eISSN: 2721-5717; pISSN: 2747-2124 https://journal.unhas.ac.id/index.php/zonalaut

ZONA LAUT

JOURNAL OF OCEAN SCIENCE AND TECHNOLOGY INNOVATION

Analysis Of Changes In Fluid Flow Characteristics Around Spar-3vp On Flow Velocity Variations In Extreme Conditions

*Galy Rakaziwi dan Fuad Mahfud Assidiq
Department of Ocean Engineering, Faculty of Engineering, Hasanuddin University, Indonesia
*Email: rakaziwigaly@gmail.com

Abstract

This research discusses the SPAR offshore wind turbine, a floating foundation turbine used for wind power generation in the deep sea. SPARs are designed with a large-diameter vertical floating cylinder weighted at its lower end, creating a structure that is less responsive to wind, waves, and currents. SPAR foundations have advantages in deep water due to their low center of mass, simple ballast stabilization, and high stability. The purpose of this research is to gain a deeper understanding of the fluid flow occurring around the SPAR, which impacts the performance and stability of the offshore wind turbine. This research will focus on CFD (computational fluid dynamics)-based numerical modeling analysis with the help of 3D analysis software ANSYS by considering several variations of fluid flow velocity to identify the pressure, velocity, and changes in fluid flow around the SPAR. The results of this study are expected to help in reducing the risk of SPAR foundation damage, improving turbine stability, and overall increasing the efficiency of offshore wind energy generation. This research has positive implications for reducing the environmental impact of energy generation compared to traditional fossil fuel sources.

Keyword : CFD, Fluid, Offshore wind turbine, SPAR, Vertikal plate

1. INTRODUCTION

SPAR Offshore Wind Turbine is a type of floating foundation turbine used to generate wind electricity. This SPAR design consists of a large diameter vertical floating cylinder that is weighted at the bottom end with a deep draft, making the structure less responsive to wind, waves and currents. Catenaries or mooring lines spread out with pulling or suction anchors ensure the SPAR stays in place. SPAR cylinders float vertically in water and usually have a ballast tank in some part of their volume. They can be made of steel, concrete, or a combination of both. SPAR foundations have a small area and a large cylindrical mass below the water surface, which makes them suitable for deep sea applications. SPAR foundations have advantages in deep water because their center of mass is lowered, the ballast stabilization is simple, and they have high stability with a large draft. Offshore wind turbines are a type of wind turbine designed with high stability because this device will be placed in deep sea which does not allow for a fixed-bottom foundation. This is one of the advantages of the SPAR design because offshore wind turbines have the potential to reduce the environmental impact of energy generation compared to traditional fossil fuel sources. However, the SPAR design for offshore wind turbines also has several weaknesses, one of which is expensive construction and maintenance costs because the design is quite complex so it requires special equipment and expertise to work on.

SPAR floats are basically one of the fundamental alternatives for wind turbine float substructures. When its location is relatively between the center of gravity and the center of buoyancy, the stability of the SPAR buoy exhibits unique properties. The buoyancy force generated by the SPAR platform supports the entire offshore wind turbine, and the tension of the mooring lines maintains the position of the SPAR-type flying substructure station [1]. Several modifications to the existing substructure such as stepped-SPAR, counterweight disc and heave plate were offered as alternatives. Initially, the stepped-SPAR model showed many advantages compared to basic SPAR, including acceptable hydrodynamic performance based on turbulent winds, use of a 4 x 1 mooring pattern as an active pitch reducer, and the possibility to be implemented in moderate waters for potential savings [2]. SPAR structures on offshore wind turbines can create complex wind fields and turbulent flows around them. This can affect turbine efficiency and generate additional dynamic stresses on the structure. SPARs are usually designed to handle high wave loads. Large waves can cause vibrations and dynamic loads that affect the performance and durability of structures. Therefore, this research will identify the fluid flow around the SPAR to reduce the risk of damage to the SPAR foundation and to increase turbine stability. In this research, 3D modeling analysis will be carried out using ANSYS software with several variations in speed (v) to determine the pressure, speed and changes in fluid flow around the SPAR.

2. METHOD

Modeling analysis will use 3D analysis software (ANSYS) with 4 variations in fluid flow velocity, namely 0.661 m/s, 6,114 m/s, 61,144, and 611,438 m/s. The research method is numerical modeling based on CFD (Computational Fluid Dynamics). This research will focus on investigating the fluid flow around the SPAR to determine the speed and pressure of the surrounding fluid. The type of SPAR that will be modeled is SPAR-3VP, namely a SPAR that has 3 damper plates. To simulate the flow conditions around the SPAR-3VP, appropriate domain and boundary conditions must be determined first. This includes determining the inlet velocity, pressure, and other relevant parameters around the SPAR. The following domains are used in modeling:

Table 1. Model domain values in meters

	X	Y	Z
+	100	30	1
-	50	30	90

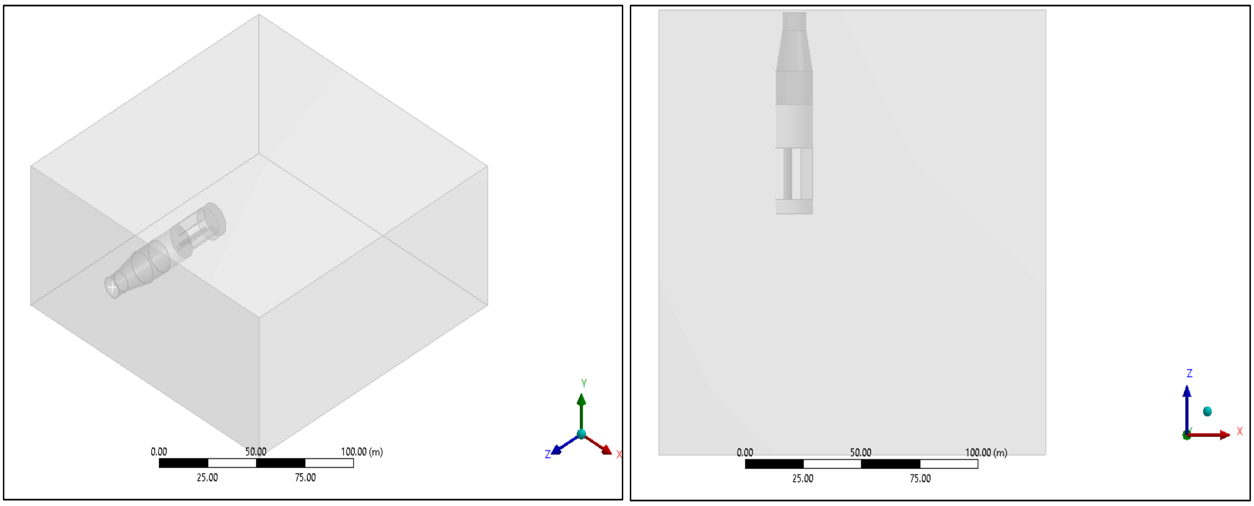

Figure 1. Domain model

In this research, the type of turbulence used is IDDES-SST (Improved Delayed Detached Eddy Simulation with Shear-Stress Transport) k omega. The choice of this type of turbulence will allow more accurate modeling of the turbulent flow around SPAR-3VP. The constant model data is; Cdes (inner) = 0.78, Cdes (outer) = 0.61, Cd1 = 20, Alpha*_inf = 1, Alpha_inf = 0.52, Beta*_inf = 0.09, a1 = 0.31, Beta_i (inner) = 0.075. The material or type of fluid used is water-liquid. For the velocity inlet section, the velocity magnitude is set based on previously determined velocity variations. Turbulent intensity is 5% and turbulent viscosity ratio = 10. Solution initialization uses the Hybrid Initialization method. In the "run calculation" section the settings will be as follows:

Tabel 2. Data setting run calculation

Time Stepping Method	Fixed
Time Step Size	1
Number of Time Step	0
Option	Data Sampling for Time Statistics
Sampling Interval	1
Max Iteration/Time Step	100
Reporting Interval	1
Profile Update	1

After the program is set based on the data above, a run calculation can be carried out by pressing the "calculate" section. Next, the research results can be set in the "results" section which will be presented in the form of an image to display changes in speed, pressure and shape of fluid flow around the SPAR based on the speed variations that have been modeled. After the modeling results are obtained, an analysis of the modeling results will then be carried out to determine the changes that occur in the fluid flow around the SPAR based on the speed variations that have been tested and to provide final conclusions from the research.

3. RESULT AND DISCUSSION

3.1 SPAR Modelling

CFD software aims to simulate physical processes, by capturing physical phenomena problems into numerical equations. Therefore, there is no guarantee that there will be a 'steady state convergence' solution to a problem. CFD simulation problems are generally non-linear and the solution technique uses an iterative process to obtain a solution until 'convergence' is achieved [3].
To carry out CFD-based numerical modeling analysis of fluid flow around the SPAR of offshore wind turbines, ANSYS Fluent software is used. Following are the modeling steps:

1. Geometry Data Preparation

Import the 3D model file of the SPAR--3VP offshore wind turbine that has been prepared previously using CAD software.

2. Grid Arrangement (Meshing)
Determine the type of grid to be used, such as structural or unstructural grid.
Generate a grid that suits the desired geometry and type of analysis. Make sure the grid is of good quality to avoid numerical errors.

3. Setting Boundary Conditions
Boundary conditions can be specified in the “setup” section. In this section, boundary conditions for the model will be determined, such as inflow velocity, turbulence model, and fluid density.

4. Physics Model Settings
The physics model is adjusted to the model to be simulated. In this modeling, the IDDES SST k omega turbulence model will be used according to the data in the research methods section.

5. Solver Settings (Solver Setup)
The solver used is the Pressure-Based type and the Velocity Formulation section is set to "Absolute".
Configure solver settings, such as number of iterations, tolerance, and more.

6. Solution Initialization
Initialize the solution with appropriate initial guesses for variables such as velocity and pressure. This initialization allows the solver to start iteration from sufficient conditions. For initialization methods, select "Hybrid Initialitation".

7. Simulation (Run Simulation)

Run a simulation using predefined settings. Make sure there are no errors or poor convergence during the simulation run by checking the output.

8. Results Analysis (Post-processing)
After the simulation is complete, the simulation results are then analyzed. ANSYS Fluent has various tools for visualizing simulation data, such as flow profiles, pressure distribution, temperature distribution, and so on.
The results of the analysis will be used to draw conclusions from the research.

9. Results Documentation
Documentation is needed to be used as documentation of results in preparing reports and to provide conclusions from the simulations carried out.

Fluid velocity and pressure analysis in the development of offshore wind turbines is essential to ensure the reliability of turbine operation, the efficiency of the energy produced, and the safety of the structure. This analysis also helps in meeting environmental permitting and regulatory requirements before building and operating the turbine and helps in identifying the wind and wave loads that the offshore wind turbine and SPAR structure will encounter. In wind turbine development, numerical simulation plays a key role in the design of blades and turbine placement to maximize energy production [4]. The interaction between fluid flow (current) and the SPAR will cause the formation of different flow patterns behind the SPAR. The flow pattern follows the shape/geometry of the structure it passes through [5].

3.2 Flow velocity around the SPAR

V1 = 0.661 m/s

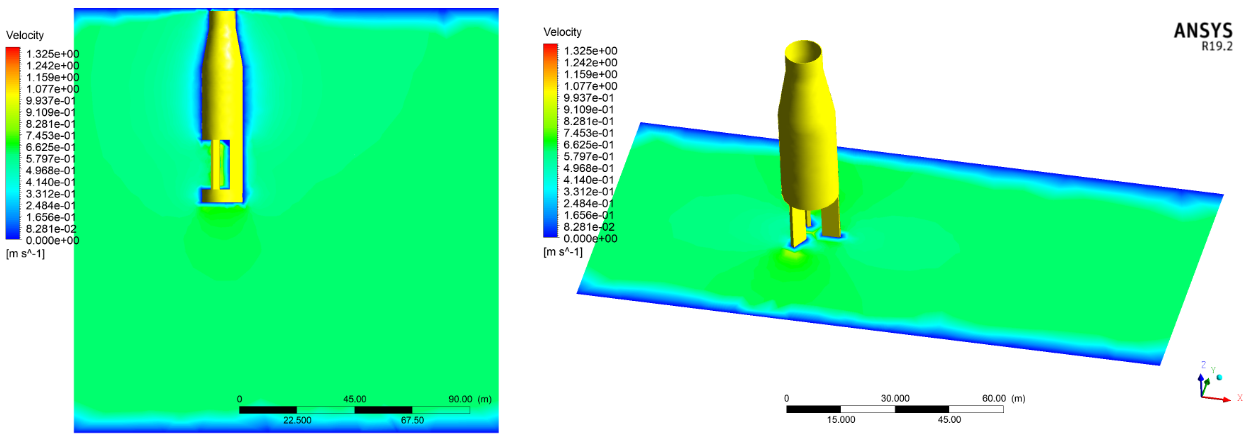

Figure 2. The fluid flow velocity around the SPAR in V1 conditions is 0.661 m/s.

V2 = 6.114 m/s

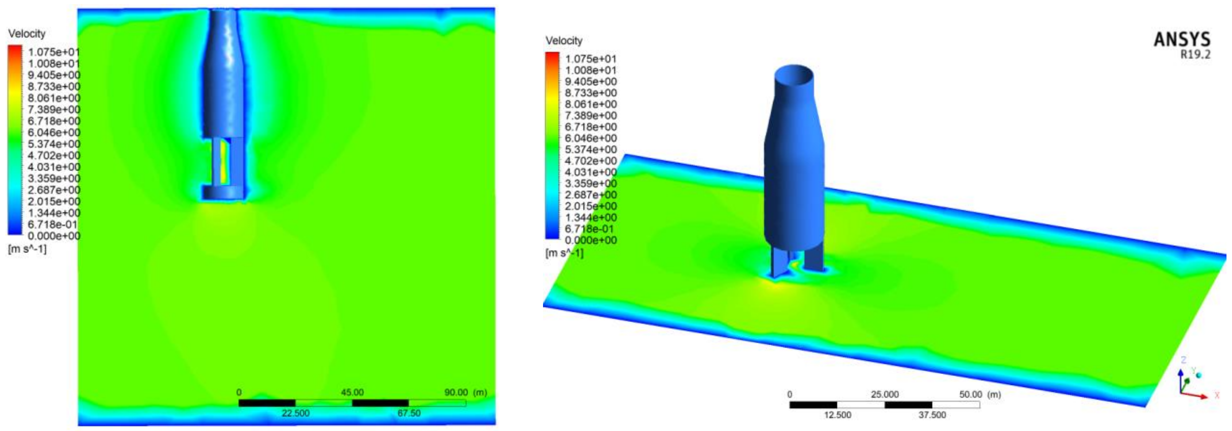

Figure 3. The fluid flow velocity around the SPAR in V2 conditions is 6,114 m/s.

- **V3 = 61.144 m/s**

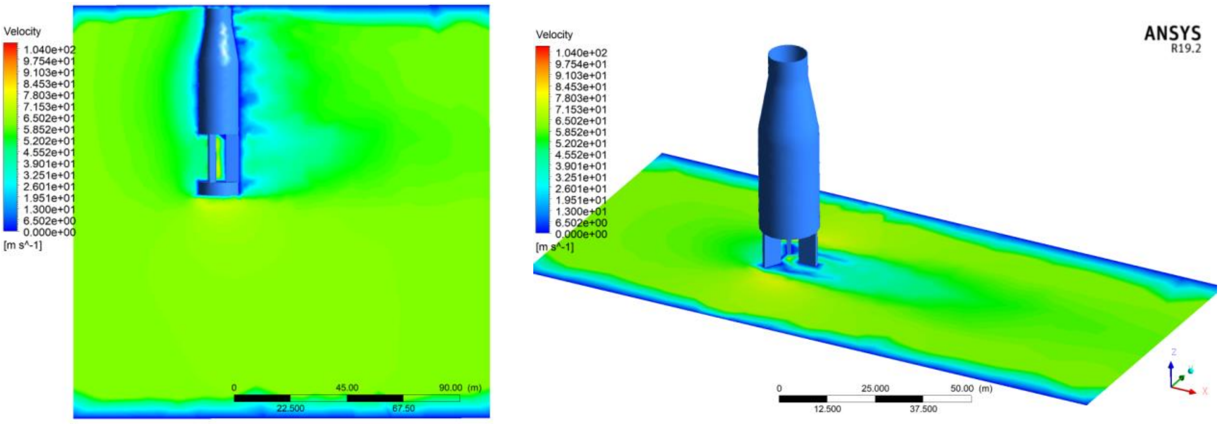

Figure 4. The fluid flow velocity around the SPAR in condition V3 is 61,144 m/s.

V4 = 611.438 m/s

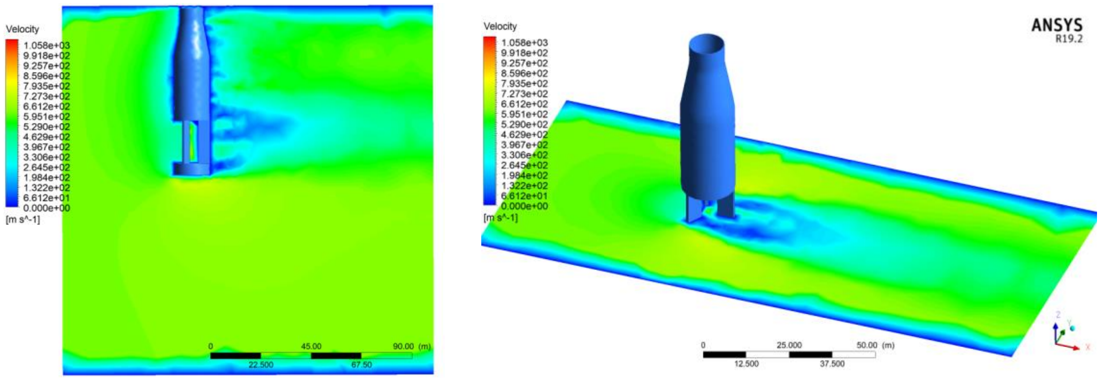

Figure 5. Fluid flow velocity around the SPAR in V4 conditions is 611,438 m/s.

From the simulation data above, the maximum velocity value of the fluid flow around the SPAR is obtained as follows:

Table 3. The maximum velocity value of fluid flow around the SPAR

Current speed (m/s)		**Current speed (m/s)**
V1	0.661	1.366
V2	6.114	11.084
V3	61.14	107.289
V4	611.438	1091.02

3.3 Flow pressure around the SPAR

V1 = 0.661 m/s

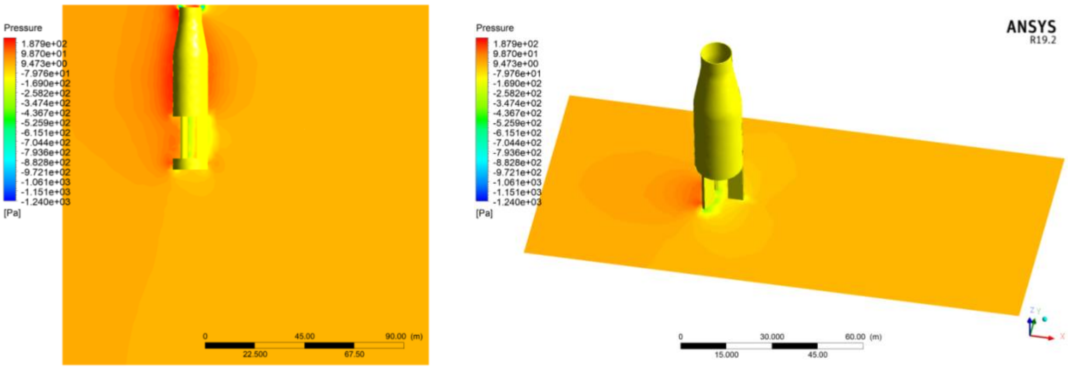

Figure 6. The fluid flow pressure around the SPAR at V1 conditions is 0.661 m/s.

- **V2 = 6.114 m/s**

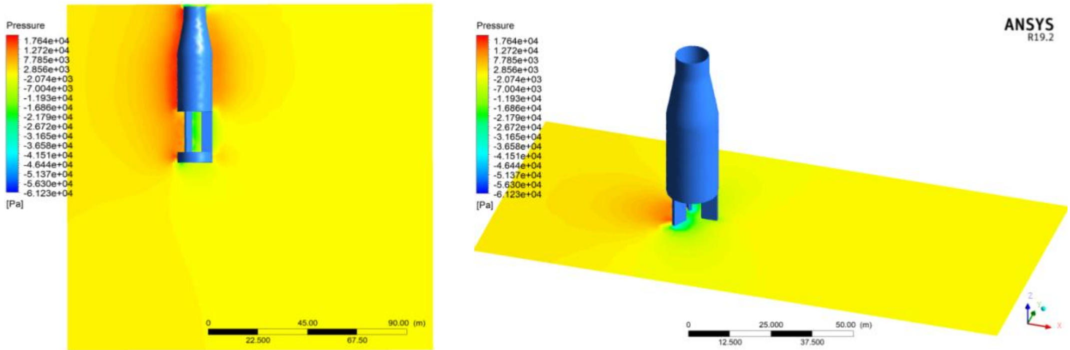

Figure 7. The fluid flow pressure around the SPAR at V2 conditions is 6,114 m/s.

- **V3 = 61.144 m/s**

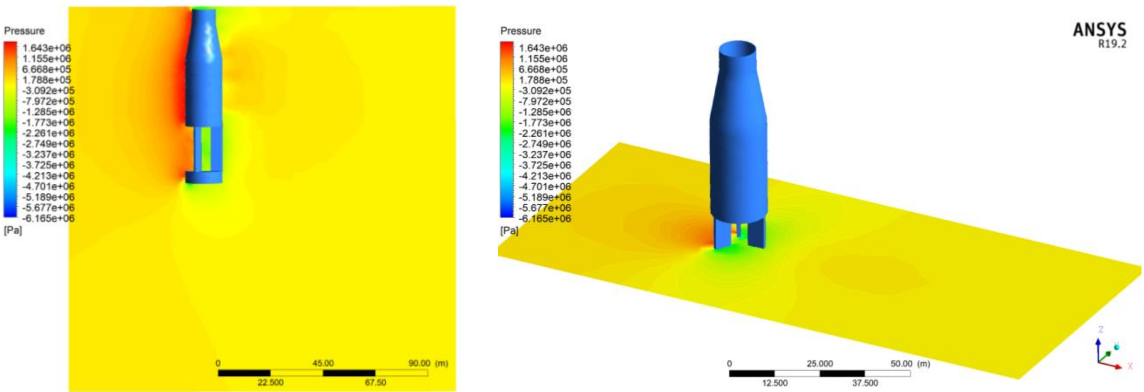

Figure 8. The fluid flow pressure around the SPAR in condition V3 is 61,144 m/s.

- **V4 = 611.438 m/s**

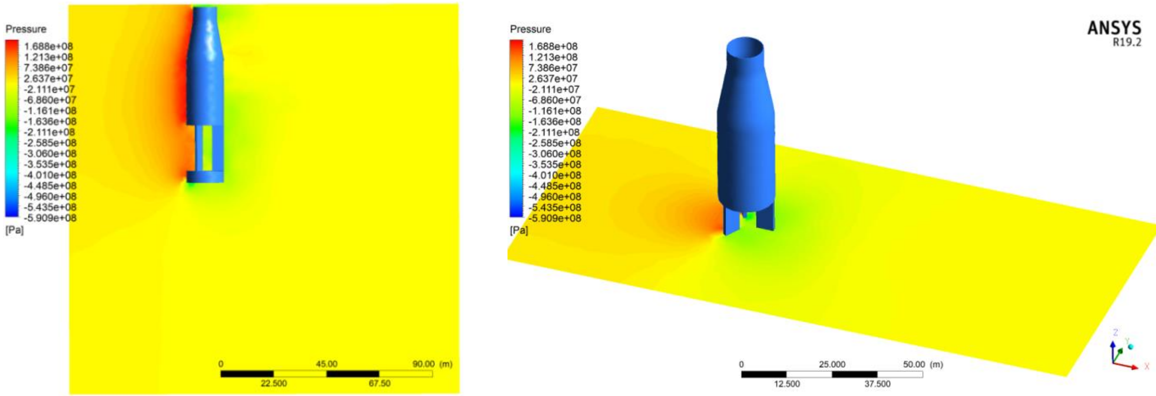

Figure 9. The fluid flow pressure around the SPAR in V4 conditions is 611,438 m/s.

From the simulation data above, the maximum pressure value (max pressure) for the fluid flow around the SPAR is obtained as follows:

Table 4. The maximum pressure value of the fluid flow around the SPAR

Current speed (m/s)		Current speed (m/s)
V1	0.661	232.549
V2	6.114	20109.8
V3	61.14	1.88677×10^6
V4	611.438	1.92571×10^8

4. CONCLUSION

Based on data from simulation results of pressure and fluid flow velocity around the SPAR with different variations in flow velocity, the following conclusions are obtained: 1) The simulation results show that the fluid flow around the SPAR is very sensitive to changes in flow velocity. Increasing flow velocity can result in significant changes in parameters such as pressure and maximum velocity. 2) Data from the simulation shows that the effect of changes in velocity in the fluid flow around the SPAR tends to be exponential. This indicates that small changes in velocity can produce large changes in fluid flow parameters. 3) Simulations show that the maximum pressure in the fluid flow around the SPAR can reach very high levels when the flow velocity reaches a significant level. This can be observed in the V4 611,438 m/s simulation, which shows changes in pressure and fluid flow velocity around the SPAR which increase significantly when compared to the previous 3 simulations.
This research has potential practical applications in the design and operation of SPARs, particularly in optimizing fluid flow velocity to manage existing pressure and ensure safe and efficient performance. This research is still limited by the parameters used and the lack of understanding in operating the ANSYS Fluent software. Therefore, more in-depth research is needed to obtain maximum results. Model testing can be one way to obtain more valid data results by comparing them with test results using numerical methods (CFD).

REFERENCES

[1] A. Randa, “Permodelan Unjuk Kerja Turbin Pusaran (Vortex) Untuk Model Alat Uji Pembangkit Listrik Menggunakan Software Ansys Fluent”, Fakultas Teknik, Universitas Lampung, 2023.
[2] F. M. Assidiq, et. al., “Influence of Vertical Plate on the Pitching Motion of a SPAR Wind Floater in Waves”, CCORE 2022, SPESS, pp. 30-42, 2023.
[3] F. M. Assidiq, et. all., “Experimental Investigation on the Characteristics of Pitch Motion for a Novel SPAR type FOWT in Regular Waves”, Kapal: Juarnal Ilmu Pengetahuan dan Teknologi Kelautan, 20(2), 163-174, 2023.
[4] Sorensen, J. Norker, and J. K. Madsen. "Unsteady aerodynamics experiment II." Wind Energy 6.3: 219-240, 2003.

[5] S. Achmad, “Analisis Pengaruh Variasi Konfigurasi Helical Strakes pada Aliran Fluida dan Gerakan Struktur SPAR”, Teknik Kelautan, Institut Teknologi Sepuluh Nopember, 2018.