Journal of Solid and Fluid Mechanics (JSFM), 15(1): 1-3, 2025

Shahrood University of Technology

Journal of Solid and Fluid Mechanics (JSFM)

DOI: **10.22044/jsfm.2025.14993.3888**

Manufacturing Engineering of Iran

Numerical Investigation of Rotational Domain Size on Aerodynamic Performance of the Caradonna-Tung Rotor in Hover Conditions

Seyed Alireza Shojaei[1], Mostafa Monfared Mosghani[2]

[1]MSc of Aerospace Engineering, Melek-Ashtar University of Technology, Shiraz, Iran
[2]Assist. Prof. of Mechanical Engineering, Malek-Ashtar University of Technology/Faculty of Naval Aviation, Shiraz, Iran
*Corresponding author: mmonfared@mut.ac.ir

Received: 2024/31/08 Revised: 2025/15/02 Accepted: 2025/08/03

Abstract

Accurate numerical simulations of helicopter rotor aerodynamics are critical for optimizing rotorcraft design and performance. This study explores the influence of rotational domain size on computational fluid dynamics (CFD) simulations of the Caradonna-Tung rotor under hover conditions. Using the Moving Reference Frame (MRF) approach in ANSYS Fluent, three different domain sizes (5 cm, 12 cm, and 18 cm) were analyzed to determine their effects on pressure coefficient distribution and lift force prediction. The rotor, featuring a 114 cm radius, 19 cm chord length, and NACA 0012 airfoil, operates at a collective pitch angle of 8 degrees and a rotational speed of 1750 RPM. The results indicate that while the 12 cm and 18 cm domain sizes yield highly accurate predictions in comparison to experimental data, the 5 cm domain size underperforms. Furthermore, simulations reveal the presence of tip vortex structures, which are not confirmed by experimental results. The findings suggest that the rotational domain should be positioned at least 8% of the rotor diameter away from the upper surface to ensure reliable aerodynamic predictions. This study provides a framework for optimizing CFD simulations, balancing accuracy and computational efficiency for rotorcraft design.

Keywords: Computational Fluid Dynamics, Helicopter Rotor, Hover Flight, Moving Reference Frame, Rotational Domain Size, Pressure Coefficient, Lift Distribution

1. Introduction

The aerodynamic behavior of helicopter rotors presents unique challenges due to their inherent unsteady and highly vortical flow structures. Unlike fixed-wing aircraft, helicopter rotors generate strong tip vortices, significantly influencing the surrounding flow field. Computational Fluid Dynamics (CFD) has emerged as a powerful tool for analyzing rotor aerodynamics, allowing for detailed insights into flow physics that are difficult to capture experimentally.

Rotor performance is heavily influenced by the size of the rotational domain used in CFD simulations. An inadequate domain size can introduce artificial flow constraints and lead to inaccuracies in aerodynamic predictions. The Caradonna-Tung rotor serves as an ideal benchmark model due to its extensive experimental data, providing a reliable reference for validating numerical methods. However, the placement and extent of the rotational domain remain a critical factor that has not been fully addressed in previous studies.

This study systematically investigates the effect of rotational domain size on the accuracy of aerodynamic predictions. By comparing simulation results with experimental data, the objective is to establish best practices for selecting the optimal domain size to balance computational cost and accuracy.

2. Methodology

A three-dimensional, compressible flow model was implemented using ANSYS Fluent. The Moving Reference Frame (MRF) approach was employed to simulate rotor rotation while maintaining a steady-state solution framework. The computational domain consisted of three different rotational region sizes (5 cm, 12 cm, and 18 cm) to examine their effects on simulation accuracy. The k-epsilon turbulence model was utilized to account for turbulence effects, ensuring that the complex vortical structures around the rotor were adequately resolved.

The computational grid was structured with refined meshing near the blade surfaces to ensure accurate boundary layer resolution. A grid independence study was conducted to verify solution convergence. Mesh refinement was particularly concentrated along the leading and trailing edges of the airfoil to enhance prediction fidelity. Boundary conditions included a velocity inlet and a pressure outlet, ensuring stable flow behavior within the computational domain.

To validate the numerical results, the simulation data were compared against experimental measurements from Caradonna and Tung's rotor tests. The primary evaluation metrics included the pressure coefficient distribution along the blade span and the integrated lift force predictions.

Figure 1 illustrates the computational domain setup for different rotational region sizes. Figure 2 provides a detailed view of the meshing strategy employed to capture boundary layer effects accurately.

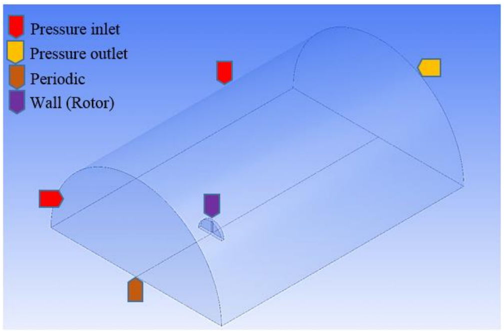

Figure 1 computational domain setup for different rotational region sizes

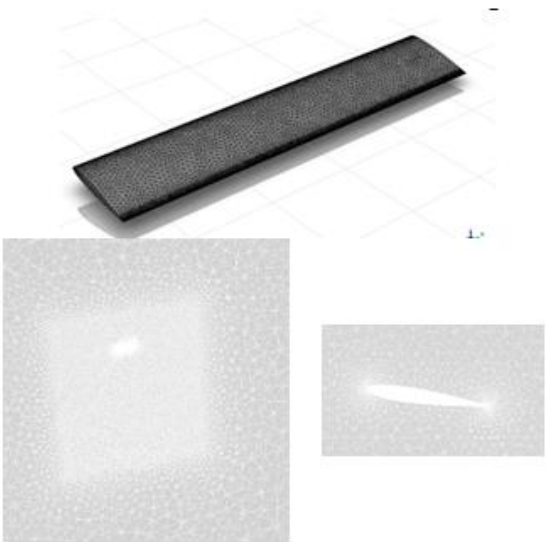

Figure 2 Mesh structure used to capture boundary layer effects

3. Results and Discussion

The simulation results were analyzed to evaluate the impact of rotational domain size on aerodynamic predictions. The findings indicate that:

- The 12 cm and 18 cm domains closely align with experimental pressure coefficient distributions, while the 5 cm domain exhibits deviations.
- Tip vortex structures were observed in the CFD results, although they were not confirmed by experimental measurements.
- Increasing the rotational domain size improved solution stability and accuracy.

The study confirms that placing the rotational domain boundary at least 8% of the rotor diameter above the blade surface is critical for achieving reliable results. Figure 3 compares the pressure coefficient distributions at different blade span locations, demonstrating the improvement in accuracy with larger domain sizes.

A key observation is that the 5 cm domain led to over-prediction of the pressure coefficient near the leading edge, suggesting that boundary-induced effects influence the solution accuracy. The increased domain size mitigated these discrepancies, resulting in a more consistent agreement with experimental data.

Additionally, the computational cost associated with increasing domain size was evaluated. While the 18 cm domain provides highly accurate results, it also demands significantly more computational resources. The 12 cm domain emerges as a balanced choice, offering both accuracy and computational efficiency. Figure 3 shows the lift distributions at different blade span locations.

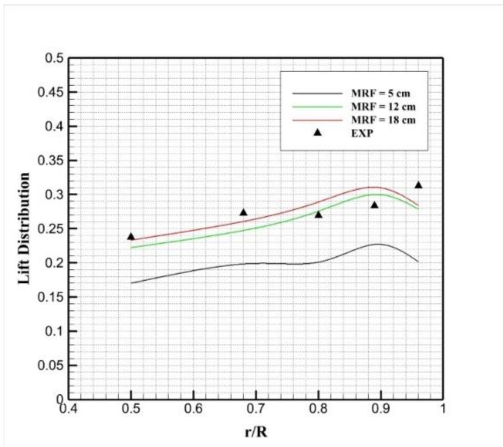

Figure 3 Lift distribution along the blade at multiple span-wise locations

4. conclusion

This study highlights the importance of selecting an appropriate rotational domain size for CFD simulations of rotor aerodynamics. The results demonstrate that a minimum rotational region size of 12 cm is necessary for accurate aerodynamic predictions. Furthermore, the study reinforces the need for careful domain boundary placement to minimize numerical inaccuracies.

The comparison between numerical and experimental results underscores the importance of carefully defining computational boundaries. While CFD simulations can accurately predict lift force and pressure distribution, discrepancies at the blade tip indicate the need for further experimental validation beyond the 0.96 span-wise location.

Future work will explore transient simulations and different turbulence models to further enhance prediction capabilities. Additionally, incorporating wake modeling approaches, such as the sliding mesh technique, may provide further insight into rotor flow dynamics beyond steady-state assumptions.

This research provides a framework for improving the accuracy and efficiency of rotor simulations, contributing to the broader understanding of helicopter aerodynamics and enhancing rotorcraft design methodologies.

5. References

[1] F. X. Caradonna and C. Tung, Experimental and Analytical Studies of a Model Helicopter in Hover, Journal of Aircraft, Vol. 24, 1987, pp. 231-238.

[2] Fluent User's Guide, Fluent Inc. July 1998, Lebanon, NH 03766, USA.

[3] Barakos, G., et al., CFD simulation of helicopter rotor flow based on unsteady actuator disk model. *Chinese Journal of Aeronautics*, 2020.

[4] Pérez Gordillo, A.M., et al., *Numerical and Experimental Estimation of the Efficiency of a Quadcopter Rotor Operating at Hover*. Energies, 2019. 12(2): p. 261.