

CALCULATING THE AERODYNAMIC CHARACTERISTICS OF THE MAIN ROTOR IN CONSIDERATION OF THE INFLUENCE OF THE HELICOPTER FUSELAGE BY CFD METHOD

Pham Thanh Dong*, Dang Ngoc Thanh, Pham Vu Uy

Le Quy Don Technical University

Abstract

This paper presents an approach using the CFD method to build a calculated model for studying aerodynamic interaction between the main rotor and helicopter fuselage. The calculated model consists of two components which are the main rotor and helicopter fuselage, that is capable of determining the aerodynamic characteristics of the main rotor in consideration of the influence of the helicopter fuselage. The presented numerical method is very useful to simulate the velocity field and the airflow around the full helicopter, calculate the lift coefficient of the main rotor in the presence and absence of the induced influence from helicopter fuselage. The results of this study show a good agreement with data from other published work.

Keywords: Helicopter rotor; rotor-fuselage aerodynamic interaction; CFD method.

1. Introduction

The need to correct or rectify the effects of aerodynamic interactions that were unforeseen or mispredicted at the design stage has historically been one of the most common causes of delay in the advancement of a new helicopter design from prototype to production. Interference between the wakes and other flow disturbances induced by the helicopter's rotors, fuselage, and lifting surfaces can produce strong loads on geometrically distant parts of the configuration. Any unsteadiness in these loads, or change in these loads as the flight condition of the aircraft is altered, can have a very strong negative influence on the dynamics of the vehicle. Experience within the helicopter industry suggests that the nature and form of the aerodynamic interactions that arise from even minor configurational changes to a fuselage can be extremely difficult to predict, and is lack of predictive capability attaches a significant degree of risk to any departures from a successful configuration.

In [1], Tan Jianfeng and Wang Haowen used the unsteady potential-based panel method to consider aerodynamics of finite thickness multi-bladed rotors, and the full-

* Email: mrbook29@gmail.com

span free-wake method is applied to simulating dynamics of rotor wake. These methods are tightly coupled through trailing-edge Kutta condition and by converting doublet-wake panels to full-span vortex filaments. A velocity-field integration technique is also adopted to overcome singularity problem during the interaction between the rotor wake and blades. By method as CFD-FASTRAN solver, Sijun Zhang, Madhaveswer Gentela and T. Fuchiwaki [2] realized the computational simulation of flows over the generic rotor fuselage interaction (ROBIN) configuration. The helicopter model studied in this work consists of a fuselage, a four-bladed rotor, and tails. The results of this work show that low Mach number preconditioning and dual time-stepping greatly improve the accuracy and efficiency of CFD-FASTRAN for helicopter flows. But in this work, the influence of the fuselage (fuselage) on the aerodynamic characteristics of the main rotor is not shown. In the studies of Xu HeYong and Bum Seok Lee and colleagues [3, 4] used the CFD method on the basis of unstructured meshing to simulate the aerodynamic characteristics of the rotor blade with fuselage of helicopter. Both of these works focused on calculating the pressure distribution on the helicopter fuselage in case the helicopter moves alone in space or when the helicopter fuselage moves in the air from the rotor blade. The influence of the fuselage on the aerodynamic properties of the rotor blade was not studied. In [6], authors used vortex theory to study helicopter rotor-fuselage aerodynamic interaction. In this paper, with CFD method, authors will focus on a particularly poorly understood element of the interactional aerodynamic environment of this configuration, namely the effect on the performance of the main rotor of its operation in close proximity to the fuselage. The interaction of the main rotor wake with that of the fuselage, and more directly, its impingement on the frame of fuselage itself, adds both unsteadiness and nonlinearity to the performance of the main rotor.

2. Simulating aerodynamic interactions of helicopter main rotor-fuselage by CFD method in ANSYS Fluent

Computational fluid dynamics is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. CFD is particularly dedicated to the fluid, in that motion and how the fluid flow behavior influences process, that may include heat transfer and possibly chemical reactions. The physical characteristics of the fluid motion can usually be described through fundamental mathematic equations usually in partial differential form which govern a

process of interest and are often called governing equations in CFD. In order to solve these mathematical equations, they are converted into computer programs or software packages using high level computer programming languages as ANSYS Fluent. The CFD is based on the Navier-Stokes equations, and these equations describe the pressure, temperature; velocity and density of a moving fluid are related.

For studying airflow around the helicopter main rotor-fuselage in flight by CFD method, in this work, the 3D models of the main rotor and helicopter fuselage, which is used for numerical simulation, are done using Inventor 2015. The presented method was applied in order to determine the thrust distribution along the span of the Mi-8 helicopter main rotor blade. The input data are given for the helicopter Mi-8 [5]:

Tab. 1. The input data of helicopter model

Rotor area	$S = 355.7 \text{ m}^2$
Rotor diameter	$D = 21.29 \text{ m}$
Number of blades	5
Blade airfoil	NACA 23012M
Blade chord	$B = 0.52 \text{ m}$
Blade twist	9°
Rotor rotational speed	$\Omega = 20.94 \text{ rad/s}$
Rotor solidity	$\sigma = 0.0777$

The enclosure around the main rotor and fuselage is made in ANSYS Workbench 16.1 as shown in Fig. 1, 2. The meshing of the main rotor and helicopter fuselage is done using ANSYS ICEM CFD 16.1 with type of unstructured mesh. The continuum from ANSYS Workbench is imported, then the continuum is divided into different parts like inlet, outlet, wall, main rotor and fuselage and the required meshing conditions are applied and the continuum is meshed (Figs. 1, 2, 3).

The lift force is calculated using the following relation:

$$F_T = C_T \frac{1}{2} \rho V^2 A \quad (1)$$

Here ρ is the density of fluid, C_T is the lift coefficient; A is swept area of rotor, $A = \pi R^2$, R is radius of rotor; $V = \Omega R$ is tip speed of rotor.

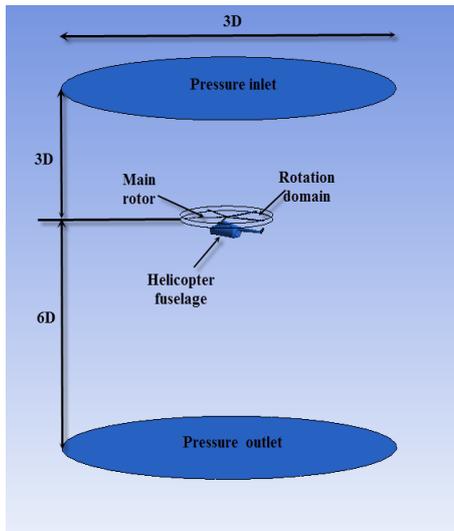


Fig. 1. Boundary condition of helicopter main rotor-fuselage model

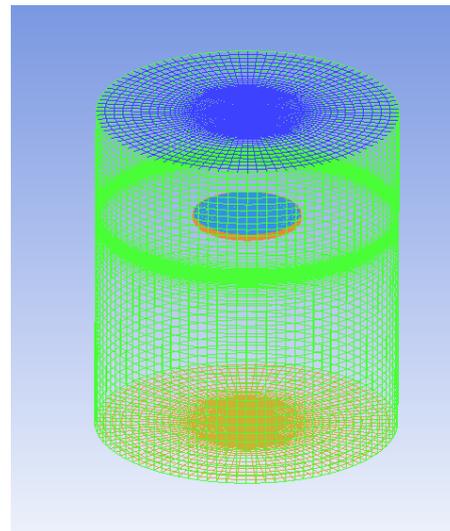


Fig. 2. Meshing station part of calculation domain

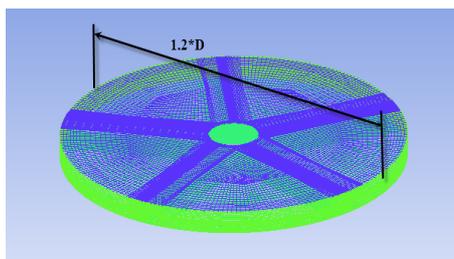
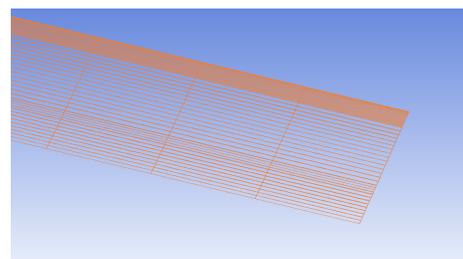


Fig. 3. Meshing rotation part of calculation domain and meshing blades of rotor



The simulation of this continuum is done in application ANSYS Fluent ver. 16.1. In this calculation domain, initially the meshing of the continuum is checked and once the software approves it, the models, materials and boundary conditions are set. For detail, the turbulence model used for this kind of simulation is the $k-\epsilon$ model. This is a two equation model in which one equation corresponds to the turbulent kinetic energy (k) and the other is the Specific dissipation rate (ϵ). The working fluid in this simulation is air. The important boundary conditions in an External Flow Analysis are pressure at inlet and outlet of the continuum. The outlet boundary condition is given as pressure and its value is given as 0 Pa. The rest of the faces of the continuum are mentioned as wall, which means that these faces are under no-slip condition, i.e. there is zero velocity on these faces. This no-slip condition means that the flow conditions will not apply outside these walls and adjacent to these walls.

3. Results and discussions

After setting up the solution methods and controls for this simulation, we get some images of results as shown in Figs. 4-8.

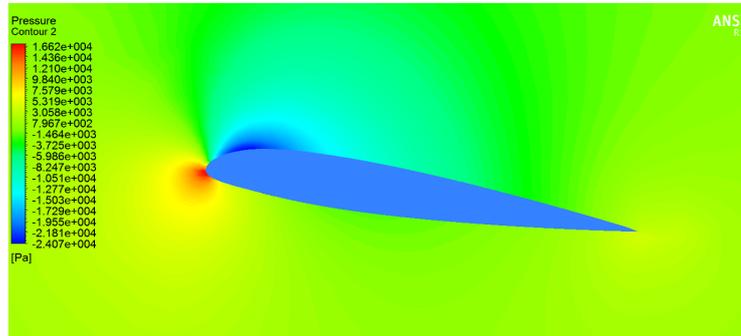


Fig. 4. Pressure contour around the airfoil of rotor blade

The both helicopter rotor and rotor-fuselage models have fairly consistent results. The pressure difference below the blade surface is relatively large, especially at the leading edge of the blade (Fig. 4). Figs. 5, 6 and 7 show that the distribution of velocity field in the both models is similarities.

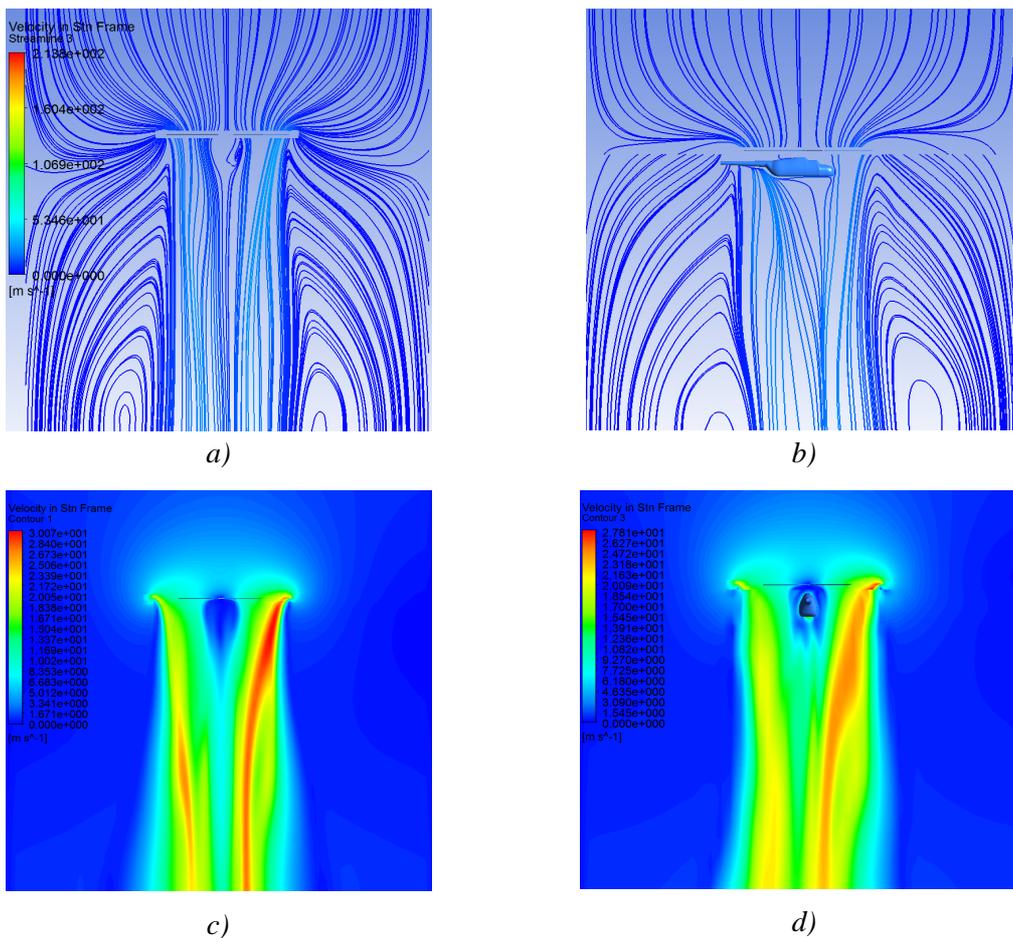


Fig. 5. Contour of velocity on vertical plane in the models of main rotor without fuselage (a, c) and with fuselage (b, d)

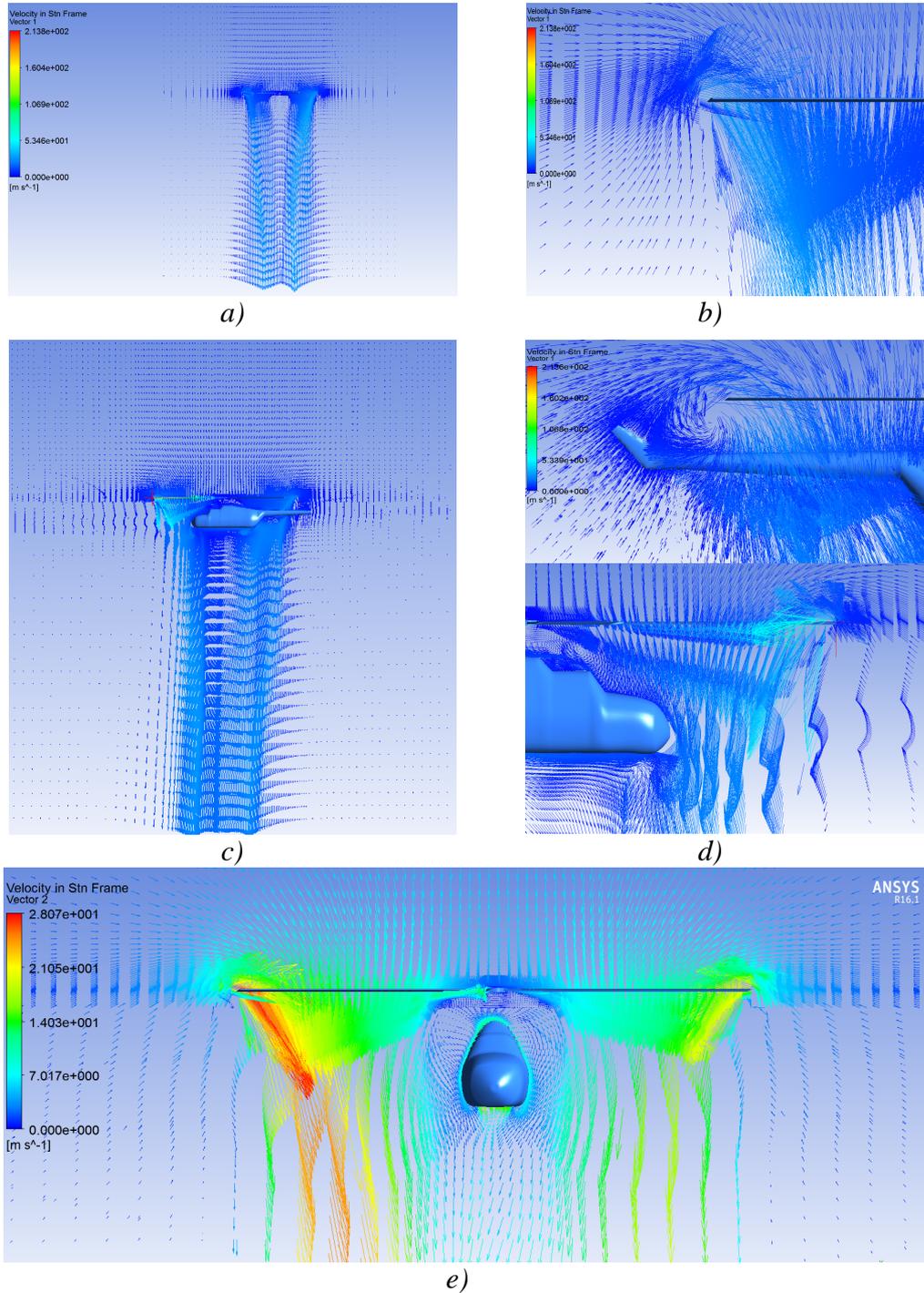


Fig. 6. Field of velocity vector on vertical plane in the models of main rotor without fuselage (a, b) and with fuselage (c, d, e)

The stream line of air are shown very clearly in Figs. 5a, 5b. When the helicopter operates, the air flow through the rotor will be accelerated, and the velocity value

increase with the length of the blade (Fig. 6e). Images of the vorticity distribution on the rotor from models without fuselage and with fuselage are very similar (Fig. 7).

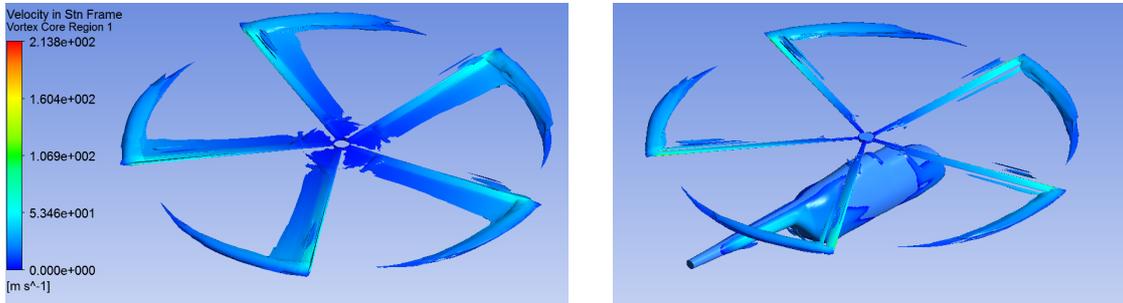


Fig. 7. Vorticity discriminant in the models of main rotor without fuselage and with fuselage

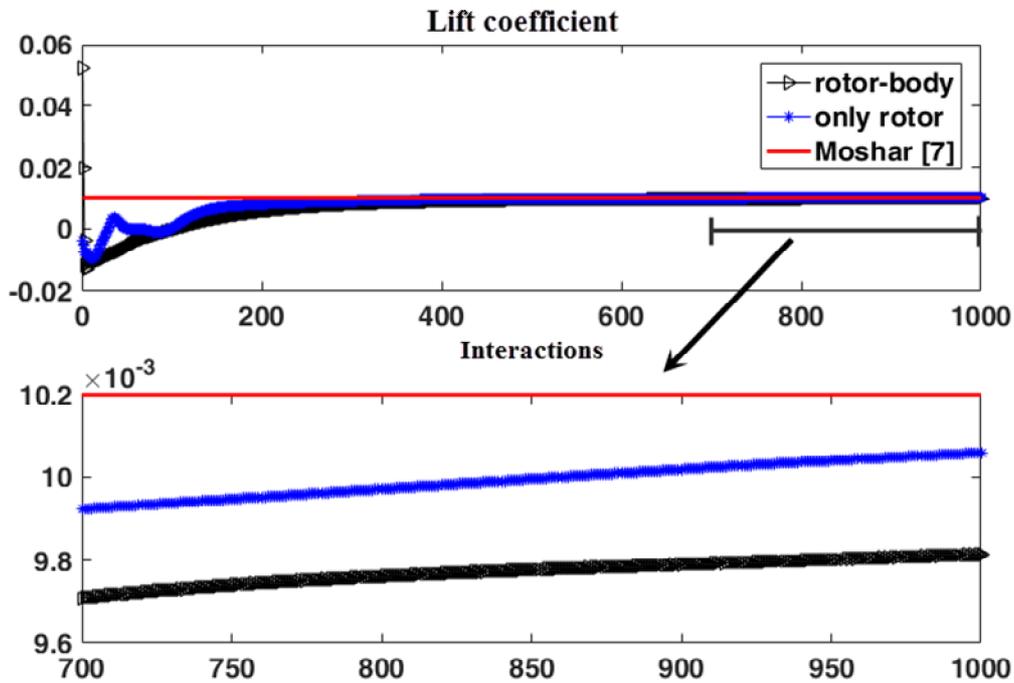


Fig. 8. Rotor lift coefficient and rotor-fuselage lift coefficient convergence

The lift coefficients of the helicopter rotor in the alone rotor model converges gradually to value 0.0101, that in accordance with the calculation results in the work of Мощарь П.И. [7]. In rotor-fuselage model lift coefficients of the rotor converges to value 0.00983. Thus, the lift coefficients of the helicopter rotor in the alone rotor model and rotor-fuselage model have a certain changes. However, the change is not great. Accordingly, the lift coefficient of the helicopter rotor in the alone rotor model will decrease when the helicopter fuselage is added to the model (Fig. 8).

4. Conclusions

In the paper, the authors have set up a mathematical model to study the aerodynamic characteristics of the main rotor in consider influence of helicopter fuselage by CFD method. Computational simulation and comparison of individual main rotor model with the main rotor-fuselage model in hover regimes. The results are consistent with the theory calculation by vortex model of main helicopter rotor in [5], that influence of helicopter fuselage on the aerodynamic characteristics of the main rotor in terms of lifting force is not much.

CFD is an effective method for simulating aerodynamics of helicopter rotor and can be used to verify the accuracy of other theoretical models.

References

1. Tan J, Wang H (2013). Panel/full-span free-wake coupled method for unsteady aerodynamics of helicopter main rotor blade. *Chinese Journal of Aeronautics*, 26(3), pp. 535-543.
2. Sijun Z, Madhaveswer G, Fuchiwaki T (2009). CFD Developments for Unsteady Aerodynamics of Rotorcraft in Forward Flight with Realistic Motion Schedules. *AIAA Computational Fluid Dynamics 19th*.
3. Xu HY, Ye ZY (2012). Numerical simulation of rotor-fuselage aerodynamic interaction based on unstructured dynamic overset grids. *Science China Technological Sciences*, 55(10), pp. 2798-2807.
4. Bum SL, Mun SJ, Oh JK (2010). Numerical Simulation of Rotor-Fuselage Aerodynamic Interaction Using an Unstructured Overset Mesh Technique. *International Journal of Aeronautical and Space Sciences*, 11(1), pp. 1-9.
5. Данилов В.А. (1988). *Вертолет Ми-8 -Устройство И Техническое Обслуживание*, Москва, “Транспорт”.
6. Pham Thanh Dong, Nguyen Anh Tuan, Dang Ngoc Thanh, Pham Vu Uy (2018). Numerical Method to Study Helicopter Main Rotor-fuselage Aerodynamic Interaction. *International Conference on Fluid Machinery and Automation Systems 2018 (ICFMAS2018)*/Ta Quang Buu Library, Hanoi University of Science and Technology (HUST).
7. Моцарь. П.И., Удовенко В.А. (2009). “Расчет углов атаки сечений лопасти и аэродинамических характеристик винта, зная распределение интенсивности вихревого слоя, в рамках метода дискретных вихрей”. *Открытые информационные и компьютерные интегрированные технологии*, № 43, стр. 114-123.

TÍNH TOÁN ĐẶC TRƯNG KHÍ ĐỘNG CỦA CÁNH QUAY CÓ XÉT ĐẾN SỰ ẢNH HƯỞNG CỦA THÂN TRỰC THĂNG BẰNG PHƯƠNG PHÁP CFD

Tóm tắt: Bài báo trình bày một cách tiếp cận sử dụng phương pháp CFD để xây dựng mô hình toán nghiên cứu tương tác khí động giữa cánh quay và thân trực thăng. Mô hình tính toán gồm hai thành phần là cánh quay và thân trực thăng, có khả năng xác định đặc tính khí động lực học của cánh quay xét đến ảnh hưởng của thân trực thăng. Phương pháp số trình bày trong bài báo rất ưu việt trong việc mô phỏng trường tốc độ và dòng không khí chảy bao xung quanh toàn bộ trực thăng, tính toán hệ số lực nâng của cánh quay trong trường hợp có và không có sự tác động cảm ứng từ thân trực thăng. Kết quả nghiên cứu cho thấy sự tương đồng với số liệu đã công bố trong công trình nghiên cứu quốc tế khác.

Từ khóa: Cánh quay; tương tác khí động; mô phỏng số thủy động lực học.

Received: 08/4/2019; Revised: 03/4/2020; Accepted for publication: 06/4/2020

