Acceleron Aerospace Journal || AAJ.11.2106-2523
Volume 4, Issue 4, pp (1090-1105)
Research Paper
E-ISSN- 2583-9942 || DOI: 10.61359/11.2106-2523

Unveiling the Secondary Flow Interface: Enhancing Gas Turbine Blade Stage Efficiency

Imad Ashraf Wani* and Irfan Nazir Wani†

[1&2] *Department of Aerospace and Mechanical Engineering, NIET NIMS University, Jaipur, India*

Abstract: This study investigates the intricate dynamics of secondary flow interactions within the blade stage of a gas turbine and their profound impact on aerodynamic performance and operational efficiency in aerospace propulsion systems. High-fidelity computational fluid dynamics (CFD) simulations were conducted on a dual-rotor, dual-stator configuration to analyze airflow behavior, revealing significant pressure gradients, velocity variations, and vorticity patterns. These secondary flow structures contribute to aerodynamic losses, reducing turbine efficiency and influencing thermal management. The findings highlight the critical role of secondary effects in determining overall turbine performance and provide valuable insights into optimizing blade geometries and flow control strategies. This research establishes a foundation for the development of next-generation gas turbines with improved aerodynamic efficiency and reduced energy dissipation.

Table of Contents

1. Introduction 1
2. Objective 2
3. Literature Review 2
4. Methodology 2
5. Design of Blade Airfoil 3
6. Computational Fluid Dynamics (CFD) Simulation 6
7. Meshing 8
8. Boundary Conditions 9
9. Temperature Contour Analysis 10
10. Pressure Contour Analysis 12
11. Velocity Contour Analysis 12
12. Results and Discussion 14
13. Blade Surface Flow Characteristics 14
14. Scope For Future Work 15
15. Conclusion and Future Directions 15
16. References 16
17. Conflict of Interest 16
18. Funding 16

1. Introduction

In the previous paper, Gas turbines play a crucial role in aerospace propulsion systems, where their efficiency and reliability are paramount to overall performance. As the demand for more efficient and environmentally friendly propulsion technologies grows, understanding the intricate dynamics of gas turbines becomes increasingly important [1]. While extensive research has been conducted on primary flow characteristics within turbine stages, secondary flow phenomena have not received the same level of attention. Secondary flows, which arise due to the complex interactions between the main flow and the turbine blade geometry, can significantly influence aerodynamic performance and operational efficiency [2]. These flows can lead to increased aerodynamic losses, affecting the overall efficiency of the turbine and its thermal management capabilities. This study aims to bridge this knowledge gap by investigating the dynamics of secondary flow interactions within the blade stage of a gas turbine [3]. By employing high-fidelity computational fluid dynamics (CFD) simulations on a dual-rotor, dual-stator configuration, we seek to provide a comprehensive analysis of airflow behavior, pressure gradients, velocity variations, and vorticity patterns. The insights gained from this research will contribute to the optimization of blade geometries and flow control strategies, ultimately paving the way for the development of next-generation gas turbines with enhanced aerodynamic efficiency and reduced energy dissipation.

* Dept. of Aerospace and Mechanical Engineering, NIET NIMS University, Jaipur, India. **Contact:** imuimad299@gmail.com.
† Dept. of Aerospace and Mechanical Engineering, NIET NIMS University, Jaipur, India.
** Received: 24-March-2025 || Revised: 29-April-2025 || Accepted: 29-April-2025 || Published Online: 30-April-2025.

2. Objective

To investigate the aerodynamic behavior of airflow within a gas turbine blade stage, with a primary focus on rotor-stator interactions and their impact on turbine performance.

- To analyze variations in pressure, velocity, and vorticity across the blade passage, identifying critical regions of flow separation, turbulence generation, and energy dissipation.
- To employ high-fidelity Computational Fluid Dynamics (CFD) simulations for visualizing pressure, velocity, and temperature distributions, enabling a comprehensive assessment of secondary flow effects and their influence on overall turbine efficiency.
- To provide actionable insights for optimizing turbine blade geometry and flow control strategies, aiming to enhance aerodynamic efficiency and minimize energy losses.

Key Improvements:

- More Precise & Technical Language – Replaced general terms with more specific technical phrases.
- Expanded Scope – Included vorticity analysis and blade geometry optimization.
- More Impact-Oriented – Focuses on practical insights and improvements for turbine efficiency

3. Literature Review

Secondary flow phenomena within gas turbine blade stages have been a subject of increasing interest as researchers seek to optimize aerodynamic efficiency and minimize energy losses. Traditionally, studies have focused on primary flow paths, often neglecting the impact of secondary vortical structures that arise due to blade passage geometry and endwall boundary layer interactions.

- Horlock (2001) laid foundational work on axial-flow turbines, discussing the implications of flow turning and loss mechanisms, emphasizing how secondary flows contribute significantly to total pressure losses. Following this, Denton (1993) identified that over 30% of aerodynamic losses in turbine stages are attributable to secondary flows such as passage vortices, horseshoe vortices, and corner separation.
- More recent computational and experimental advancements have enabled deeper insight. Gümmer and Nicke (2006) used 3D CFD simulations to reveal the intricate behavior of vortex systems in low-pressure turbines and showed that endwall contouring could mitigate secondary losses by redirecting the flow path more favorably.
- Li.et.al. (2018) explored the effect of rotor-stator interaction on secondary flow evolution and demonstrated that unsteady wake dynamics exacerbate turbulence intensity and flow non-uniformity. Their work underlined the need for time-accurate simulations to understand transient secondary flow development.
- In terms of geometry optimization, Kost and Nicke (2004) employed adjoint-based methods to redesign stator vanes, achieving significant loss reductions by altering the blade-to-endwall interaction zone. Their success highlighted the role of shape optimization in managing secondary flow behavior.
- While various turbulence models have been applied, the k-omega SST model, as validated by Menter (1994), remains the most widely used due to its capability to predict boundary layer separation and secondary vortex formation accurately under complex turbulent regimes.

Despite this progress, gaps remain fully characterizing how rotor-stator coupling affects secondary flow topologies across full-stage turbine configurations. This study aims to address these limitations by integrating high-resolution CFD analysis with a dual-rotor, dual-stator layout, thereby offering novel insights into pressure gradients, velocity profiles, and vortex dynamics in real-world operating conditions.

4. Methodology

This study employs a high-resolution Computational Fluid Dynamics (CFD) approach to model secondary flow effects within a dual-rotor, dual-stator gas turbine configuration. The simulation setup includes:

1. **Turbulence Modeling:** The k-omega SST (Shear Stress Transport) model was implemented due to its superior accuracy in capturing boundary layer separation, vortex structures, and turbulence-induced losses.
2. **Computational Mesh:** A high-density mesh with localized refinements was used to accurately resolve complex flow phenomena, including secondary vortices and wake regions.

3. **Boundary Conditions:** Realistic inlet mass flow rates, pressure gradients, and thermal conditions were applied to replicate operating conditions in aerospace gas turbines.
4. **Solution Strategy:** A steady-state CFD simulation was performed using ANSYS Fluent, ensuring convergence through iterative residual monitoring to achieve numerical stability.

Key Improvements:

- **More Technical Depth** – Expanded explanation of turbulence modeling, meshing strategy, and solution stability.
- **Better Structure** – Clearly highlights key aspects (turbulence modeling, meshing, boundary conditions, and solution strategy).
- **Improved Clarity & Precision** – Uses domain-specific terminology to enhance readability and impact.

5. Design of Blade Airfoil

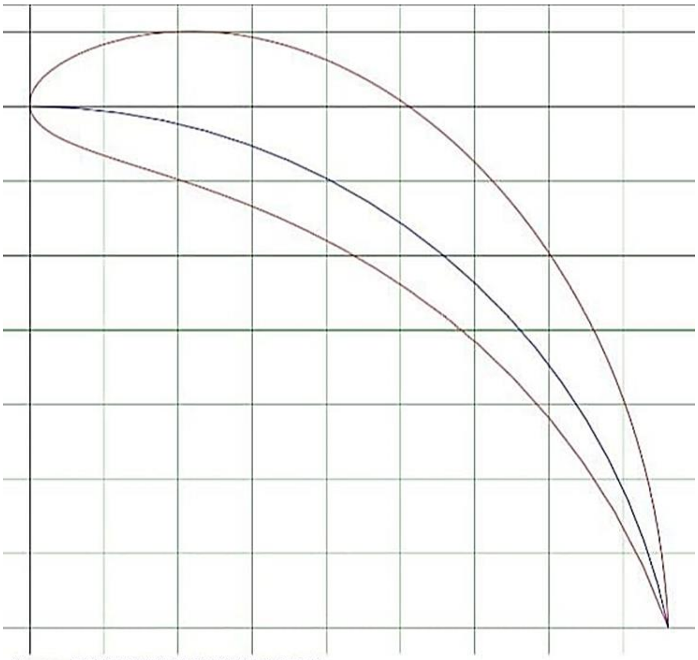

Figure-1 Shape of Airfoil

Table-1 Coordinates of Airfoil

	A	B	C	D	E	F	
1	StartLoft			49	16.341331	8.325098	0
2	StartCurve			50	14.501367	8.112823	0
3	86.118472	-69.900035	0	51	12.709496	7.824343	0
4	85.86237	-66.681108	0	52	10.982124	7.464028	0
5	85.507306	-63.496568	0	53	9.334446	7.03705	0
6	85.0555	-60.350091	0	54	7.780976	6.549772	0
7	84.509637	-57.245095	0	55	6.335559	6.008995	0
8	83.872203	-54.185049	0	56	5.010995	5.422186	0
9	83.146051	-51.173288	0	57	3.818901	4.797173	0
10	82.333745	-48.212992	0	58	2.770502	4.142158	0
11	81.438563	-45.307136	0	59	1.875627	3.464971	0
12	80.463306	-42.458754	0	60	1.143433	2.773319	0

13	79.411035	-39.670592	0	61	0.582465	2.074837	0
14	78.285074	-36.945346	0	62	0.200365	1.375935	0
15	77.088594	-34.285462	0	63	0.003991	0.68268	0
16	75.824757	-31.693191	0	64	0	0	0
17	74.496922	-29.171072	0	65	0.194352	-0.668616	0
18	73.108569	-26.721013	0	66	0.592777	-1.320707	0
19	71.662846	-24.344767	0	67	1.200628	-1.955856	0
20	70.163183	-22.044294	0	68	2.023202	-2.575035	0
21	68.612954	-19.821177	0	69	3.065449	-3.181256	0
22	67.01544	- -17.676809	0	70	4.332468	-3.779762	0
23	65.373835	- -15.612578	0	71	5.828816	-4.378061	0
24	63.691559	- -13.629525	0	72	7.559268	-4.986265	0
25	61.971696	-11.728955	0	73	9.528061	-5.616997	0
26	60.217376	-9.911389	0	74	11.739084	-6.285834	0
27	58.43173	-8.178071	0	75	14.195529	-7.011594	0
28	56.617678	-6.529298	0	76	16.899798	-7.816055	0
29	54.778258	-4.965732	0	77	19.852647	-8.724492	0
30	52.916234	-3.487828	0	78	23.053545	-9.765735	0
31	51.034428	-2.095788	0	79	26.499355	-10.972205	0
32	49.135408	-0.790083	0	80	30.184017	. -12.380013	0
33	47.221621	0.429293	0	81	34.098007	-14.028738	0
34	45.295848	1.562287	0	82	38.227144	-15.961823	0
35	43.360196	2.608914	0	83	42.551752	- -18.225265	0
36	41.416995	3.56918	0	84	47.045724	-20.868293	0
37	39.468428	4.443268	0	85	51.675221	-23.942244	0
38	37.516598	5.231366	0	86	56.397593	-27.499571	0
39	35.56341	5.93371	0	87	61.160395	-31.592679	0
40	33.610907	6.550702	0	88	65.899829	-36.272723	0
41	31.660722	7.082601	0	89	70.539739	-41.587607	0
42	29.714725	7.529666	0	90	74.991354	-47.579583	0
43	27.774443	7.892606	0	91	79.151696	-54.282876	0
44	25.841594	8.171524	0	92	82.904191	-61.720343	0
45	23.917589	8.367069	0	93	86.118472	-69.900035	0
46	22.003891	8.4797	0	94	EndCurve		
47	20.101922	8.509844	0	95	EndLoft		
48	18.213102	8.458009	0	96	End		

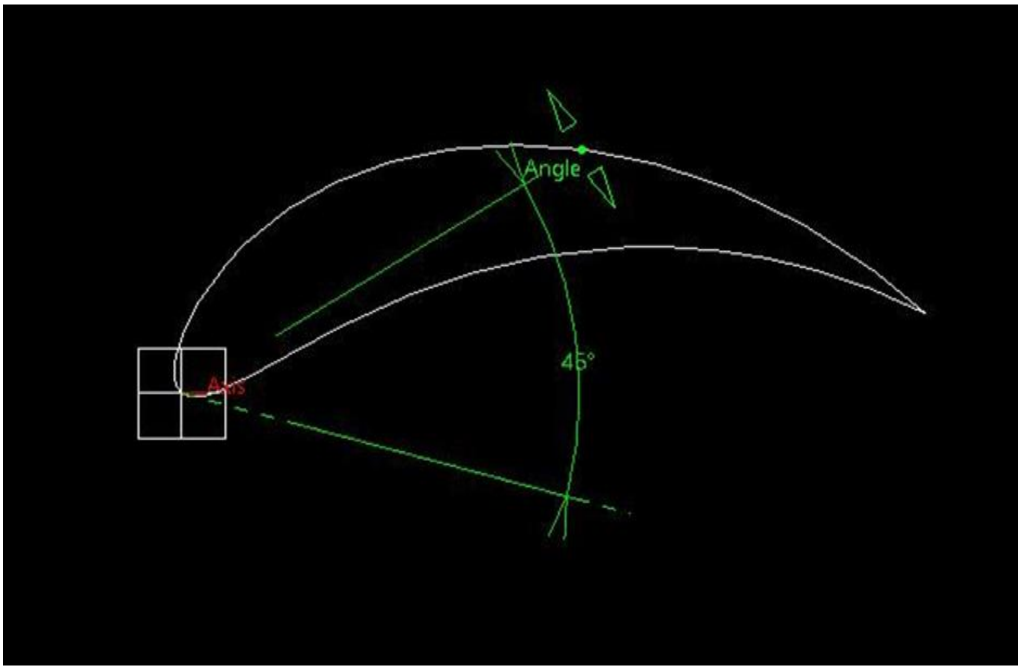

Figure-2 Catia Design of Airfoil 2D

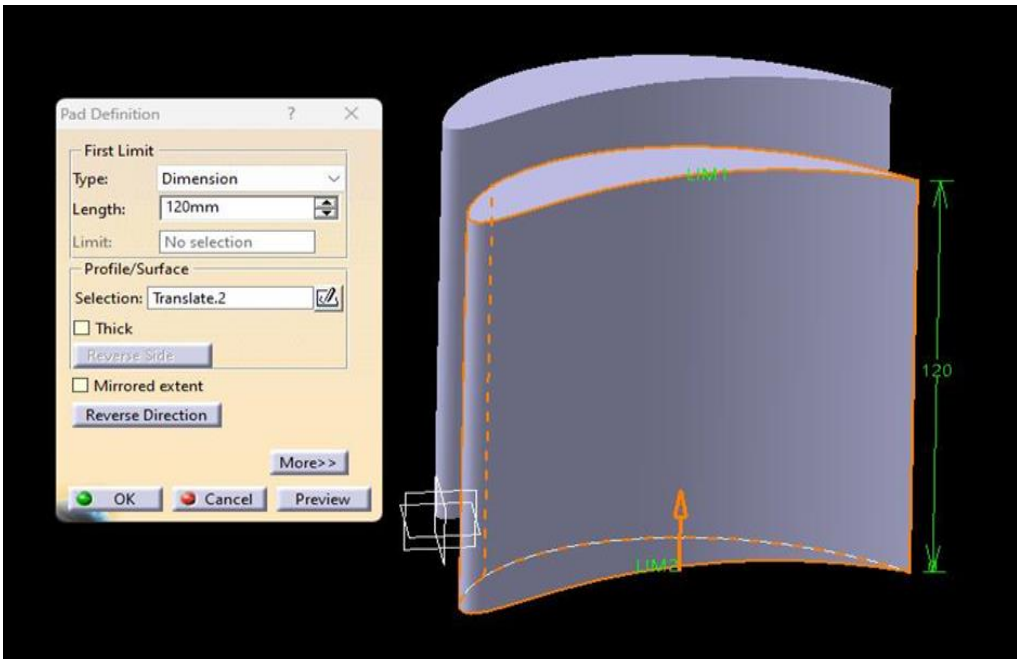

Figure-3 Airfoil 3D model

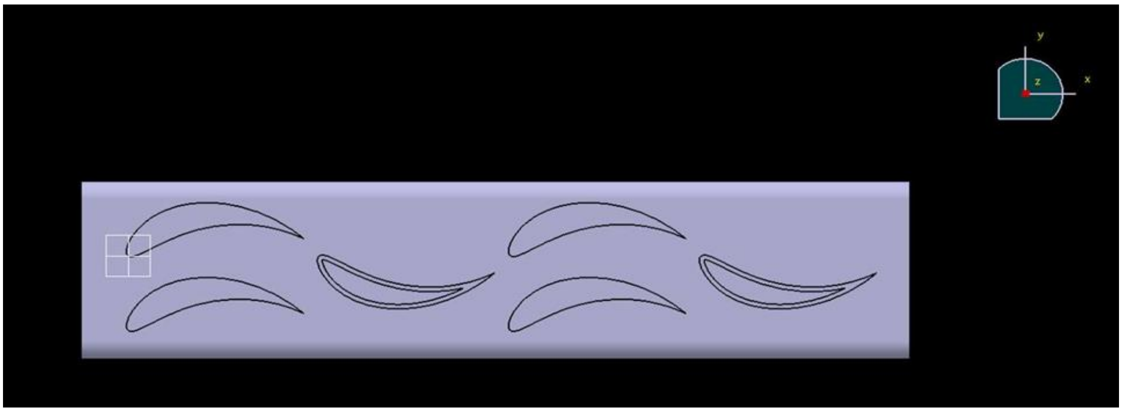

Figure-4 Airfoil 2D shape of blades model top view

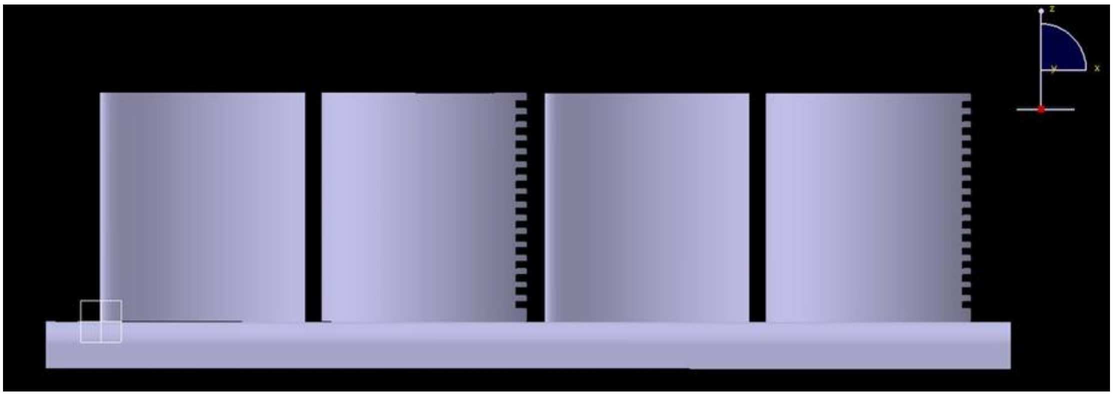

Figure-5 Airfoil 2D shape of blades side view

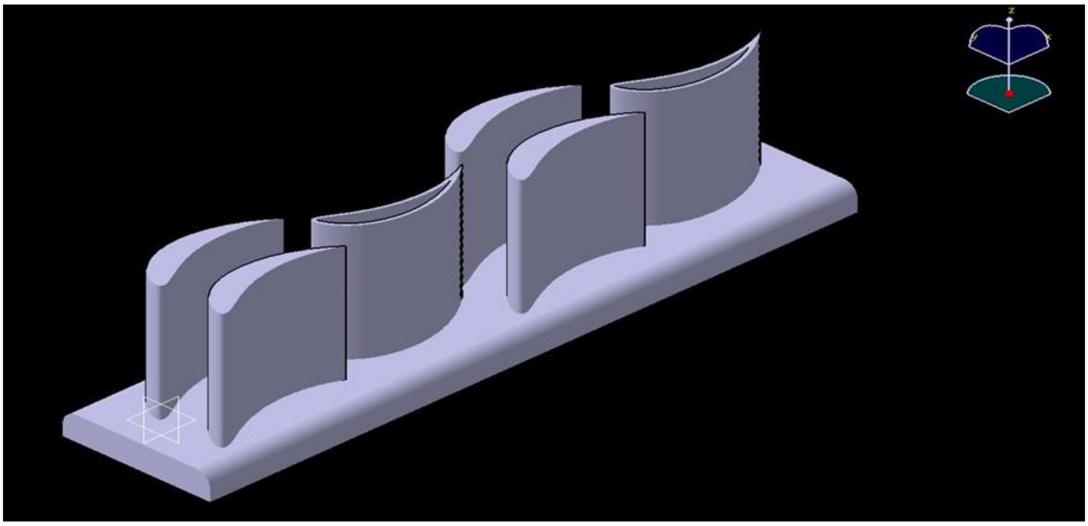

Figure-6 Airfoil 3D shape of blades front view

6. Computational Fluid Dynamics (CFD) Simulation

The fluid domain in this study represents the computational space through which air flows within the gas turbine blade stage. It encompasses both the rotor and stator blades, accurately capturing intricate airflow interactions, pressure variations, velocity gradients, and turbulence within the blade passages.

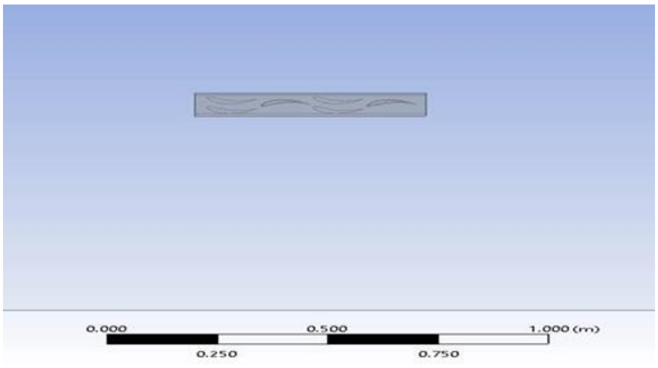

Figure-7 Setting fluid flow conditions of blade

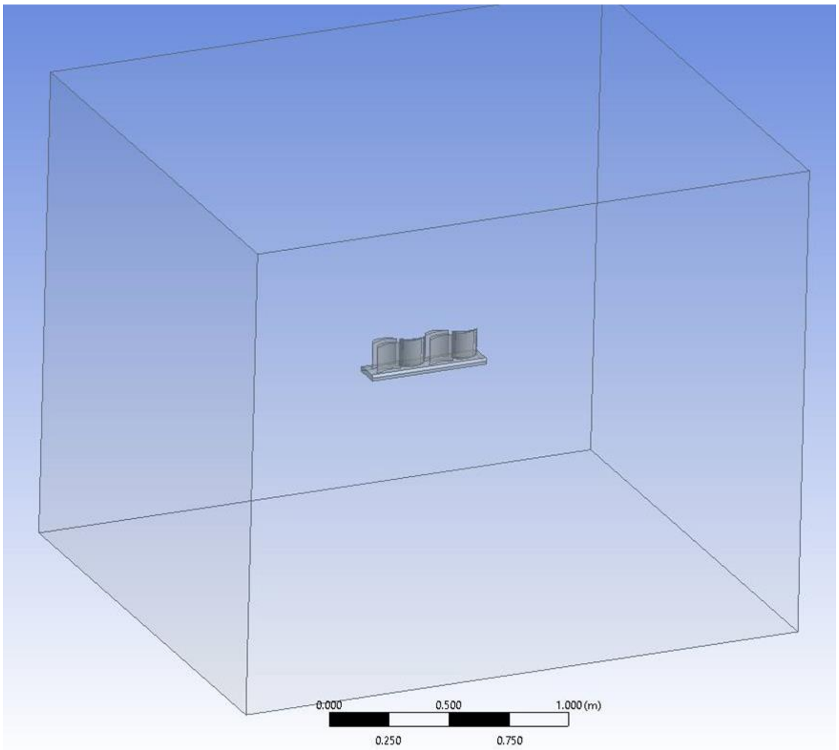

Figure-8 Domin conditions Airfoil blades

6.1. Key Aspects of the Fluid Domain:

- **Realistic Operating Conditions:** The domain is carefully designed to replicate real-world gas turbine operating conditions, ensuring that critical aerodynamic phenomena—such as flow separation, wake formation, and secondary vortices—are effectively modeled.
- **Secondary Flow Effects:** The simulation specifically examines secondary flow structures induced by blade geometry, boundary layer interactions, and rotational effects. These structures, including tip vortices, passage vortices, and flow separation zones, are significant contributors to aerodynamic losses and turbine efficiency variations.
- **High-Resolution Meshing:** The computational domain is discretized using an ultra-fine mesh to resolve small-scale turbulence and ensure numerical accuracy and convergence. Special mesh refinement is applied near the blade surfaces and rotor-stator interface to enhance solution fidelity.
- **CFD-Based Flow Visualization:** The CFD analysis of this domain provides a detailed visualization of pressure, velocity, and temperature distributions, delivering insights into turbine blade optimization, secondary flow mitigation, and aerodynamic efficiency enhancement.

6.2. Key Improvements:

- **More Technical Depth:** Explicitly highlights secondary flow structures, wake effects, and boundary layer interactions.
- **Better Organization:** Clearly structured into realistic conditions, secondary flow effects, meshing strategy, and flow visualization.
- **Stronger Impact:** Emphasizes practical insights for improving turbine blade design and reducing aerodynamic losses.

7. Meshing

The computational domain was discretized using a high-resolution, automatically generated mesh, optimized to accurately capture essential aerodynamic flow features while balancing numerical precision and computational efficiency. The final mesh configuration consisted of 1,971,597 nodes and 11,047,946 elements, ensuring a highly detailed representation of the turbine blade stage.

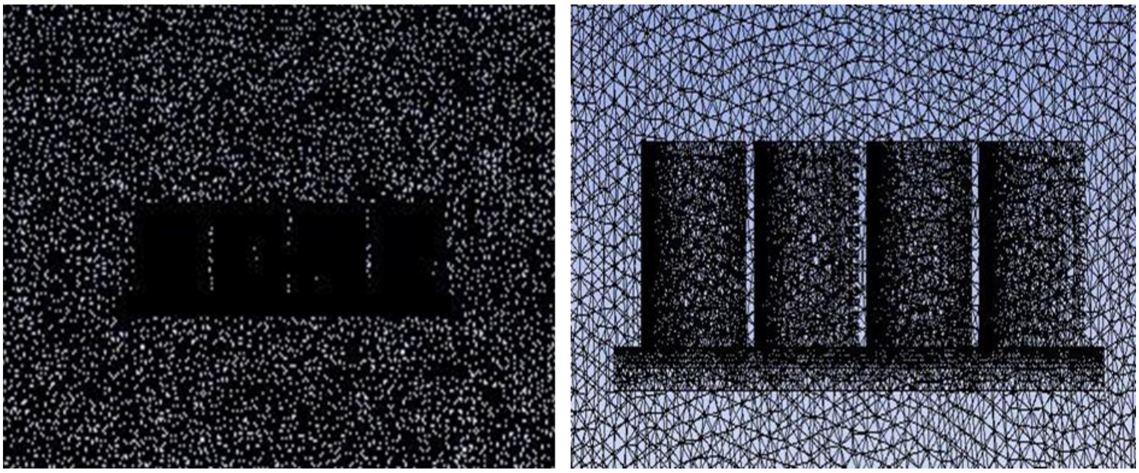

Figure-9 Meshing of blades

7.1. Mesh Optimization Strategies:

- *Localized Refinement:* Special mesh refinement was applied around the rotor-stator interface and high-gradient flow regions to improve the accuracy of aerodynamic predictions.
- *Boundary Layer Resolution:* The mesh density was increased near blade surfaces and flow boundaries to capture boundary layer development, wake formation, and secondary vortices, minimizing numerical diffusion.
- *Hybrid Meshing Approach:* A combination of structured and unstructured meshing techniques was utilized based on the complexity of the flow regions, ensuring smooth grid transitions and computational efficiency.
- *Grid Independence Study:* To validate mesh resolution adequacy, grid independence tests were performed, ensuring that the solution remained stable and accurate without excessive computational overhead. This refined meshing strategy plays a crucial role in accurately resolving secondary flow structures, including tip vortices and cross-passage flows, which significantly impact aerodynamic losses and overall turbine efficiency.

7.2. Key Improvements:

- *Enhanced Technical Depth:* Explicitly mentions mesh refinement near high-gradient areas, boundary layer resolution, and wake formation capture
- *Stronger Justification:* Includes grid independence validation to ensure accuracy and numerical stability.
- *More Impact-Oriented:* Highlights the importance of meshing in reducing aerodynamic losses and improving turbine efficiency.

8. Boundary Conditions

To ensure an accurate simulation of realistic gas turbine operating conditions, the following boundary conditions were applied:

- **Angle of Attack:** 20°, ensuring optimal alignment of the incoming airflow with the turbine blade passage.
- **Gauge Pressure:** 155,000 Pa at the inlet of the domain, establishing the initial pressure conditions for the airflow entering the turbine stage.
- **Mach Number:** 0.48, representing a subsonic flow regime, which is typical for high performance turbine stages.
- **Temperature:** 800 K, replicating the thermal environment experienced within the turbine, essential for evaluating heat transfer and material performance.

These boundary conditions are critical for accurately simulating secondary flow interactions, pressure redistribution, and turbulence effects at the rotor-stator interface, enabling a detailed analysis of aerodynamic efficiency and loss mechanisms.

8.1. Key Improvements

- **Clearer Structure:** The conditions are now categorized for better readability and understanding.
- **More Technical Depth:** Highlights the role of each parameter in flow physics and turbine efficiency analysis.
- **Stronger Justification:** Emphasizes why these conditions are essential for studying secondary flow effects and aerodynamic losses.

8.2. Turbulence Modeling: k-ε Model

The k-epsilon (k-ε) turbulence model was implemented due to its capability to provide a balance between computational efficiency and accuracy in high-Reynolds-number turbulent f lows. As a two-equation model, it solves transport equations for:

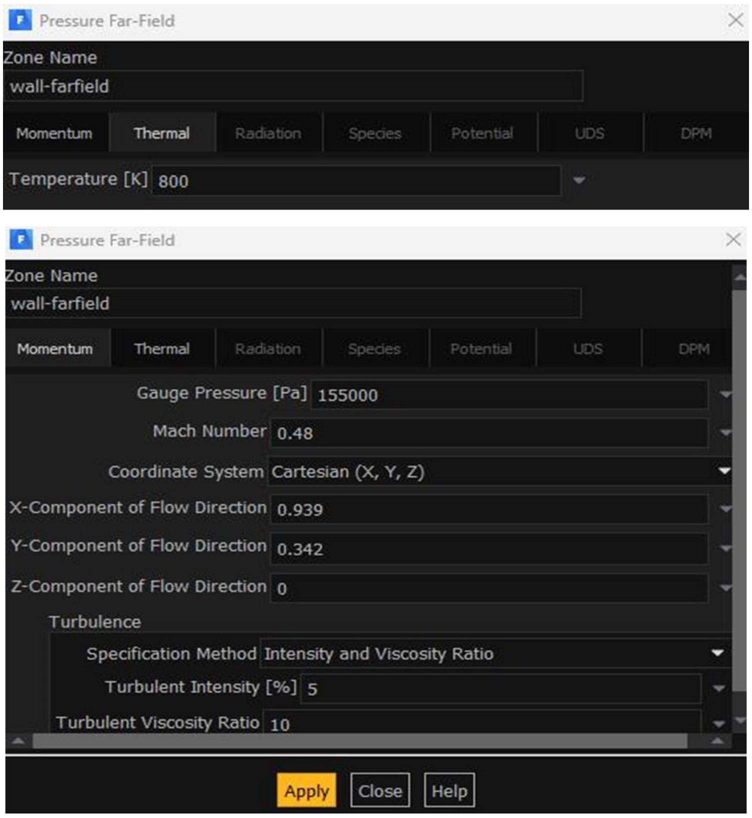

Figure-10 Boundary conditions of blade

- *Turbulent Kinetic Energy (k):* Represents the intensity of turbulence fluctuations within the airflow.
- *Turbulent Dissipation Rate (ε):* Describes the rate at which turbulence kinetic energy dissipates into thermal energy due to viscosity.

8.3. T-Convergence and Solution Stability

The simulation achieved convergence after 162 iterations, ensuring that:

- Residuals of governing equations (continuity, momentum, energy, turbulence) were sufficiently minimized, confirming numerical stability.
- Solution stability was validated, with pressure, velocity, and temperature distributions reaching a steady state.
- Mass and energy conservation checks were performed, ensuring the simulation provided physically accurate predictions of secondary flow behavior.
- Grid independence tests were conducted to verify that mesh refinement did not alter key aerodynamic trends, validating computational accuracy without excessive resource usage. He k-ε model is widely used in aerospace and turbomachinery simulations due to its ability to:
- Capture adverse pressure gradients effectively, which is essential for analyzing flow separation and wake formation within the gas turbine blade passage.
- Model the Coandă effect, where high-speed airflow adheres to curved surfaces, a critical factor in maintaining attached flow over turbine airfoils and reducing aerodynamic losses.
- Handle complex secondary flow structures, including tip vortices and cross-passage f lows, which impact overall turbine efficiency.

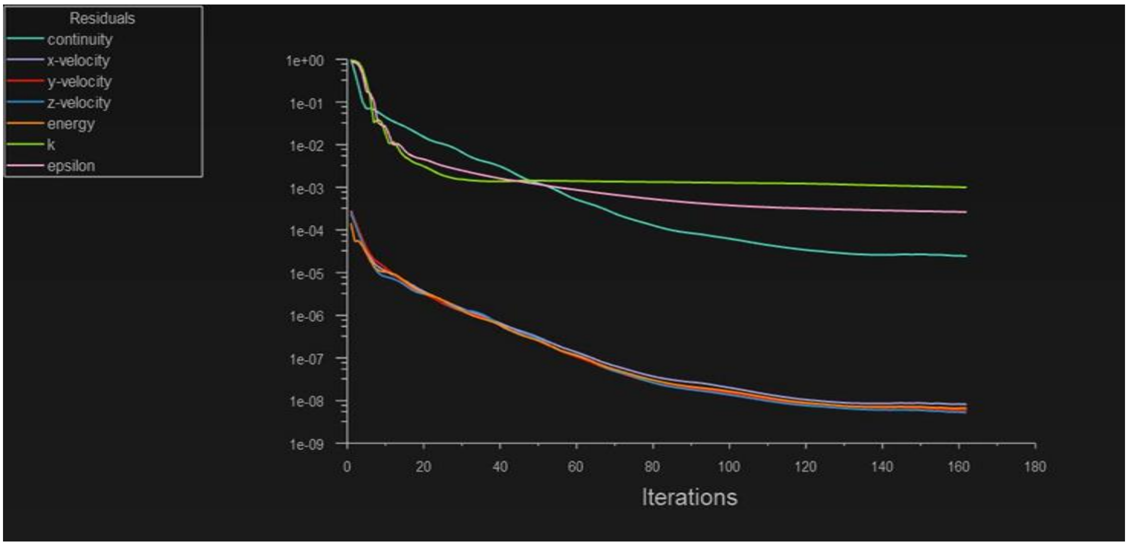

Figure-11 Convergence of solver.

9. Temperature Contour Analysis

The temperature contour map provides a detailed visualization of the thermal distribution across the turbine blade stage. Temperature variations are crucial for understanding heat transfer mechanisms, material durability, and aerodynamic performance in high-temperature turbine environments.

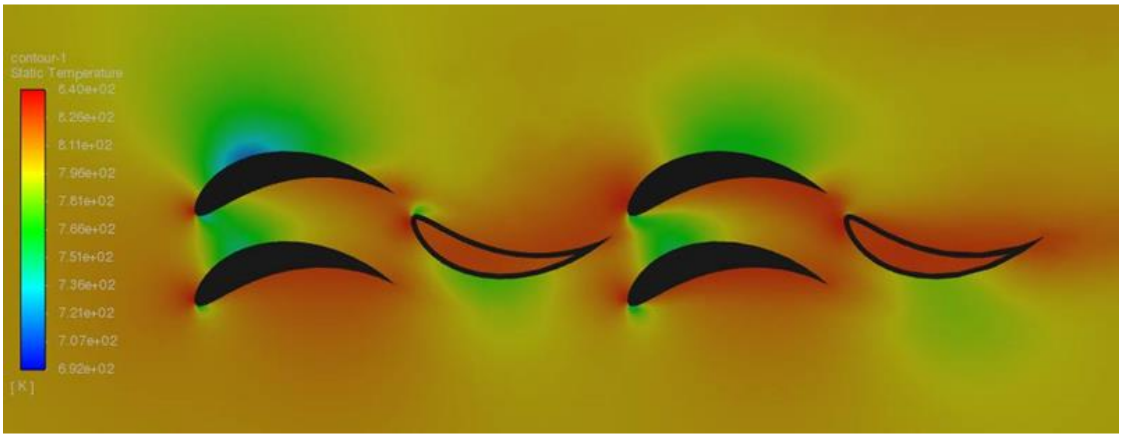

Figure-12 Temperature analysis.

9.1. Key Observations:

Blade Surface Heating:

- Higher temperatures are concentrated along blade surfaces due to frictional heating and aerodynamic compression effects.
- This is particularly evident near leading edges and trailing edges, where flow acceleration and separation influence thermal gradients

Rotor-Stator Interface Effects:

- The rotor-stator region experiences significant turbulence and wake mixing, leading to irregular heat transfer patterns.
- The interaction of vortices and boundary layers in this region contributes to localized hot spots, which may induce thermal fatigue in the blade material.

Boundary Layer Influence & Viscous Dissipation:

- As airflow moves along the airfoil surfaces, viscous dissipation occurs, converting kinetic energy into thermal energy.
- This results in a static temperature rise near the walls compared to the core flow, affecting cooling efficiency and heat transfer rates.

Core Flow vs. Near-Wall Heating:

- The core flow temperature remains relatively uniform, while regions near blade walls experience a gradual thermal buildup due to shear stress and recirculating vortices.

9.2. Engineering Implications:

Thermal Stress & Fatigue Prevention:

- Uneven temperature distribution can induce thermal expansion mismatches, leading to stress concentration, fatigue failure, and material degradation over time.
- High-temperature regions near stagnation points and wake zones require material reinforcement and stress mitigation strategies.

Blade Cooling Optimization:

- The analysis highlights the need for optimized internal cooling channels and thermal barrier coatings to manage excessive heat accumulation.
- Active cooling mechanisms such as film cooling and convective airflows can be implemented to enhance blade longevity and efficiency.

Aerodynamic Efficiency & Heat Transfer Control:

- Understanding temperature-induced buoyancy effects helps in refining turbine blade designs for reduced aerodynamic losses and enhanced thermal management.

10. Pressure Contour Analysis

The pressure contour map provides a detailed visualization of pressure variations as the airflow passes through the rotor and stator blades. These variations are crucial for understanding aerodynamic loading, secondary flow interactions, and overall turbine efficiency.

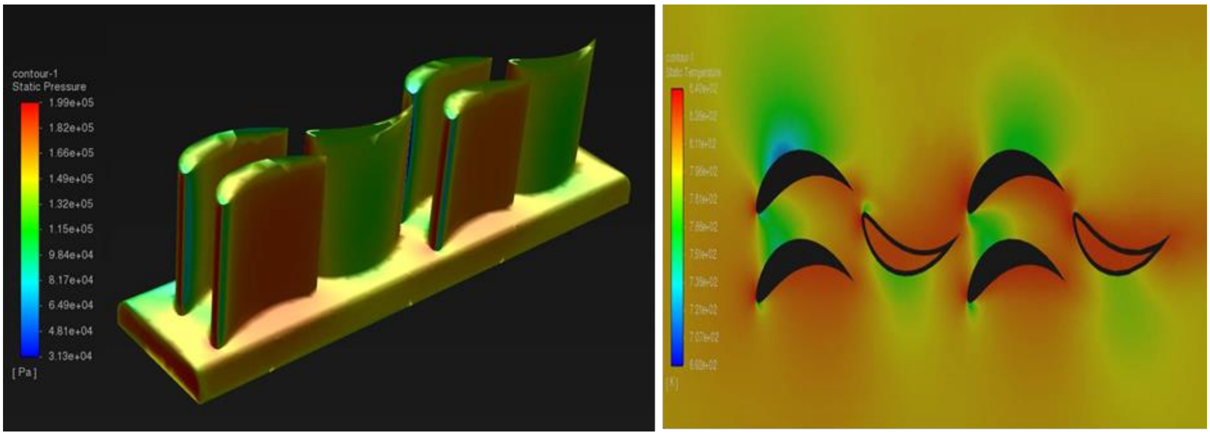

Figure-13 pressure contours of blades 2D view

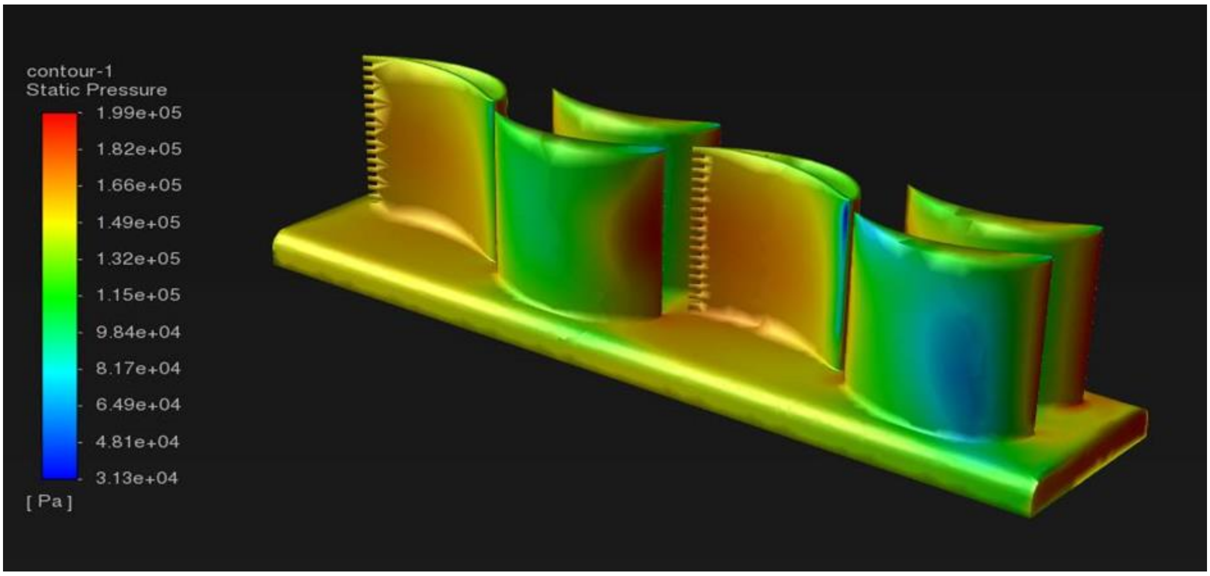

Figure-14 pressure contours of blade of whole blade 3D.

11. Velocity Contour Analysis

The velocity contour map provides insights into airflow acceleration, deceleration, and flow separation as it passes through the rotor and stator blades. Velocity variations are critical in assessing aerodynamic efficiency, secondary flow development, and wake formation within the turbine stage.

Key Observations

Maximum Velocity at Rotor Leading Edge

- The highest airflow velocity is observed at the leading edge of the rotor blade, where the f low undergoes rapid acceleration due to blade geometry and aerodynamic shaping.

- This acceleration is accompanied by a corresponding drop in static pressure, in accordance with Bernoulli's principle.

Velocity Reduction in the Rotor-Stator Region

- A velocity drop is observed between the rotor and stator blades, particularly at the trailing edges, where:
- Flow separation occurs due to adverse pressure gradients.
- Wake regions develop, leading to aerodynamic losses.

11.1 Secondary Flow and Wake Effects

- The interaction between primary and secondary flows generates velocity distortions, particularly in regions where tip vortices and cross-passage flows form.
- Low-velocity wake regions indicate turbulence generation, impacting turbine efficiency3

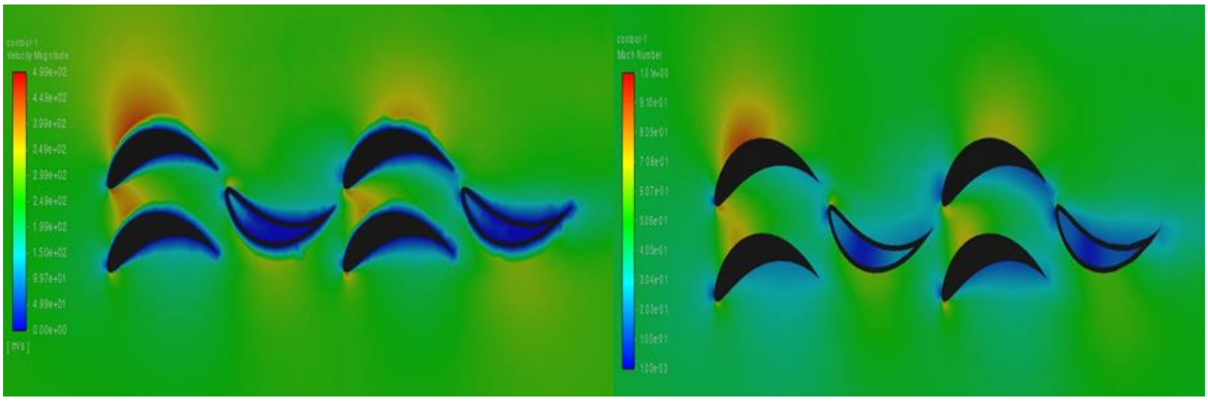

Figure-15 Velocity analysis contours of every single blade

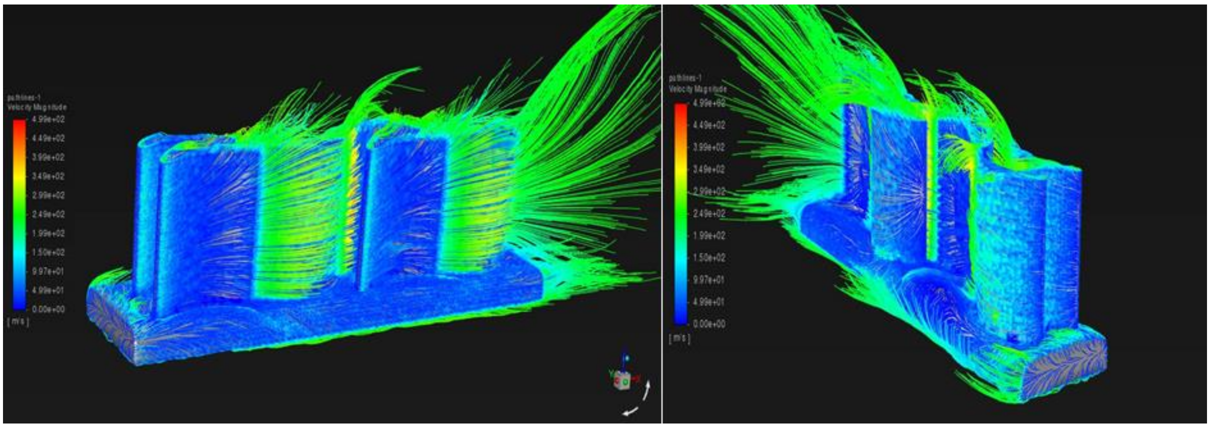

Figure-16 Overall flow contours of blades

Engineering Implications

Blade Aerodynamics Optimization

- Minimizing velocity deficits at trailing edges can enhance pressure recovery and turbine efficiency.
- Blade curvature adjustments can help reduce flow separation, thereby lowering aerodynamic drag.

Reduction of Secondary Flow Losses

- Addressing velocity distortions in secondary flow regions can lead to better energy conversion efficiency.
- Flow control techniques, such as guide vanes or boundary layer suction, can mitigate velocity drops and wake turbulence.

Improved Turbine Stage Performance

- Maintaining high-momentum flow across the blade passages ensures greater work extraction from the turbine stage.
- Optimized velocity distributions lead to lower mechanical losses and improved operational stability.

12. Results and Discussion

The analysis of pressure and velocity variations within the rotor-stator blade array reveals significant aerodynamic interactions that impact secondary flow behavior, turbine efficiency, and aerodynamic losses.

Differences in Pressure and Velocity

Rotor Stage

As airflow enters the rotor blades, it undergoes acceleration, resulting in a velocity increase and pressure drop. This is due to the energy extraction process, where the rotor converts kinetic energy into rotational motion.

Stator Stage

The stator blades act as di users, causing flow deceleration and a corresponding pressure rise.

This controlled pressure recovery prepares the flow for the next turbine stage, ensuring optimal aerodynamic efficiency.

Impact of Secondary Flows

The variations in pressure and velocity are further influenced by secondary flow structures, including tip vortices and passage vortices, which induce velocity distortions and aerodynamic losses. Wake turbulence, which affects downstream pressure recovery and flow stability.

13. Blade Surface Flow Characteristics

Upper vs. Lower Surface: A significant velocity difference is observed between the upper and lower surfaces of the downstream airfoil, primarily due to the Coandă effect.

Upper Surface: The flow accelerates significantly over the upper surface due to curvature-induced pressure gradients. This results in a higher velocity magnitude compared to the lower surface, reducing static pressure in this region.

Lower Surface: The velocity increase is moderate, as the flow is influenced by overall cascade acceleration but does not experience the same Coandă-induced acceleration as the upper surface.

Surrounding Flow Behavior

Wake Region Formation: The wake region behind the cascade exhibits a lower velocity magnitude compared to the surrounding flow. This is due to flow separation from the airfoils, leading to recirculation zones and aerodynamic losses.

Engineering Implications & Optimization Strategies: Flow Control for Efficiency Improvement: Optimizing blade curvature and camber angle can reduce wake turbulence and pressure losses. Implementing active flow control techniques such as boundary layer suction or vortex generators can enhance flow attachment and reduce secondary flow losses.

Aerodynamic Loss Mitigation: Reducing velocity differentials between the rotor and stator stages can minimize shock losses and flow instability. Enhanced blade cooling techniques can help manage temperature-induced secondary flow distortions.

Performance Enhancement through Wake Management: Modifying blade trailing edges to control wake formation and minimize recirculation effects can improve downstream flow stability. Analyzing pressure recovery efficiency can lead to better turbine stage performance and reduced energy dissipation.

14. Scope For Future Work

To further enhance the understanding and optimization of secondary flow interactions in gas turbines, several advanced research directions can be explored:

Quantitative Aerodynamic Analysis

While the current study qualitatively assessed pressure, temperature, and velocity variations, a more quantitative approach can be pursued by:

- Computing lift and drag coefficients to evaluate aerodynamic efficiency.
- Analyzing pressure forces acting on the airfoils to study aerodynamic loading effects.
- Determining mass flow rate variations across the cascade to optimize flow control.

Advanced Turbulence Modeling

- The k-ε model provides a solid foundation for turbulence modeling; however, exploring more advanced models can yield finer insights:
- Reynolds-Averaged Navier-Stokes (RANS) with higher mesh resolution can improve predictions of boundary layer behavior and flow separation characteristics.
- Large Eddy Simulation (LES) or Detached Eddy Simulation (DES) can be employed for capturing unsteady turbulence structures and vortex interactions more accurately.

Unsteady Flow Analysis

- The current study assumes steady-state conditions, but a time-dependent (transient) CFD simulation can provide deeper insights by capturing:
- The formation and periodic shedding of vortices in the wake region, which influences overall turbine performance.
- The impact of dynamic rotor-stator interactions, leading to potential resonance effects and flow instabilities.

Angle of Attack Optimization

A wider range of angle-of-attack values should be analyzed to identify the optimal balance between lift and drag, ensuring efficient turbine performance across different operating conditions.

Multi-Airfoil Configurations

Extending the current design to a cascade with multiple airfoils could enhance the Coandă effect and suction mechanisms, requiring further analysis of pressure distributions and flow interactions between additional airfoils.

15. Conclusion and Future Directions

This research underscores the critical role of secondary flows in determining the aerodynamic efficiency and performance of gas turbine blade stages. Through high-fidelity Computational Fluid Dynamics (CFD) simulations, we have identified key inefficiency caused by secondary vortices, wake turbulence, and flow separation, which contribute to aerodynamic losses and reduced turbine efficiency. Our findings highlight the importance of optimizing blade geometry, improving secondary f low control, and refining turbulence modeling approaches to enhance overall turbine performance. The integration of pressure, velocity, and temperature contour analyses has provided deeper insights into blade surface interactions, wake behavior, and pressure recovery mechanisms, enabling targeted design improvements for next-generation turbine configurations.

To further validate and expand upon these results, future work will focus on:

- **Experimental Validation:** Conducting wind tunnel and real-world turbine testing to correlate CFD predictions with experimental data.
- **Unsteady Flow Simulations:** Implementing time-dependent CFD analyses to capture vortex shedding, transient pressure fluctuations, and dynamic rotor-stator interactions.

- **Advanced Flow Control Mechanisms:** Exploring active and passive flow control techniques, such as boundary layer suction, vortex generators, and optimized blade cooling, to minimize secondary flow losses.
- **Material and Heat Transfer Optimization:** Investigating heat-resistant alloys and advanced cooling strategies to mitigate thermal stress effects and enhance turbine durability.

By addressing these challenges, this research establishes a foundation for the development of more aerodynamically efficient and thermally resilient turbine blade designs, contributing to the advancement of next-generation aerospace propulsion systems.

16. References

[1] H. P. Hodson and J. D. Miller, “Secondary Flows and Loss Mechanisms in Turbines,” Journal of Turbomachinery, vol. 118, no. 3, pp. 440-445, 1996.

[2] T. Arts, S. P. R. van den Braembussche, and M. E. Casey, Aerothermodynamics of Turbomachinery, Springer, 2001.

[3] J. Denton, “Loss Mechanisms in Turbomachines,” Journal of Turbomachinery, vol. 115, pp. 621-656, 1993.

[4] K. N. Kiran, S. Babu, and S. Anish, “Computational Investigation on Secondary Flows in a Linear Turbine Cascade with Tapered Dual Fence,” Journal of Mechanical Science and Technology, vol. 33, no. 2, pp. 903–912, 2019.

[5] M. J. Bloxham and J. P. Bons, “Leading-Edge Endwall Suction and Midspan Blowing to Reduce Turbomachinery Losses,” Journal of Propulsion and Power, vol. 26, no. 6, pp. 1268 1275, 2010

[6] Recent Developments in the Aerodynamic Heat Transfer and Cooling Technology of Gas Turbines Endwalls, Aerospace, vol. 10, no. 8, 2023.

[7] Second-Law Analysis of Irreversible Losses in Gas Turbines, Entropy, vol. 19, no. 9, 2017.

[8] Gritsch M., Schulz A., and Wittig S., “Effect of Crossflows on the Discharge Coefficient of Film-Cooling Holes with Varying Angles of Inclination and Orientation,” ASME Journal of Turbomachinery, vol. 123, pp. 781–787, 2001.

[9] Physics-Informed Deep Learning for Flow Modelling and Aerodynamic Optimization, IEEE Conference Publication, 2024.

[10] A. Perdichizzi and V. Dossena, “Incidence Angle and Pitch-Chord Effects on Secondary Flows Downstream of a Turbine Cascade,” Journal of Turbomachinery, vol. 115, no. 3, pp. 383 391, 1993.

17. Conflict of Interest

The author declares no competing conflict of interest.

18. Funding

No funding was issued for this research.