HYDRODYNAMIC AND AERODYNAMIC ANALYSIS OF OFFSHORE WIND TURBINE FOUNDATION MODEL USING FINITE ELEMENT METHOD

PHÂN TÍCH THỦY ĐỘNG LỰC HỌC VÀ KHÍ ĐỘNG LỰC CHÂN ĐẾ ĐIỆN GIÓ BẰNG PHƯƠNG PHÁP PHẦN TỬ HỮU HẠN

Tien Dung Nguyen[1], Ngoc Thai Huynh[2], Hoang Trong Tran Huy[2,*]
[1]Faculty of Fundamental Sciences, PetroVietnam University, Ba Ria, Vung Tau District, Vietnam
[2]Ho Chi Minh City University of Industry and Trade, Ho Chi Minh City, Vietnam
**Corresponding author email: huyhtt@huit.edu.vn*

ABSTRACT

With the increasing demand for electricity, people have to pay more and more for electricity consumption. There are many natural energy sources to generate electricity. However, wind, water and solar energy will help reduce the cost of generating electricity. However, to generate electricity from wind power, offshore wind power is the strongest and can generate the least expensive electricity. However, it requires the manufacture of a base which is also quite expensive. Therefore, in this study, the design of offshore wind power base is studied. To reduce the cost of testing, this study proposes to perform hydrodynamic and aerodynamic simulations using the finite element method. The analysis results indicated that the design dimensions of the diameter of legs and the diameter of the brace rods were less affected by wind and water. Because the water and wind pressures acting on the model changed insignificantly when the leg diameter and brace rods diameter increased from 1500 mm to 1700 mm and 700 mm to 900 mm, respectively.

Keywords: *Offshore wind power foundation; Hydrodynamics; Aerodynamics; Finite element method.*

TÓM TẮT

Với nhu cầu sử dụng điện ngày càng tăng, con người phải trả nhiều tiền cho việc tiêu thụ điện. Có nhiều nguồn năng lượng tự nhiên để tạo ra điện. Tuy nhiên, năng lượng gió, nước và mặt trời sẽ giúp giảm chi phí tạo ra điện. Tuy nhiên, để tạo ra điện từ năng lượng gió, năng lượng gió ngoài khơi là mạnh nhất và có thể tạo ra điện ít tốn kém nhất. Tuy nhiên, nó đòi hỏi phải chế tạo một chân đế cũng khá tốn kém. Do đó, trong nghiên cứu này, đã nghiên cứu thiết kế chân đế điện gió ngoài khơi. Để giảm chi phí thử nghiệm, nghiên cứu này đề xuất thực hiện mô phỏng thủy động lực học và khí động học bằng phương pháp phần tử hữu hạn trong Fluent tool of ANSYS. Kết quả phân tích chỉ ra rằng kích thước thiết kế của đường kính chân và đường kính thanh giằng ít bị ảnh hưởng bởi gió và nước. Bởi vì áp lực nước và gió tác động lên mô hình thay đổi không đáng kể khi đường kính chân và đường kính thanh giằng tăng lần lượt từ 1500 mm lên 1700 mm và 700 mm lên 900 mm.

Từ khóa: *Chân đế điện gió ngoài khơi; Thủy động lực học; Khí động lực học; Phương pháp phần tử hữu hạn.*

☞

1. INTRODUCTION

There are many and endless sources of natural energy, such as wind power, solar energy, water... Among them, this source of energy can generate electricity for daily life and industry. Generating electricity from natural energy is a big challenge for researchers. There are many projects that convert wind energy into electricity that have been published. The higher the wind, the stronger the wind, which means the more electrical energy is obtained. Therefore, creating a structure for a wind power foundation is a big challenge for researchers. Wind power foundation are also manufactured in many ways, such as floating offshore wind turbines [1] have been applied to analyze the aerodynamics of the turbine and the hydrodynamics of the offshore wind power platforms structure. The experiment was also performed to validate the numerical simulation results of the proposed model. The wind power platforms structure was made of many parts which are connected by joints which combine adhesive bonding with welding [2]. These joints have helped to significantly reduce the time and cost of manufacturing offshore wind turbine platform models by welding method. However, the proposed joints only meet the needs of medium-sized structures. The cost of placing floating foundations offshore is very high. Therefore, the mooring systems model [3] have been proposed and tested with three different types of mooring systems such as catenary mooring system, tension-leg mooring system, and semi-taut mooring system with numerical simulation models to simulate complex wind turbine dynamics. The optimal U-type Darrieus Wind Turbine [4] was achieved by GA, PSO, SA, and quadratic Response Surface (RS) method combined with NSGA - II based on the analysis of aerodynamic loads and structural forces of fluid-structure interactions. Numerical results indicated that the proposed model achieves higher efficiency than conventional two- and four-blade turbines. To reduce the impact of wind and sea waves on the Offshore Wind Turbine monopile, a vibration absorption model has been developed [5]. Numerical simulation results of finite element method and fluid dynamics confirm that the proposed vibration control model has improved the service life, reliability and durability of the Offshore Wind Turbine monopile structure. The jacket support structure for offshore wind turbine has been optimized by Topology optimization method [6-8] which has reduced the weight of the structure by 38.24% and significantly improved the stress concentration. The durability and service life of the jacket monopile foundations are significantly improved by optimizing the proposed structural model using a genetic algorithm [9] based on a surrogate model that is converted from a finite element model. The surrogate model can save up to 98.61% of the design time of the jacket monopile foundations model. Wind Energy with Integrated Servo-Control is applied [10] to optimize Several control co-design of floating offshore wind turbines. The results of the proposed method have reduced the energy cost by 4%. The proposed model has significantly improved the structural stability. To reduce the vibration impact of wind and sea waves on offshore wind turbines, distributed tuned mass [11] damper was applied. From the numerical results of finite element analysis, it is shown that the proposed damper model achieves superior performance while wind and sea waves impact on offshore wind turbines.

2. DESIGN OFFSHORE WIND POWER FOUNDATION MODEL

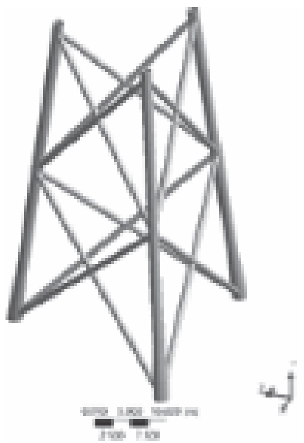


Figure 1. Offshore wind power foundation model

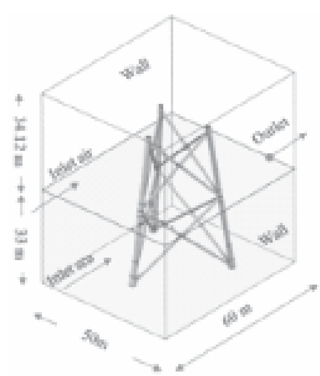


Figure 2. Set up the computational space domain and boundary conditions

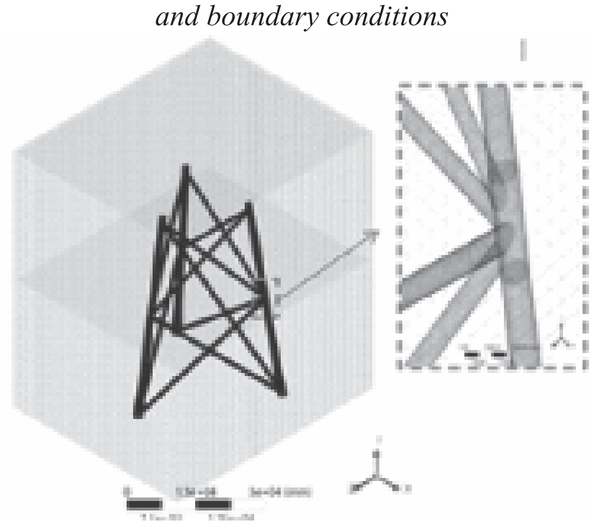


Figure 3. Result of element grid setup

Offshore of the Changhua project – CHW2204. This project is being designed and constructed at the National Energy Center in Vung Tau city, including 33 Suction Bucket Jacket (SBJ) wind power bases with types B, C, D and E corresponding to deep water and shallow water areas. The maximum height for SBJ type E is 83.4 m and the lowest for type B is 73.5 m corresponding to the design depth of the seabed (DSBL) of -40.1 m and -31.5 m respectively. The OWTs wind power base configuration in this study includes 3 main legs and 2 X-bracing floors, this model is designed on Inventor software. Then these models are imported into ANSYS to set up FEM, Fluent simulation. The jacket structure is 60 m high, with the top and bottom widths being 19 m and 32 m respectively. The structure is located at a seabed depth of -31 m. The material used for the main structure is S355 steel (except for secondary structures such as boat landing, stairs, etc.) with mechanical properties such as density ρ = 7850 kg/m^3, Young's modulus E = 2.1 x 10^{11} N/m^2, shear modulus G = 8.0 x 10^{10} N/m^2.

3. FINITE ELEMENT MODEL

One of the factors affecting the durability of offshore structures is the flow load of water and wind. And the method of analyzing the interaction between fluid and structure (Fluid – Structure Interaction) is chosen in the structural analysis. The assessment of the mutual impact caused by the interaction between fluid and structure in structural dynamics requires establishing a computational domain for the fluid called the computational space domain. The simulation space is the area surrounding the object limit during the simulation process. Due to the complexity of the simulation, limited computer resources and simulation ☞

time, the simulation space needs to be divided in the most reasonable way with the height, width and length of the computational domain having specific dimensions as shown in Figure 2 to ensure that the entire base structure is in the fluid environment and is true to reality. Because the base structure is affected by 2 fluid flows of seawater and air, the simulation space is divided into 2 main components including seawater and air simulation.

Setting up boundary conditions for the model helps the software solver read and apply automatically and systematically:

- Inlet boundary type: Is the flow area "entering" the calculation domain;

- Outlet boundary type: Is the flow area "leaving" the calculation domain;

- Walls boundary type: Is the area surrounding the model.

In this study, we will focus on analyzing the influence of fluid flow on the base structure. Then the influence of the Tower turbine will be converted into the load acting on the base structure at the interface location.

The ANSYS Meshing function is used to set up the meshing. The purpose is to divide the CAD model into smaller domains and are continuously connected through nodes and edges called elements, on which a set of equations are solved. The meshing work is done through the settings in the software. The mesh in this study is set up for 2 stages, the first stage is set up for Fluent simulation, the second stage is set up for static structural analysis. The software uses the tetrahedral meshing method.

In this study, by applying FEM, the model is divided into small computational domains called meshes including geometric elements and nodes from the Meshing function in ANSYS software. In this study, the settings for meshing the model are detailed as follows:

For the fluid component (Sea - Air): the mesh size is 1000mm, automatically divided by the software.

The structure of the wind power base requires higher precision, so the mesh size is 100mm and follows the tetrahedron meshing method:

- Tetrahedron meshing method: Tetrahedrons;

- Independent patch algorithm: Patch Independent;

- Maximum element mesh size: 100 mm.

Then the number of nodes and geometric elements are: 1212033 and 3087908 respectively as presented in Figure 3. The selection of parameters to be included in the problem is essentially the assignment of conditions and attributes to solve the problem. The parameters included will determine the accuracy and optimization of the simulation.

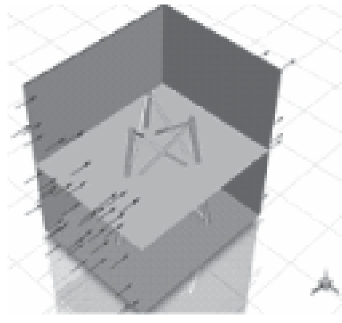


Figure 4. Simulation in Ansys Fluent

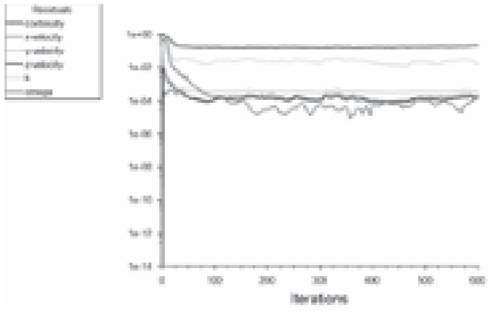


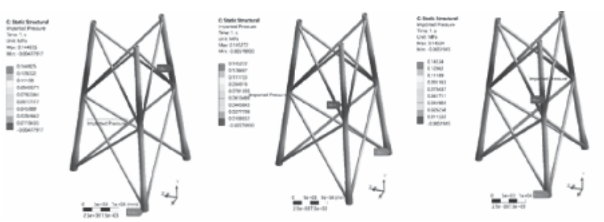
Figure 5. Simulation charts in ANSYS Fluent

The Fluent simulation problem settings include setting the properties and boundary conditions of the CFD model. The boundary conditions are set according to the upstream magnitude field perpendicular to the wall boundary, which will create a hydrodynamic field suitable for the interaction between the fluid and the structure. The solution method will be "Pressure-velocity coupling". The pressure-based solver is selected in this problem to perform pressure correction. Fluid properties, boundary conditions and numerical approach of CFD model (ANSYS – Fluent).

4. RESULTS AND DISCUSSION

4.1. Pressure acting on the base model with the change of variable D_2

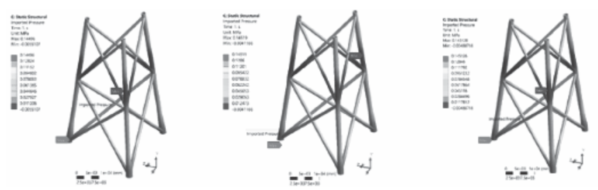
Through the Fluent simulation process, the results obtained are the pressure acting on the wind power base structure frame, shown in Figures 4.2a to 4.2c.



a) D_2 = 700 mm b) D_2 = 800 mm c) D_2 = 900 mm
Figure 6. Effect of variable D_2 on pressure acting on the model

When the variable D_2 increased from 700 mm to 800 mm, the pressures acting on the offshore wind power foundation model were 0.144825 Mpa and 0.145272 MPa, respectively As the diameter D_2 continued to increase to 900 mm, the pressure acting on the wind power foundation model was 0.14534 MPa. This problem indicated that the diameter dimension D_2 only made a small changed to the pressure acting on the offshore wind turbine foundation model. This result was smaller than the results obtained from published works [1-5].

4.2. Pressure acting on the base model with the change of variable D_1



a) D_1 = 1700 mm b) D_1 = 1600 mm c) D_1 = 1500 mm
Figure 7. Distribution of the pressure field

Similarly When the diameter dimension D_1 increased 1600 mm to 1700 mm, the pressures acting on the offshore wind power foundation model were 0.144825 Mpa and 0.145272 MPa, respectively As the diameter D_1 continued to increase to 1800 mm, the pressure acting on the wind power foundation model was 0.14534 MPa. This problem indicated that the diameter dimension D_2 only made a small change to the pressure acting on the wind turbine foundation model. This result is smaller than the results obtained from published works [1-5].

5. CONCLUSIONS

In this investigation, the offshore wind power foundation model was designed by INVENTOR software. The hydrodynamic and aerodynamic analysis impacted on the studying ☞

model was carried out by using finite element analysis method in fluent tool of ANSYS. The obtained results indicated that when the diameter dimension D_1 increased 1600 mm to 1700 mm, the pressures acting on the offshore wind power foundation model were 0.144825 Mpa and 0.145272 MPa, respectively. As the diameter D_1 continued to increase to 1800 mm, the pressure acting on the wind power foundation model was 0.14534 MPa. When the variable D_2 increased from 700 mm to 800 mm, the pressures acting on the offshore wind power foundation model were 0.144825 Mpa and 0.145272 MPa, respectively As the diameter D_2 continued to increase to 900 mm, the pressure acting on the wind power foundation model was 0.14534 MPa. This problem indicated that the diameter dimensions D_1 and D_2 only made a small changed to the pressure acting on the offshore wind turbine foundation model. ❖

References:

[1]. A. Otter, J. Murphy, V. Pakrashi, A. Robertson, and C. Desmond, "*A review of modelling techniques for floating offshore wind turbines*". Wind Energy, vol. 25, no. 5, pp. 831-857, 2021, doi: 10.1002/we.2701.

[2]. M. Albiez et al., "*Hybrid joining of jacket structures for offshore wind turbines – Validation under static and dynamic loading at medium and large scale*". Engineering Structures, vol. 252, 2022, doi: 10.1016/j.engstruct.2021.113595.

[3]. M. Barooni, T. Ashuri, D. Velioglu Sogut, S. Wood, and S. Ghaderpour Taleghani, "*Floating Offshore Wind Turbines: Current Status and Future Prospects*". Energies, vol. 16, no. 1, 2022, doi: 10.3390/en16010002.

[4]. S.-H. Ju and C.-H. Hsieh, "*Optimal wind turbine jacket structural design under ultimate loads using Powell's method*". Ocean Engineering, vol. 262, 2022, doi: 10.1016/j.oceaneng.2022.112271.

[5]. A. Kampitsis, K. Kapasakalis, and L. Via-Estrem, "*An integrated FEA-CFD simulation of offshore wind turbines with vibration control systems*". Engineering Structures, vol. 254, 2022, doi: 10.1016/j.engstruct.2022.113859.

[6]. X. Tian et al., "*Optimization design of the jacket support structure for offshore wind turbine using topology optimization method*". Ocean Engineering, vol. 243, 2022, doi: 10.1016/j.oceaneng.2021.110084.

[7]. C. Zhang, K. Long, J. Zhang, F. Lu, X. Bai, and J. Jia, "*A topology optimization methodology for the offshore wind turbine jacket structure in the concept phase*". Ocean Engineering, vol. 266, 2022, doi: 10.1016/j.oceaneng.2022.112974.

[8]. X. Tian, G. Liu, W. Deng, Y. Xie, and H. Wang, "*Fatigue constrained topology optimization for the jacket support structure of offshore wind turbine under the dynamic load*". Applied Ocean Research, vol. 142, 2024, doi: 10.1016/j.apor.2023.103812.

[9]. S. Zheng, C. Li, and Y. Xiao, "*Efficient optimization design method of jacket structures for offshore wind turbines*". Marine Structures, vol. 89, 2023, doi: 10.1016/j.marstruc.2023.103372.

[10]. N. J. Abbas, J. Jasa, D. S. Zalkind, A. Wright, and L. Pao, "*Control co-design of a floating offshore wind turbine*". Applied Energy, vol. 353, 2024, doi: 10.1016/j.apenergy.2023.122036.

[11]. S. Elias, "*Vibration improvement of offshore wind turbines under multiple hazards*". Structures, vol. 59, 2024, doi: 10.1016/j.istruc.2023.105800.