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 Fluid Flow Conditions And Pressure Around Spar With Five Vertical Plate Damping System

*Dimas Fitrawan and Fuad Mahfud Assidiq
Department of Ocean Engineering, Faculty of Engineering, Hasanuddin University, Indonesia
*Email: dimasfitrawan30@gmail.com

Abstract

Spar type Floating wind Turbine is a floating structure that utilizes the speed of the sea wind to drive a turbine that can generate electrical energy .SPAR is a cylindrical floating structure that has a deep draft and is reliable in operating in the deep sea for a long time.in this study using a SPAR model with a damping system of five vertical plates located at the bottom of the SPAR structure where in this study the analysis about fluid flow conditions and pressures that are around the SPAR.this study was conducted by calculation numerical based CFD (Computational Fluid Dynamics) with four variations of fluid flow velocity used to observe changes in fluid flow conditions and pressure around the SPAR.this study aims to observe the conditions and influences around the SPAR type floating wind turbine structure which is expected to be a solution to energy problems in the world.

Keyword : CFD, FLUIDA, SPAR

1. INTRODUCTION

The increase in energy demand causes increased consumption of hydrocarbons as raw materials for energy production. Because exploration and exploitation of hydrocarbons, especially in shallow sea areas, is becoming increasingly scarce, exploration and exploitation of hydrocarbons is moving towards the deep sea. With the shift to exploration and exploitation in the deep sea, a platform structure is needed that is able to withstand the environmental conditions in the deep sea. An offshore platform is a structure or building built offshore. According to the type of structure, offshore platforms are divided into three groups, namely, piled structures (fixed structure) or also known as a pile-supported platform, floating structure (floating structure) and flexible structure (compliant structure). SPAR is a type of offshore platform in the form of a floating production unit (floating structure) in the form of a vertical cylinder (single column) with a fairly deep draft. The vertical cylinder primarily functions as a deck support (deck)[1]. SPAR is a floating structure that is reliable for operating in the middle of the sea for long periods of time in extreme environmental conditions. Because it will operate in deep waters (deep water) with a long period of time, it is necessary to carry out various analyzes in planning [2]

A spar type floating wind turbine is an example of a renewable energy source that utilizes sea breeze speed to drive a turbine that can produce electrical energy. Floating turbine wind This type of SPAR is a floating structure that utilizes buoyancy to support facilities on the deck to produce renewable energy that utilizes sea breeze speed. The vertical design of the SPAR with a very large draft can increase stability which can reduce the possibility floating turbine wind The reverse SPAR type and spar design are also very efficient because they can be placed in deep water [3]. 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 complicated so special equipment and expertise are needed to make it.

This research focuses on the flow and pressure conditions that occur around the SPAR. Despite accelerating commercial viability, offshore platforms are subject to high fluctuations in current loading. Complex structural interactions can cause difficulties for the SPAR in absorbing current impacts and also in the post-impact section have an impact on changes in flow turbulence. This phase difference will adjust the behavior of the substructure because the power produced fluctuates [4]. The influence of fluid flow velocity can create flow turbulence and provide dynamic pressure on the SPAR structure. Several modifications to the existing substructure such as stepped SPARs, balancing discs and heave plates are offered as alternatives. Initially, the stepped SPAR model showed many advantages over the basic SPAR model, including acceptable hydrodynamic performance based on turbulent winds. Vertical plates are the most advanced SPAR damping system solution. This is due to the fact that damping of pitch motion is more important than damping of heave motion. It is possible to realize sufficient reduction of pitch motion through the use of five vertical plates instead of heave plates[5]. So the analysis of fluid flow conditions and pressure around the SPAR is greatly influenced by variations in speed, structural design and the 5 vertical plate damper system located at the bottom of the spar structure. This analysis will produce parameters such as flow shape and pressure around the spar based on the influence of four speed variations.

2. METHOD

In researching the SPAR model (5VP) using the 3 D modeling method using numerical calculations in the form of CFD (computational fluid dynamics) where this research aims to analyze fluid flow and pressure conditions around the spar based on four variations in fluid flow velocity, namely V1=0.611 m/s, V2=6.114 m/s, V3=61.144 m/s and V4=611.438 m/s. Furthermore, the turbulence used is DES-SST (Detached Eddy Simulation) with Shear-Stress K omega transport with the material used in this simulation is that the solid type uses aluminum material and the fluid type uses water-liquid material. At the boundary conduction stage, zone inlet is used

As the direction of fluid flow entry with a turbulence insensitivity of 5%, the solution stage is used pressure velocity scheme SIMPLE with turbelency kinetic and specific Dissipation rate, namely secondary order upwind, data solution controls that is pressure = 0.3, density=1, body forces=1, momentum=0.7 and turbelency kinetic=0.8. Next on solution initialization use metode hybrid instalazation.

Next, the data that has been input is then entered calculate and the research results will be presented in the form of image visualizations showing changes in speed, pressure and shape of fluid flow around the SPAR based on variations in fluid flow velocity. Next, analysis of the modeling results is carried out to determine the conditions of changes that occur in fluid flow and pressure around the SPAR based on speed variations that have been tested to provide conclusions for this research.

3. RESULT ANDA DISCUSSION

The SPAR 5VP modeling was carried out using CFD simulation (computational fluid dynamics) through ANSYS Fluid Flow (Fluent) where in the SPAR 5VP model there are 5 vertical plates which function as dampers which influence fluid flow conditions and pressure around the SPAR. The following are the stages in the SPAR 5VP modeling simulation

1. Geometric modeling At this stage the SPAR 5VP model modeled in CAD software is imported into ANSYS Fluid Flow (Fluent) at this stage, the boundaries of the simulation area (Domain) are determined with coordinates +x=100 m,-x=50 m,+y=30 m,-y=30 m,+z=1 m and -z=90 m.

2. The mesh stage is where the path conforming method is determined with the inlate, outlate, wall and spar sections which will become objects in the simulation later.

3. The Setup stage begins with the general model viscous Detache Eddy Simulation (DES) k-omega with water-liquid material, at boundary conditions with a velocity type inlate zone with variations in speed, namely V1=0.611 m/s, V2=6.114 m/s, V3= 61.144 m/s and v4=611.438 m/s. In the solution a SIMPLE scheme is used with kinetic energy turbulence and specific dissipation rate in the form of second order upwind. Then the solution hybrid initialization and run calculation are carried out with 100 time steps/max iterations.

4. In the results stage, simulation modeling locations are determined to obtain a visualization of the flow and pressure conditions around the SPAR based on 4 variations in fluid flow velocity. This visualization will later describe the conditions and pressure that are visible from above and from the side.

SPAR 5VP Modelling Simulation Result

Fluid Flow Conditions Around SPAR 5VP

- Simulation results at speed V1=0.611 m/s

Side view Top view

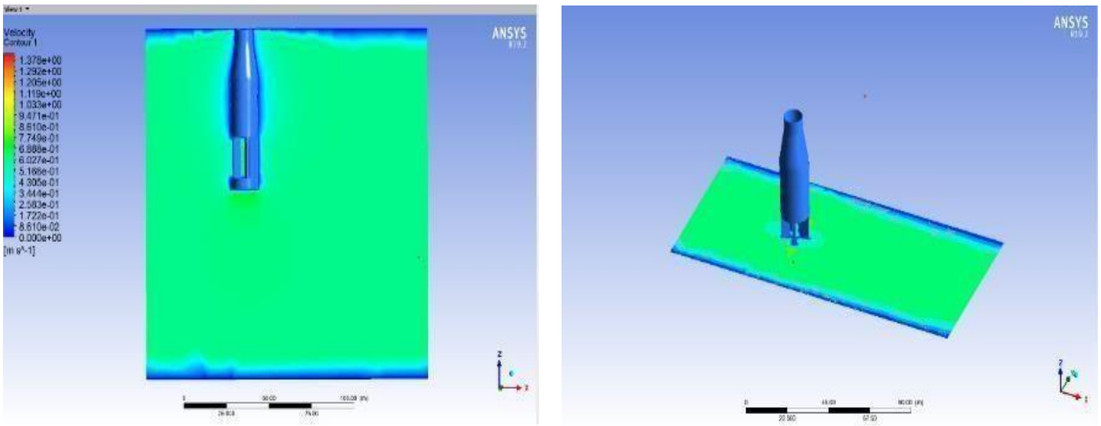

Figure 1. Visualization of fluid flow conditions around SPAR at speed v1=0.611 m/s.

- Simulation results at speed V2=6,114 m/s

Side view Top view

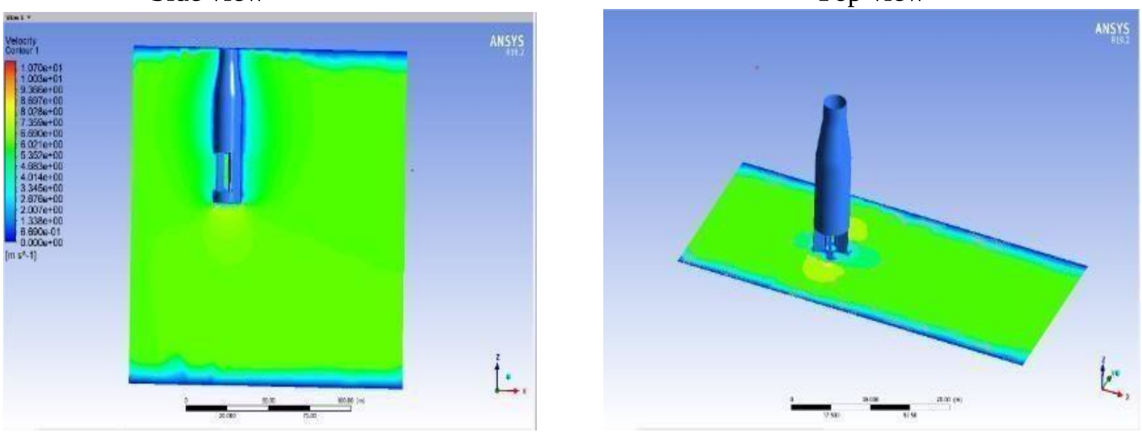

Figure 2. Visualization of fluid flow conditions around SPAR at speed v2=6,114 m/s.

❖ Simulation results at speed V3=61,144 m/s

Side View Top View

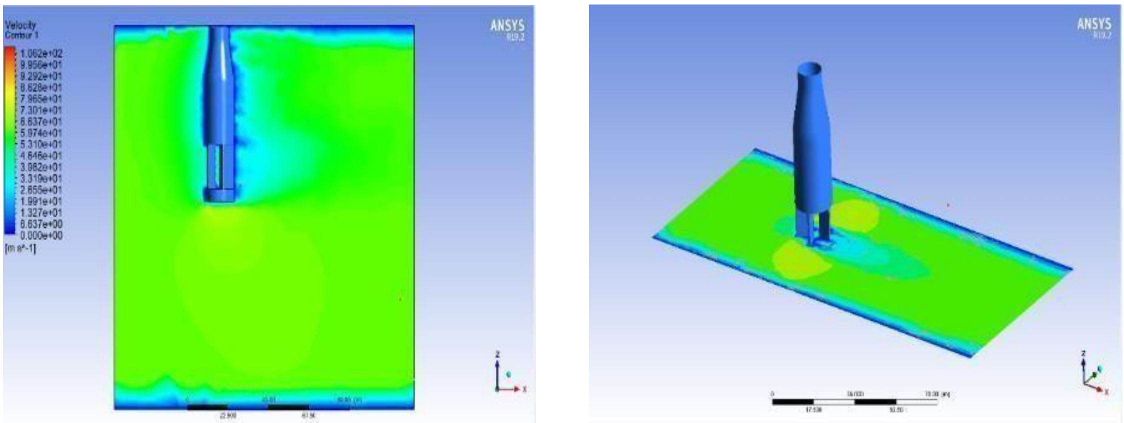

Figure 3. Visualization of fluid flow conditions around SPAR at speed v3=61,144 m/s..

❖ Simulation results at speed V4=611,438 m/s

Side View Top View

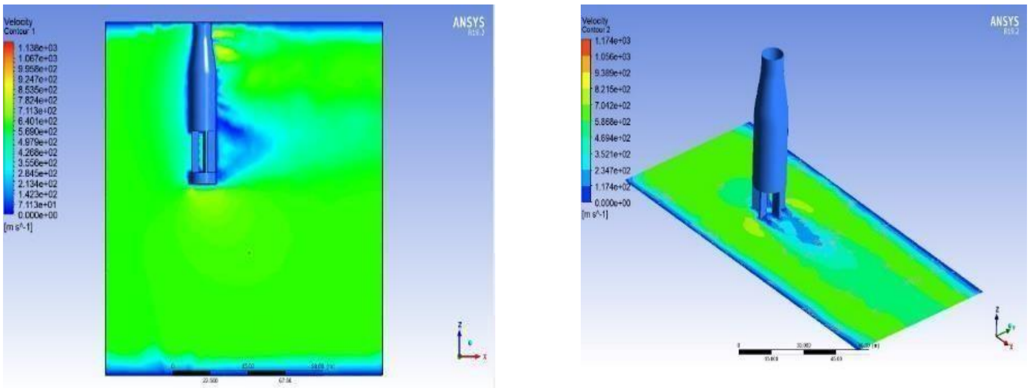

Figure 4. Visualization of fluid flow conditions around SPAR at speed v4= 611,438 m/s.

Pressure conditions around SPAR 5VP

❖ Simulation results at speed V1=0.611 m/s

Side view Top view

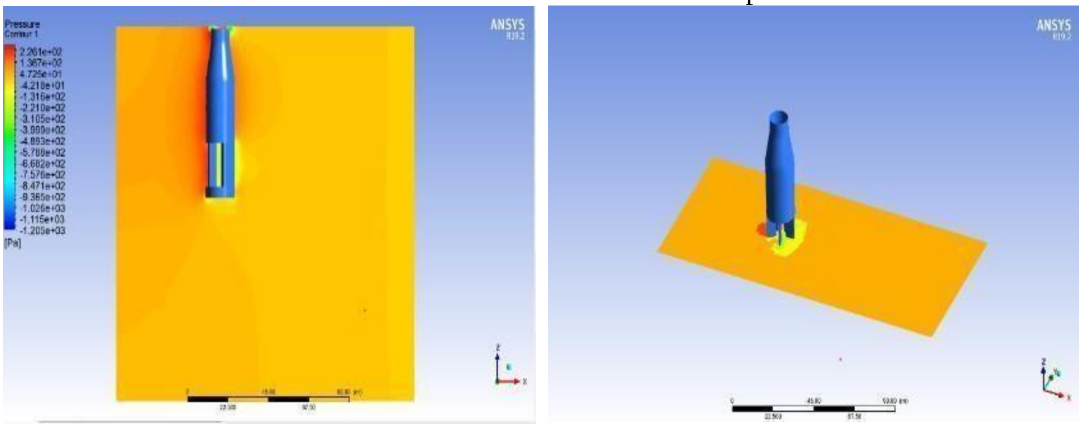

Figure 5. Visualization of Pressure around SPAR at speed v1=0.611 m/s

- Simulation results at speed V2=6,114 m/s

Side view Top view

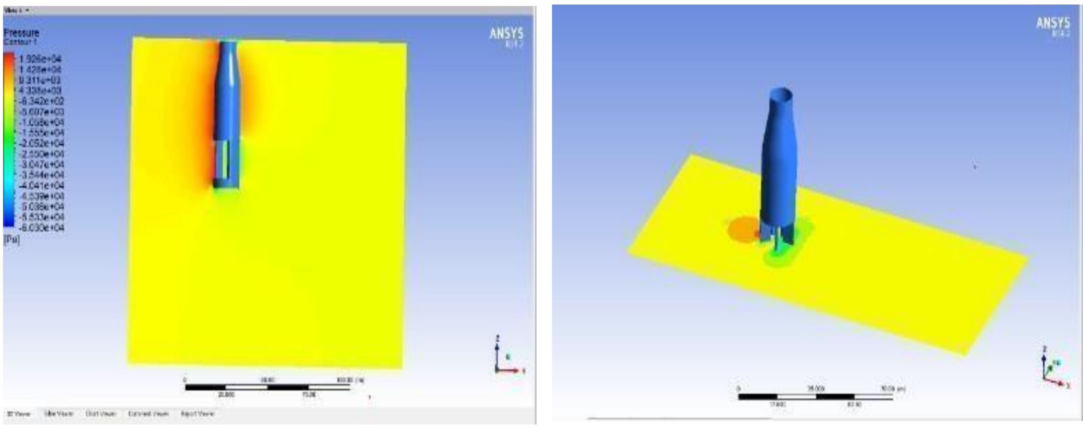

Figure 6. Visualization of pressure around SPAR at speed V2=6,114 m/s.

- Simulation results at speed V3=61,114 m/s

Side view Top view

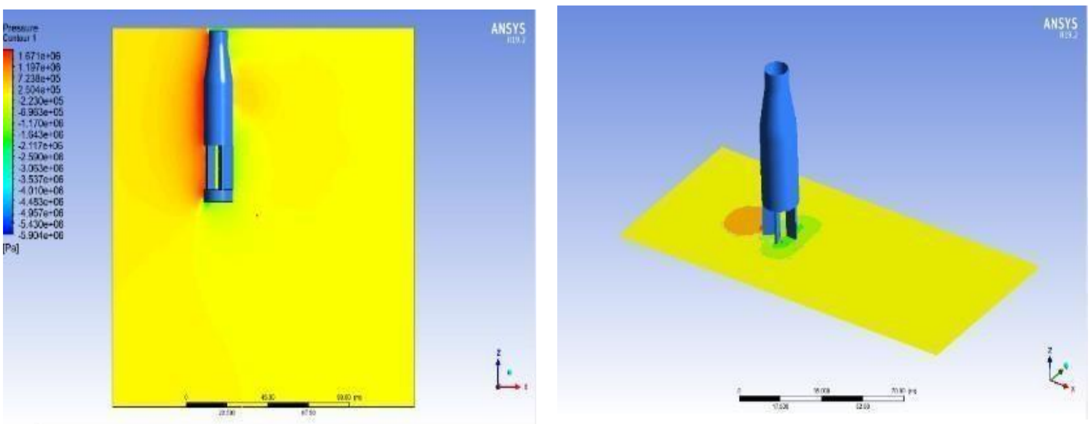

Figure 7. Visualization of pressure around SPAR at speed V3=61,114 m/s.

❖ Simulation results at speed V4=611,438 m/s

Side view Top view

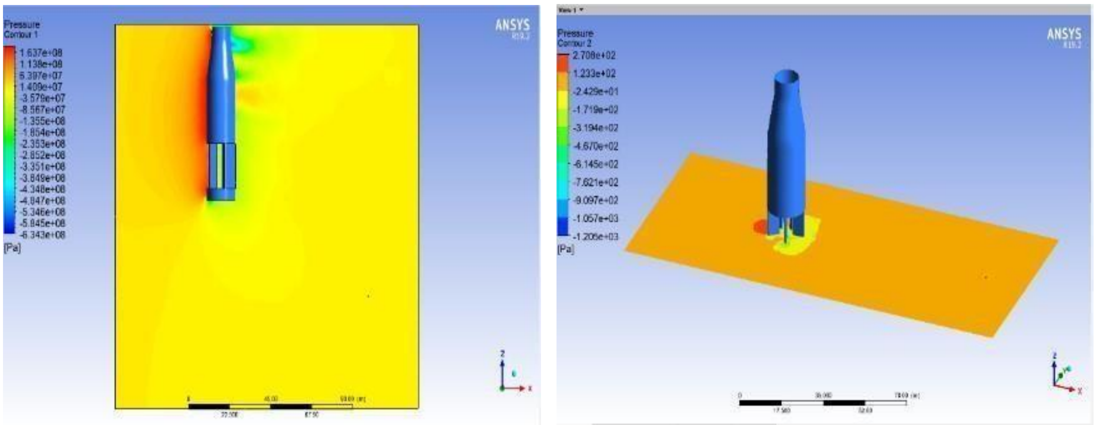

Figure 8. Visualization of pressure around SPAR at speed v4=611,438 m/s.

From the simulation test results for fluid flow conditions and pressure, it can be seen from the visualization in the eight images that the fluid flow and pressure conditions around the SPAR structure are greatly influenced by variations in speed, the SPAR structure and the five-plate damping system located at the bottom of the SPAR structure.

4. CONCLUSION

Based on data from simulation results of fluid flow pressure around the SPAR with different speed variations, we can make several conclusions:

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 velocity changes 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] Indiyono, P. 2004.'' Hidrodinamika Bangunan Lepas Pantai''. Surabaya: SIC.
[2] Matos, V. L. Ribeiro, E. O., Simos, A. N. 2010. 2nd Order''Pitch and Roll Motionsof a Semi-Submersible Platform'' Wind Energy 6.3 (2003): 219- 240, 2003.
[3] Soedjono, J. J. 1998. ''Diktat Mata Kuliah Konstruksi Bangunan Laut II''.
[4] Surabaya:Jurusan Teknik Kelautan ITS.
[5] Assidiq, Fuad M., et. Al.,''infuence of vertical plate in the pitching motion of a SPAR Wind Floater

inWaves''. CCORE 2022, SPESS, PP,30-40,2023.
[6] Assidiq, Fuad M., et. Al., Experimental Investigation on the Characteristics of Pitch Motion for a Novel SPAR type FOWT in Regular Waves'' . Kapal: Jurnal Ilmu Pengetahuan dan Teknologi Kelautan, 20 (2) (2023):163-174, 202