Numerical simulation and Experimental Research on Wind Field Performance of Array Fan

Junlai Li, Panpan Jia, Cheng Zhao, Yonghe Xie
School of Naval Architecture and Maritime, Zhejiang Ocean University
Zhoushan city, Zhejiang province, China
School of Transportation, Wuhan University of Technology
Wuhan city, Hubei province, China

ABSTRACT

In life, we often need a uniform velocity of the wind field to meet our daily needs. In this study, methods based on 2 × 2 fan array (FA) experiments and numerical simulations were first compared, and then these methods were used to study the array effect on nine FAs. As the number of fans increased, the array effect of FAs increased in the rectifier tube; however, as the length of the rectifier tube increased, the range of uniform velocity of the wind field also increased.

KEY WORDS: Axial flow fan, CFD; FAs; Streamlines; Monitors.

INTRODUCTION

Floating offshore wind turbines have attracted the attention of scholars in recent years. General researchers used one or more axial flow fans to generate wind energy to rotate the fan blades of experiments with a floating offshore wind turbine. Jae Hyuk Jung and Won Gu Joo (2019) showed that the fluid was unstable after passing through the axial flow fan, and a large vortex was generated before and after the axial flow fan, which affected the results of the experiment. A fan array method was used to obtain a relatively uniform wind at a certain distance in the rectifier tube. This can meet the experiment of floating offshore wind turbines, thereby ensuring the accuracy of the experimental results.

First, we considered the minimum wind velocity of the generator blades of a floating offshore wind turbine as the standard. We obtained the minimum wind speed at the outlet of the wind rectifier through the computational fluid dynamics (CFD) method and then calculated the number of axial flow fans that produced the lowest wind velocity. Because the FA experiment could not fully meet the ideal conditions of the numerical simulation, we first used 2 × 2 FAs to conduct the experiment and numerical simulation to study the errors of the two methods. Based on the above results, the number of axial fans was increased, and the fan ring array was used to experimentally study the array effect of nine FAs on the rectifier tube. Finally, the accuracy of the experimental results was verified again by CFD, and further discussion was conducted.

Sun et al. (2012) provided a brief overview of the status quo of floating offshore wind turbine generation in different countries worldwide and the impact of technical, economic, and environmental issues on the development process. The researchers found that wind energy, as the fastest-growing renewable resource, is becoming increasingly popular worldwide. In 2006, Nielsen et al. (2006) described two types of integrated dynamic analysis for floating offshore wind turbines and compared different simulated models with wind turbine model experiments to adapt a maximum power control strategy and constant power control strategy. Hably et al. (2013) stated that high-altitude wind energy has attracted increasing attention in the field of new renewable resources and proposed a new type of wind power generation system that uses experiments and numerical simulations to verify the feasibility of high-altitude wind power generation.

Physical experiments are generally used to verify the accuracy of wind field experiments. When simulating natural wind, we often consider the name of the fan that can be seen everywhere in the market as an axial fan. After natural wind is rotated by the axial flow fan and passes through the rectifier tube, the artificial wind generated can meet the wind velocity required in an experiment of the floating offshore wind turbine. The advantage of the axial flow fan is that it has a low unit energy consumption and high efficiency.

Energy saving is increasingly popular among researchers. Ito et al. (2009) verified through physical experiments and CFD methods that the optimized design of the blade-tip clearance of small axial fans (axial fans with a diameter of 85 mm) can improve the aerodynamic performance of axial fans and significantly reduce fan noise. Panigrahi and Mishra (2014) used the CFD method to simulate six typical airfoil sections through coefficients of lift (C_l) and coefficients of drag (C_d) to evaluate a blade profile that can improve the efficiency of mine ventilation. By studying the traditional fan of NASA Rotor 67, Shahsavari and Nili-Ahmadabadi (2020) conducted calculation experiments and simulated the turbofan cycle based on the principles of...