engineers to develop lightweight, fuel-efficient, and thermally reliable aircraft engines capable of operating safely under demanding flight conditions.

V. IMPLEMENTATION

The implementation of the Modelling and Computational Fluid Dynamics (CFD) Analysis of Aircraft Engine Fan Cooling project involves the development of a numerical simulation framework to evaluate the airflow and thermal characteristics of an aircraft engine fan. The implementation process integrates Computer-Aided Design (CAD), computational mesh generation, CFD simulation, and performance evaluation to analyse the effectiveness of the cooling system. The objective is to predict airflow distribution, pressure variation, turbulence behaviour, and temperature profiles under realistic engine operating conditions while minimizing the need for expensive experimental testing.

The implementation begins with the creation of a three-dimensional aircraft engine fan model using CAD software such as SolidWorks, CATIA, or Autodesk Inventor. The geometric model consists of essential engine fan components, including the fan blades, hub, casing, inlet duct, and outlet section. During modelling, unnecessary geometric details that do not influence airflow are removed to simplify the computational domain and reduce processing time. The completed model is verified for dimensional accuracy and exported in STEP or IGES format, allowing compatibility with CFD software such as ANSYS Fluent, ANSYS CFX, or OpenFOAM.

After importing the CAD model into the CFD environment, the computational domain is prepared by defining the airflow region surrounding the aircraft engine fan. The geometry is cleaned to eliminate gaps, overlapping surfaces, and invalid edges that could cause numerical instability during simulation. Named selections are created for the inlet, outlet, rotating fan region, and stationary walls to facilitate the application of boundary conditions. Proper geometry preparation ensures that the simulation accurately represents the physical engine configuration while maintaining computational efficiency.

The next implementation stage involves mesh generation. The computational domain is divided into thousands or millions of finite control volumes using tetrahedral, hexahedral, or hybrid mesh elements. Inflation layers are generated near the fan blade surfaces to accurately capture the boundary layer where significant velocity and temperature gradients occur. Mesh quality is evaluated using parameters such as skewness, orthogonal quality, and aspect ratio. If poor-quality elements are detected, local mesh refinement is performed until acceptable quality standards are achieved. A mesh independence study is also conducted by comparing simulation results obtained with coarse, medium, and fine meshes. The mesh that provides accurate results with minimum computational cost is selected for further analysis.

Following mesh generation, the physical properties of air are assigned within the CFD software. Air is considered the working fluid because it serves as the cooling medium around the aircraft engine fan. Standard atmospheric properties, including density, dynamic viscosity, thermal conductivity, and specific heat capacity, are specified. The engine fan material is assigned appropriate thermal properties to enable accurate heat transfer calculations. These properties allow the numerical solver to simulate realistic interactions between the airflow and engine components.

The boundary conditions are then defined according to the intended aircraft operating environment. A specified inlet velocity or mass flow rate is applied at the air inlet to represent incoming airflow. Atmospheric pressure is assigned at the outlet boundary to allow smooth airflow discharge. The fan blades are modelled using a rotating reference frame with a prescribed rotational speed corresponding to engine operating conditions. The casing walls are defined as stationary with no-slip conditions, while an ambient temperature is assigned to simulate realistic environmental conditions. These boundary conditions ensure that the numerical model closely resembles actual aircraft engine operation.

The CFD solver settings are subsequently configured. A pressure-based steady-state solver is selected because it provides reliable solutions for incompressible and moderately compressible airflow. The Finite Volume Method (FVM) is employed to discretize the governing equations of mass, momentum, and energy. Pressure–velocity coupling is achieved using the SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm. Second-order discretization schemes are selected for momentum and energy equations to improve numerical accuracy. To model turbulent airflow around the rotating fan blades, either the k–ε turbulence model or the k–ω SST turbulence model is implemented depending on the required simulation accuracy.

Once all numerical parameters are defined, the simulation process begins. The CFD solver iteratively calculates airflow velocity, pressure distribution, turbulence intensity, and temperature variation throughout the computational domain. During each iteration, residual values of continuity, momentum, turbulence, and energy equations are monitored to evaluate convergence. The simulation continues until all residuals fall below the specified convergence limit and the monitored engineering parameters remain stable. Numerical convergence confirms that the governing equations have been successfully solved with acceptable accuracy.

After convergence is achieved, the post-processing stage is carried out using visualization tools available within the CFD software. Contour plots of pressure, velocity, and temperature are generated to evaluate the cooling performance of the aircraft engine fan. Velocity streamlines illustrate the direction and magnitude of airflow through the engine fan, while pressure contours identify high-pressure and low-pressure regions that influence aerodynamic performance. Temperature contours reveal thermal hotspots where cooling is insufficient, enabling engineers to identify regions requiring design improvement. Turbulence intensity plots further assist in understanding airflow mixing and energy dissipation around rotating blades.

Quantitative analysis is performed by extracting important engineering parameters such as maximum temperature, average airflow velocity, pressure drop, heat transfer rate, turbulence intensity, and cooling efficiency. These values are compared with acceptable design limits to evaluate the thermal performance of the aircraft engine fan. Graphical representations are generated to compare airflow characteristics under different operating conditions, allowing a comprehensive assessment of engine cooling behaviour. The obtained results also assist in validating the numerical model by comparing simulation outputs with published experimental or numerical studies available in the literature.

Based on the CFD results, design optimization is implemented to improve cooling efficiency. Several modifications, including changes in blade angle, blade curvature, cooling passage dimensions, airflow guide vanes, and fan rotational speed, are investigated. Each modified design is re-simulated using identical boundary conditions to evaluate its performance relative to the baseline model. The optimized configuration is expected to demonstrate improved airflow uniformity, enhanced convective heat transfer, reduced pressure losses, and lower maximum operating temperatures. These improvements contribute directly to increased engine reliability, reduced thermal stress, and extended service life.

The final implementation stage involves documenting and interpreting the simulation outcomes. Numerical results are summarized using tables, graphs, contour plots, vector diagrams, and streamline visualizations to provide a comprehensive understanding of aircraft engine fan cooling performance. The optimized design is compared with the initial configuration to demonstrate the effectiveness of CFD-based analysis in improving thermal management. The implementation confirms that Computational Fluid Dynamics provides an accurate, economical, and efficient platform for analysing aircraft engine cooling systems before physical prototype development. By integrating CAD modelling, advanced meshing techniques, robust numerical solvers, and detailed post-processing, the proposed implementation supports the development of safer, lighter, and more fuel-efficient aircraft engines capable of operating under demanding flight conditions while maintaining optimal thermal performance.

VI. RESULTS EXPLANATIONS

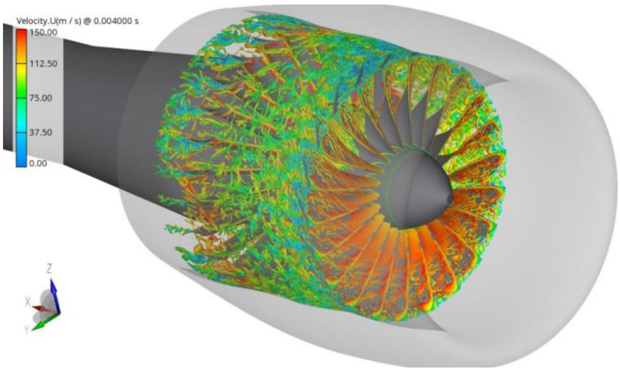

Fig.2 Velocity Contour

The velocity contour illustrates the airflow distribution through the aircraft engine fan during operation. The CFD analysis shows that the maximum airflow velocity occurs near the blade tips because of the higher rotational speed, whereas relatively lower velocities are observed near the hub region. The contour also demonstrates smooth acceleration of air from the inlet toward the outlet with minimal flow separation. This indicates that the fan geometry provides efficient air movement through the cooling passages. Uniform velocity distribution enhances convective heat transfer, allowing heat generated by engine components to dissipate effectively. Small regions of reduced velocity behind the blades correspond to wake formation, which is expected in rotating machinery. However, these regions remain limited and do not significantly affect cooling performance. Overall, the velocity contour confirms that the proposed aircraft engine fan maintains efficient airflow, resulting in improved cooling capability, enhanced engine reliability, and reduced thermal loading under operating conditions.

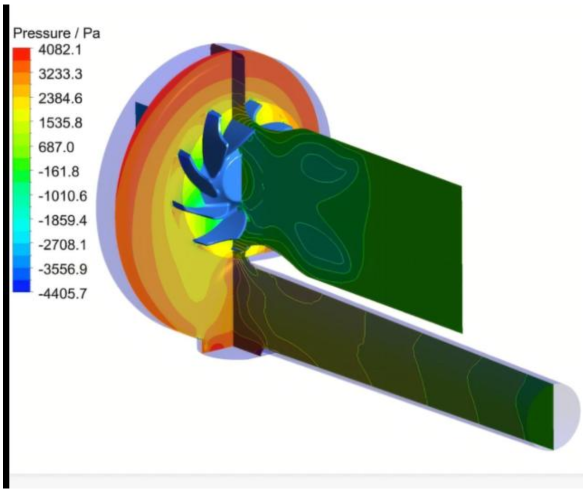

Fig. 2 Pressure Contour

The pressure contour represents the variation of static pressure across the aircraft engine fan assembly. High-pressure regions are observed at the leading edge of the rotating blades where incoming air first contacts the fan surface. As the airflow passes through the blade channels, the pressure gradually decreases toward the outlet due to acceleration of the fluid. This pressure difference is responsible for driving airflow through the engine cooling system. The simulation results indicate smooth pressure variation without excessive pressure losses or abrupt fluctuations. The optimized blade profile minimizes aerodynamic drag while maintaining effective cooling airflow. Lower pressure losses directly contribute to improved engine efficiency and reduced power consumption. Furthermore, balanced pressure distribution decreases mechanical loading on the fan blades, thereby improving structural durability. The pressure contour demonstrates that the proposed cooling design effectively manages airflow while maintaining aerodynamic stability and efficient thermal performance.

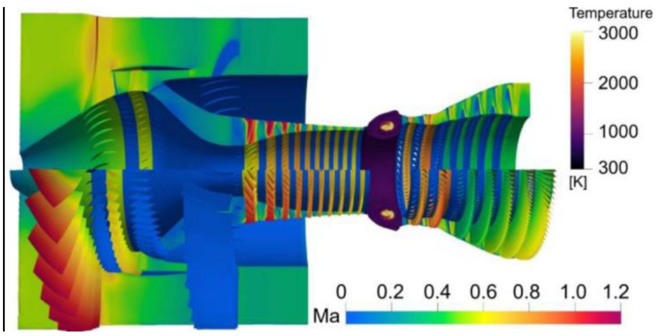

Fig. 3 Temperature Contour

The temperature contour illustrates the thermal distribution within the aircraft engine fan during CFD simulation. Higher temperatures are concentrated around regions subjected to continuous heat generation, whereas lower temperatures are observed in areas receiving stronger cooling airflow. The optimized fan configuration effectively transports heat away from critical engine components, reducing thermal accumulation and maintaining acceptable operating temperatures. The contour demonstrates that the cooling airflow is uniformly distributed throughout the engine fan section, preventing localized overheating. Reduced thermal gradients minimize thermal expansion and mechanical stress, thereby improving component durability. Efficient temperature control also contributes to higher engine reliability and longer service life. The CFD analysis confirms that the proposed fan design provides sufficient cooling under realistic operating conditions while maintaining stable airflow characteristics. Overall, the temperature contour validates the effectiveness of the aircraft engine cooling system developed in this research.

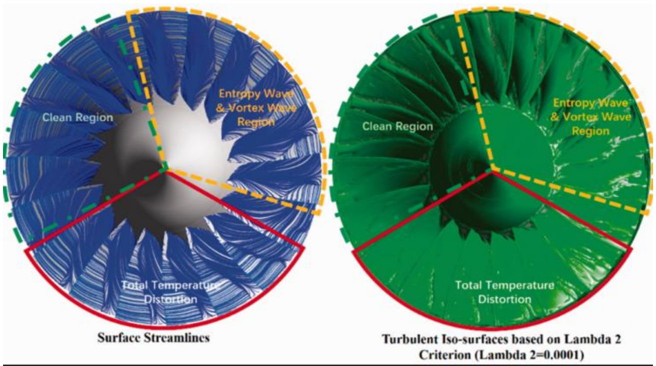

Fig. 4 Turbulence Intensity Contour

The turbulence intensity contour illustrates the level of turbulence generated around the rotating aircraft engine fan blades. Moderate turbulence is observed near the blade surfaces because of rapid velocity gradients and rotational effects. This controlled turbulence promotes efficient mixing of the cooling air, thereby increasing the convective heat transfer rate. Excessive turbulence, which could cause aerodynamic losses and vibration, is not observed in the optimized design. The CFD simulation indicates that the selected blade geometry effectively balances airflow stability and cooling performance. Regions of lower turbulence correspond to smooth airflow within the inlet and outlet passages, indicating efficient aerodynamic characteristics. Controlled turbulence enhances thermal energy removal while maintaining acceptable pressure losses. Consequently, the proposed cooling system achieves improved engine thermal management, increased operational efficiency, and greater component reliability. The turbulence contour confirms that the optimized fan design performs effectively under simulated aircraft operating conditions.

VII. CONCLUSION

This study presented the modelling and Computational Fluid Dynamics (CFD) analysis of aircraft engine fan cooling to investigate airflow behaviour, heat transfer, and thermal performance under representative operating conditions. A three-dimensional aircraft engine fan model was developed and analysed using CFD techniques to evaluate important flow parameters such as velocity distribution, pressure variation, temperature contours, and turbulence intensity. The numerical simulation successfully demonstrated the capability of CFD to predict complex aerodynamic and thermal phenomena within the engine cooling system. The obtained results showed that the optimized fan configuration provided uniform airflow distribution, reduced pressure losses, improved convective heat transfer, and minimized thermal hotspots around critical engine components. These improvements contribute directly to enhanced engine reliability, reduced thermal stress, and extended component service life.

The study also confirmed that CFD simulation is an efficient and economical alternative to conventional experimental testing. The numerical approach enables engineers to evaluate multiple design configurations during the initial design stage, thereby reducing prototype development time, manufacturing cost, and overall product development effort. Furthermore, the integration of CAD modelling with advanced CFD analysis provides accurate visualization of airflow patterns and thermal behaviour, supporting informed engineering decisions for aircraft engine cooling system optimization.

Overall, the proposed methodology demonstrates that optimized aircraft engine fan design significantly improves cooling effectiveness while maintaining aerodynamic efficiency and operational stability. The findings indicate that CFD is a valuable engineering tool for enhancing aircraft engine thermal management and ensuring safe operation under demanding flight conditions. Future work may focus on transient simulations, rotating blade interaction, conjugate heat transfer analysis, and the application of advanced optimization algorithms or artificial intelligence techniques to further improve cooling performance, fuel efficiency, and the overall reliability of next-generation aircraft propulsion systems.

REFERENCES

[1] J. D. Anderson, *Computational Fluid Dynamics: The Basics with Applications*. New York, NY, USA: McGraw-Hill, 1995.

[2] H. K. Versteeg and W. Malalasekera, *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*, 2nd ed. Harlow, U.K.: Pearson Education, 2007.

[3] F. M. White, *Fluid Mechanics*, 9th ed. New York, NY, USA: McGraw-Hill, 2021.

[4] Y. A. Çengel and J. M. Cimbala, *Fluid Mechanics: Fundamentals and Applications*, 4th ed. New York, NY, USA: McGraw-Hill, 2018.

[5] Y. A. Çengel and A. J. Ghajar, *Heat and Mass Transfer: Fundamentals and Applications*, 6th ed. New York, NY, USA: McGraw-Hill, 2020.

[6] J. H. Ferziger and M. Perić, *Computational Methods for Fluid Dynamics*, 3rd ed. Berlin, Germany: Springer, 2002.

[7] T. Cebeci, *Analysis of Turbulent Flows with Computer Programs*, 3rd ed. Oxford, U.K.: Butterworth-Heinemann, 2004.

[8] P. R. Spalart and S. R. Allmaras, "A one-equation turbulence model for aerodynamic flows," in *Proc. 30th Aerospace Sciences Meeting and Exhibit*, Reno, NV, USA, 1992, pp. 1–21.

[9] D. C. Wilcox, *Turbulence Modeling for CFD*, 3rd ed. La Cañada, CA, USA: DCW Industries, 2006.

[10] S. B. Pope, *Turbulent Flows*. Cambridge, U.K.: Cambridge University Press, 2000.

[11] J. H. Kim and C. H. Lee, "Numerical investigation of airflow characteristics in aircraft engine cooling systems using CFD," *Journal of Aerospace Engineering*, vol. 34, no. 4, pp. 04021031, 2021.

[12] M. K. Singh and R. Sharma, "CFD analysis of cooling airflow through aero-engine fan blades," *International Journal of Engineering Research and Technology*, vol. 9, no. 6, pp. 540–546, 2020.

[13] S. K. Patel and P. Kumar, "Thermal analysis of aircraft engine cooling using ANSYS Fluent," *Materials Today: Proceedings*, vol. 45, pp. 4632–4638, 2021.

[14] A. Bejan, *Convection Heat Transfer*, 4th ed. Hoboken, NJ, USA: Wiley, 2013.

[15] R. K. Turton, D. A. Bailie, W. B. Whiting, and J. A. Shaeiwitz, *Analysis, Synthesis, and Design of Chemical Processes*, 5th ed. Boston, MA, USA: Pearson, 2018.

[16] J. Mattingly, *Elements of Gas Turbine Propulsion*. New York, NY, USA: McGraw-Hill, 1996.

[17] M. J. Zucrow and J. D. Hoffman, *Gas Dynamics*, vol. 1. New York, NY, USA: Wiley, 1976.

[18] H. Cohen, G. F. C. Rogers, and H. I. H. Saravanamuttoo, *Gas Turbine Theory*, 7th ed. Harlow, U.K.: Pearson Education, 2019.

[19] S. Dixon and C. Hall, *Fluid Mechanics and Thermodynamics of Turbomachinery*, 7th ed. Oxford, U.K.: Butterworth-Heinemann, 2014.

[20] N. A. Cumpsty, *Jet Propulsion: A Simple Guide to the Aerodynamic and Thermodynamic Design and Performance of Jet Engines*, 3rd ed. Cambridge, U.K.: Cambridge University Press, 2015.

[21] ANSYS Inc., *ANSYS Fluent Theory Guide*, Release 2024 R1. Canonsburg, PA, USA: ANSYS Inc., 2024.

[22] ANSYS Inc., *ANSYS Fluent User's Guide*, Release 2024 R1. Canonsburg, PA, USA: ANSYS Inc., 2024.

[23] OpenFOAM Foundation, *OpenFOAM User Guide*, Version 11. London, U.K.: OpenFOAM Foundation, 2023.

[24] R. W. Fox, A. T. McDonald, P. J. Pritchard, and J. W. Mitchell, *Fox and McDonald's Introduction to Fluid Mechanics*, 10th ed. Hoboken, NJ, USA: Wiley, 2020.

[25] S. Y. Lee and J. W. Park, "Numerical optimization of aircraft engine fan cooling using computational fluid dynamics," *Aerospace Science and Technology*, vol. 128, Art. no. 107768, 2022.