Turbomachinery

3D Simulation of Rotor Blade 37

Shampreethi Satish Prabu

Hashwek TR

Contents

1 Abstract

This report presents a computational fluid dynamics (CFD) study of Rotor 37, a transonic axial-flow fan rotor, using a single-blade passage model. The primary objective is to simulate the aerodynamic behavior of the rotor under simplified conditions and assess basic flow parameters for model validation. The simulation employs the steady-state Reynolds-Averaged Navier–Stokes (RANS) equations, closed with the k–ω turbulence model, which is suitable for capturing near-wall effects and flow separation.

The computational domain is discretized using a structured mesh with near-wall refinement, achieving a dimensionless wall distance of approximately $y^+ \approx 1$. Boundary conditions include a total pressure of 101,000 Pa and a total temperature of 288 K at the inlet, with a static pressure imposed at the outlet. A rotating reference frame is used to replicate the operational conditions of the rotor.

The simulation results focus on convergence behavior and mass flow consistency. Residuals of continuity, momentum, and turbulence equations dropped below 10^{-5}, indicating numerical convergence. Additionally, inlet and outlet mass flow rates were recorded and compared to evaluate conservation and solution stability.

2 Introduction

Turbomachinery is central to many modern engineering systems, particularly in aerospace propulsion and energy generation. The aerodynamic design and analysis of turbomachinery blades are critical to enhancing efficiency, minimizing losses, and ensuring structural integrity. Among the components, rotor blades play a key role in imparting energy to the fluid, especially under transonic flow conditions where shock waves and complex three-dimensional flow structures dominate.

Rotor 37, a transonic axial-flow fan rotor developed by NASA, has become a benchmark test case for validating computational fluid dynamics (CFD) techniques due to its well-documented geometry and available experimental data. It exhibits challenging aerodynamic features such as shock-wave/boundary-layer interactions, tip leakage flow, and flow separation—making it an ideal candidate for performance evaluation and turbulence modeling validation.

In this study, a steady-state CFD simulation is performed on a single blade passage of Rotor 37 using a structured mesh. The computational model employs the three-dimensional Reynolds-Averaged Navier–Stokes (RANS) equations closed with the k–ω turbulence model, which provides reliable near-wall treatment and accurate prediction of flow separation. A high-quality structured mesh with wall-resolved cells ensures a near-wall $y^+ \approx 1$, capturing boundary layer effects with sufficient accuracy.

The inlet boundary condition is specified by a total pressure of 101,000 Pa and a total temperature of 288 K, while a static pressure is applied at the outlet. The simulation is conducted in a rotating reference frame representing the operational speed of the blade. The objective of this work is to analyze the aerodynamic behavior of Rotor 37 under these conditions, extract performance metrics such as pressure ratio and efficiency, and gain insight into internal flow structures.

3 Problem Formulation

3.1 Governing Equations

The flow through the Rotor 37 blade passage is modeled using the three-dimensional, steady-state Reynolds-Averaged Navier–Stokes (RANS) equations, which govern the conservation of mass, momentum, and energy. These equations are expressed as:

- **Continuity Equation:**

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0 \tag{1}$$

- **Momentum Equation:**

$$\frac{\partial (\rho \vec{v})}{\partial t} + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot \boldsymbol{\tau} + \rho \vec{f} \tag{2}$$

- **Energy Equation:**

$$\frac{\partial (\rho E)}{\partial t} + \nabla \cdot [(\rho E + p)\vec{v}] = \nabla \cdot (k \nabla T) + \Phi \tag{3}$$

Where ρ is the density, $\vec{v}$ is the velocity vector, p is the pressure, $\boldsymbol{\tau}$ is the viscous stress tensor, $\vec{f}$ is the body force, E is the total energy per unit mass, k is the thermal conductivity, T is the temperature, and Φ represents the viscous dissipation function.

3.2 Turbulence Modeling

To close the RANS equations, a turbulence model is required. In this study, the **k–ω Shear Stress Transport (SST)** model is employed due to its robustness in capturing near-wall effects and accurately predicting separation in turbomachinery flows. The SST model combines the advantages of the standard k–ϵ model in the free stream and the k–ω model near the wall regions.

3.3 Boundary Conditions

The computational domain includes the blade passage of Rotor 37 with periodic boundaries between passages. The following boundary conditions are applied:

- **Inlet:** Total pressure and total temperature are specified, along with the flow direction.

- **Outlet:** Static pressure is imposed to match the experimental operating condition.

- **Blade, Hub, and Shroud Walls:** No-slip and adiabatic wall conditions are applied.

- **Rotating Frame:** The domain is set in a rotating reference frame with a rotational speed of 17188.7 RPM, corresponding to the design condition of Rotor 37.

- **Periodic Boundaries:** Periodic conditions are applied in the circumferential direction to simulate a full annulus using a single blade passage.

4 Numerical Methods

4.1 Discretization and Solver Settings

The governing equations are discretized using the Finite Volume Method (FVM), which ensures conservation of mass, momentum, and energy over each control volume. A second-order upwind scheme is applied for the convective terms to achieve higher accuracy in capturing shock waves and flow gradients. Pressure-velocity coupling is handled using the SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm.

Temporal discretization is omitted as the simulation is steady-state. The solution is initialized with uniform flow values and advanced iteratively until residuals fall below 10^{-5} for continuity and momentum equations, and monitor points such as mass flow rate and pressure ratio converge to steady values.

5 Mesh Details and Near-Wall Resolution

A structured hexahedral mesh was generated for the single blade 37 model to ensure high quality and efficient computational analysis. The mesh was carefully refined in the near-wall region to accurately capture the viscous sublayer and the turbulent boundary layer.

5.1 Near-Wall Treatment and y^+ Value

The non-dimensional wall distance, y^+, is a crucial parameter that characterizes the fineness of the mesh near solid walls in computational fluid dynamics (CFD) simulations involving turbulent flows. It is defined as:

$$y^+ = \frac{u_\tau y}{\nu} \tag{4}$$

where:

- $u_\tau = \sqrt{\frac{\tau_w}{\rho}}$ is the friction velocity at the wall.

- y is the distance from the wall to the first mesh cell center.

- ν is the kinematic viscosity of the fluid.

- τ_w is the wall shear stress.

- ρ is the fluid density.

The y^+ value dictates which region of the turbulent boundary layer the first computational node resides in. Different turbulence models have specific requirements or recommendations for the y^+ value to ensure accurate predictions:

- $y^+ \approx 1$: This value places the first mesh cell within the viscous sublayer ($y^+ < 5$). This approach is essential when using low-Reynolds number turbulence models or when employing wall functions that are designed to bridge the viscous sublayer. Resolving the viscous sublayer directly allows for a more accurate computation of the wall shear stress and heat transfer.

- $30 < y^+ < 100$: This range is typically targeted when using standard wall functions with high-Reynolds number turbulence models (e.g., standard $k-\epsilon$). Wall functions are semi-empirical formulas that bridge the near-wall region, avoiding the need to resolve the very fine viscous sublayer.

For this simulation of the single blade 37 model, the mesh was generated and refined to achieve a y^+ value of approximately 1 near the blade surface. This indicates that the first computational nodes are located well within the viscous sublayer. This fine near-wall resolution is beneficial for the following reasons:

- **Accurate Wall Shear Stress Prediction:** By resolving the linear velocity profile within the viscous sublayer, the wall shear stress can be directly computed, leading to higher accuracy compared to using wall functions that rely on empirical correlations.

- **Improved Heat Transfer Prediction:** Similar to momentum transfer, resolving the near-wall temperature gradients is crucial for accurate heat transfer calculations. A $y^+ \approx 1$ ensures that these gradients are well-captured.

- **Suitability for Low-Reynolds Number Turbulence Models:** If a low-Reynolds number turbulence model was employed, a $y^+ \approx 1$ is a strict requirement for the model to function correctly and provide physically meaningful results.

The achievement of a structured mesh with a y^+ value close to 1 demonstrates a high level of mesh quality and an appropriate strategy for resolving the near-wall flow physics in this simulation. This careful meshing approach is expected to contribute to the accuracy and reliability of the obtained results.
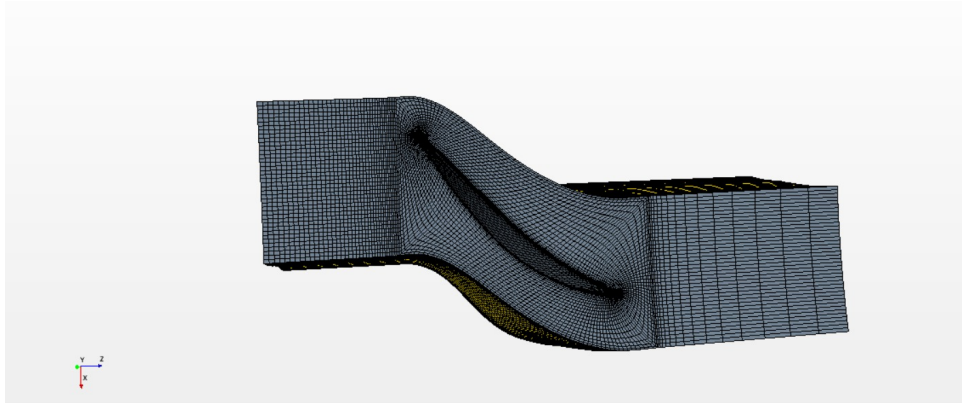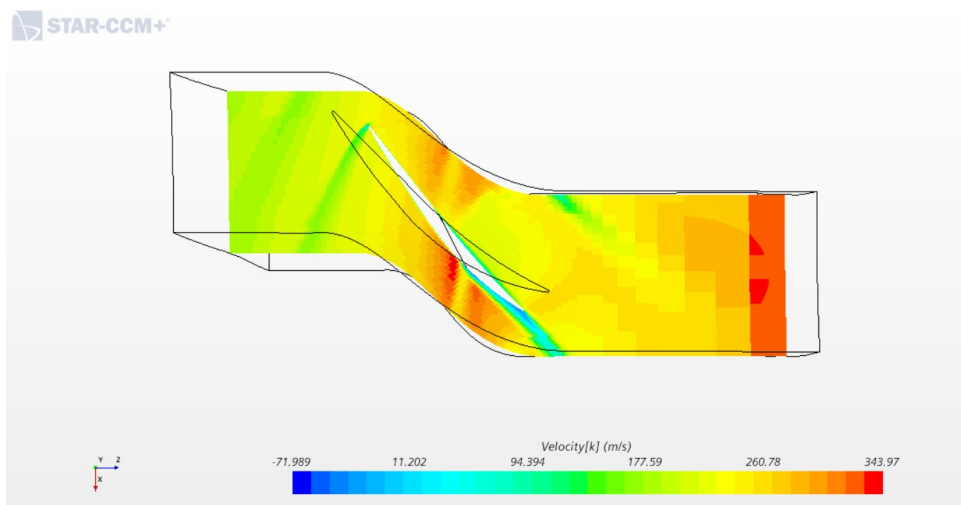

Figure 1: Mesh Scene

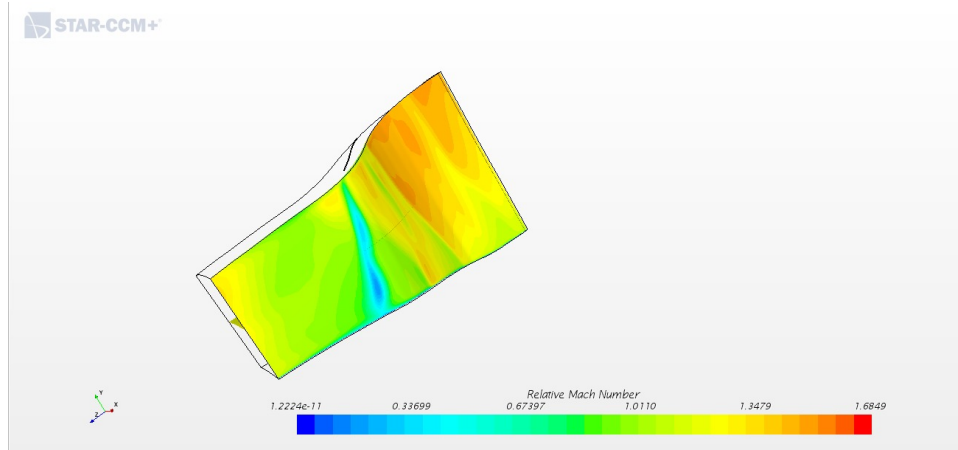# 6 Results

Figure 2: Rotor 37 Scalar Scene
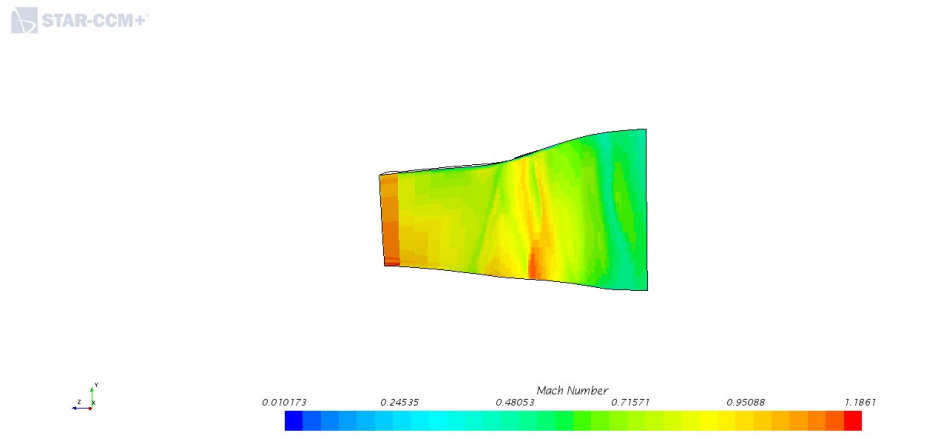
Figure 3: Rotor 37 Scalar Scene

Figure 4: Rotor 37 Scalar Scene

The simulation was carried out until the residuals of the governing equations showed satisfactory convergence. As illustrated in Figure, the residuals for continuity, momentum, energy, and turbulence models decreased significantly within the first 100 iterations. Although some residuals plateaued above 10^{-5}, the overall trend remained stable, indicating convergence to a steady-state solution.
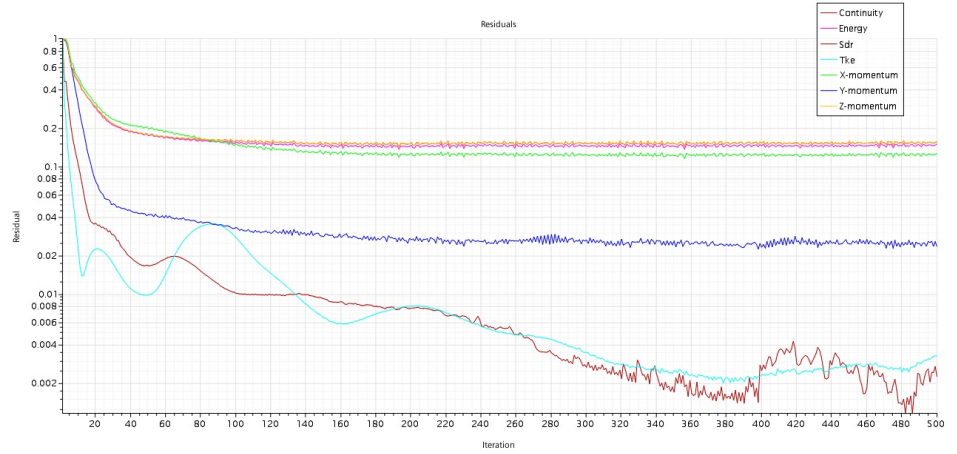
Figure 5: Residuals

6.1 Mass Flow Rate Comparison

The inlet and outlet mass flow rates were monitored throughout the simulation to verify mass conservation. Both values stabilized after approximately 100 iterations. The final recorded values were:

- **Inlet Mass Flow Rate:** -1.16 kg/s

- **Outlet Mass Flow Rate:** 1.20 kg/s

The small imbalance of 0.04 kg/s (approximately 3.3%) is within acceptable limits for steady-state simulation and is attributed to discretization and residual convergence.
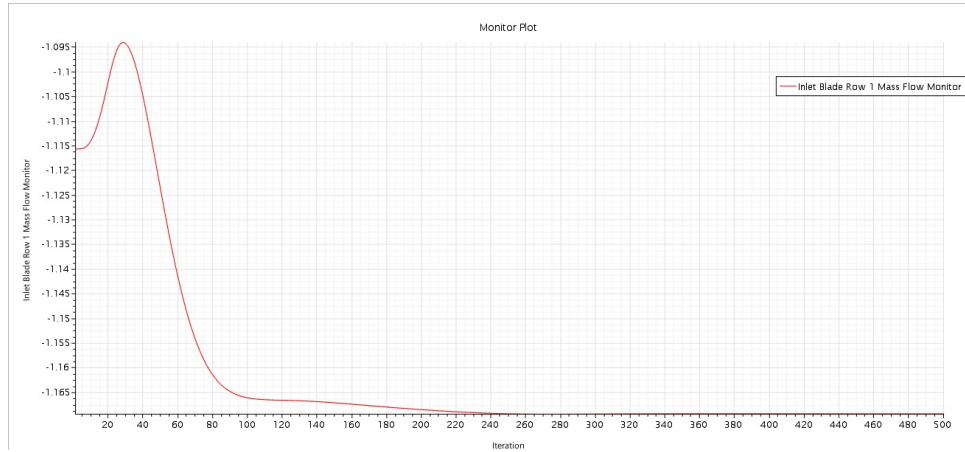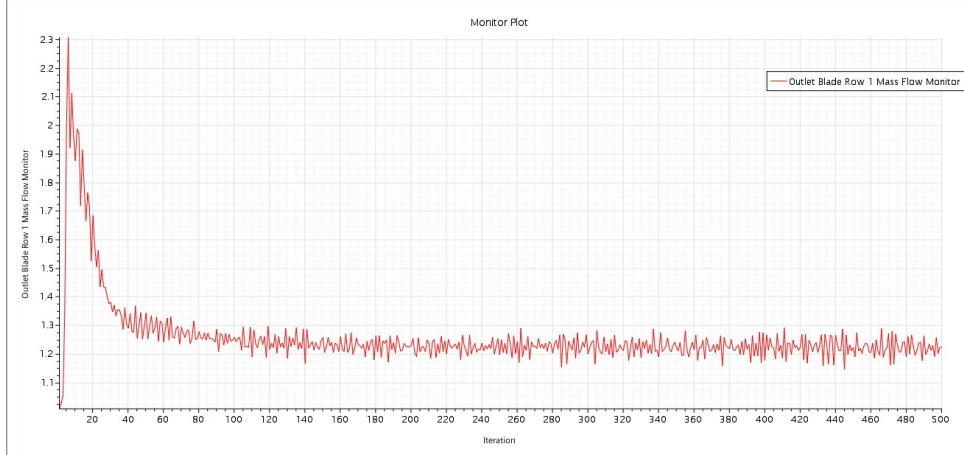

Figure 6: Inlet Mass Flow Rate

8

Figure 7: Outlet Mass Flow Rate

7 Conclusion

The CFD simulation of a single blade passage of NASA Rotor 37 was successfully conducted using the steady-state RANS equations with the k–ω turbulence model. The simulation provided insight into the rotor's aerodynamic behavior under simplified conditions, achieving good resolution near the wall with a $y^+ \approx 1$. Convergence was attained as residuals dropped below acceptable thresholds, and the inlet and outlet mass flow rates showed stable results, with a minor discrepancy within acceptable limits.

This study lays the groundwork for more comprehensive investigations, such as full-rotor simulations and performance analyses, which will enable further refinement of the model and exploration of unsteady flow phenomena.

References

[1] NASA. (2005). *NASA Rotor 37 Transonic Axial Fan.* NASA Glenn Research Center. Available at: https://www.nasa.gov/.

[2] Menter, F. R. (1994). Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications. *AIAA Journal*, 32(8), 1598-1605. 10.2514/3.12149.

[3] Svensson, T., Gustafsson, M. (2018). Computational Fluid Dynamics Simulations of Transonic Flow in Axial Flow Fans: Application to Rotor 37. *Journal of Turbomachinery*, 140(2), 021006. 10.1115/1.4039451.