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 Flow Changes In The Spar Model (4vp)

*Nurhikma, dan Fuad Mahfud Assidiq
Department of Ocean Engineering, Faculty of Engineering, Hasanuddin University, Indonesia
*Email: nurhikman20d@student.unhas.ac.id

Abstract

Research on the flow around the SPAR model was carried out with the aim of finding out how much effect the flow has on the SPAR structure used in offshore wind turbines. SPAR is a type of floating structure that can usually be used in deep seas and is an effective and reliable structure in terms of construction. Spar has the advantage that the buoyancy point (KB) is above the gravity point (KG), thus providing good stability. However, there are still obstacles to the use of SPAR structures in the deep sea, especially when environmental conditions are extreme, where SPAR structures are not able to reduce currents at a certain speed level. Then, this research aims to test the SPAR model with a four vertical plate (4VP) configuration system which is then modeled using Computational Fluid Dynamics (CFD) based numerical modeling which aims to investigate the flow around the SPAR in terms of flow velocity. Especially at certain flow velocity levels, the velocity is chosen carefully at each level starting from: V1 = 0.611m/s, V2 = 6.114, V3 = 61.144, V4 = 611.438. The addition of vertical plates to the bottom construction of the SPAR has a role in dampening the flow and also turbulent effects due to current impacts.

Keyword: CFD, Fluida, Offshore wind turbine, SPAR, Turbulensi.

1. INTRODUCTION

Spar is a type of floating structure that can be used in deep seas and is an effective and reliable structure in terms of construction. Spar has the characteristic that the buoyancy point (KB) is located above the gravity point (KG), thus providing good stability. Spars have weaknesses, namely low damping and natural period, so it is necessary to add a heave plate to the spar platform [1]. The spar structure concept allows installation in water depths of more than 100 m. The top of the structure is lighter than the bottom, thereby raising the center of buoyancy. To achieve static stability, he
using ballast weights placed low on the buoy, so that the center of gravity is lower than the center of buoyancy. Therefore, it provides high resistance to pitch and roll rotational movements[2]. In contrast to semi-submersibles and TLPs, SPAR substructures are often chosen because of their simple design and excellent stability. In addition, it has a small water plane area, deep air flow will reduce wave loads, and the additional benefits of natural periods.

Despite their rapidly increasing commercial viability, floating platforms have high fluctuations in current loading. The interaction between structures and structures is mostly complicated, which can cause difficulties for SPARs in absorbing current impacts and also in the post-impact section, it has an impact on changes in turbulent flow. This phase difference will then adjust the behavior of the substructure because the generated power fluctuates [3].

Several modifications to the existing substructure such as stepped SPARs, balancing discs and vertical plates are offered as alternatives. Initially, the stepped SPAR model showed many advantages over the basic SPAR model, including acceptable hydrodynamic performance based on wind turbulence, use of a 4 x 1 mooring pattern as an active damping pitch, and the possibility of being applied in temperate waters for potential savings. Also, compared to without the use of discs, balancing disc technology can actually increase mass, damping, and additionally reduce shock movement. Heave plates, like previous technologies, are a quick fix for increasing vertical damping but are best suited for FPSOs or mono column structures. All in all it is almost impossible to reduce large wave movements inadequately. Based on the author's best expertise, vertical plates are the most advanced SPAR damping solution. This is due to the fact that the damping of the pitch motion is more important than the damping of the heave motion. It is possible to realize a sufficient reduction in pitch motion through the use of four vertical plates instead of the heave plate, as strongly suggested in their investigation[4]. However, if the four vertical plates are not equated with the number of other vertical plates, then their function becomes ambiguous. On this basis, this research aims to study the flow conditions around the SPAR by considering fluid velocity and pressure.

2. RESEARCH METHOD

The model used is a spar with a four vertical plate (4VP) configuration system which is then modeled using numerical modeling based on Computational Fluid Dynamics (CFD) which aims to investigate the flow around the SPAR in terms of flow velocity. Especially at certain flow velocity levels, the velocity is chosen carefully at each level starting from: V1 = 0.611m/s, V2 = 6.114, V3 = 61.144, V4 = 611.438. With a viscous model using Detached Eddy Simulation (DES), RANS SST k_omega model with IDDES turbulent Shielding Function type with a turbulent intensity ratio of 5%. Then for the solution for testing the model using the SIMPLE scheme with spatial discretization of Leats Square Cell Based gradients, Second Order Pressure, Momentum Bounded Central Differentiation, Turbulent Kinetic Energy Scene Order Upwind, Dissipation Rate Second Order Upwind. Then, further testing is carried out by reviewing the results of the ANSYS Fluid Flow (Fluent) calculations.

3. RESULT AND DISCUSSION

3.1. SPAR Modeling

SPAR modeling in this research uses ANSYS Fluid Flow (Fluent) which starts with input geometry that has previously been engineered using AUTOCAD.

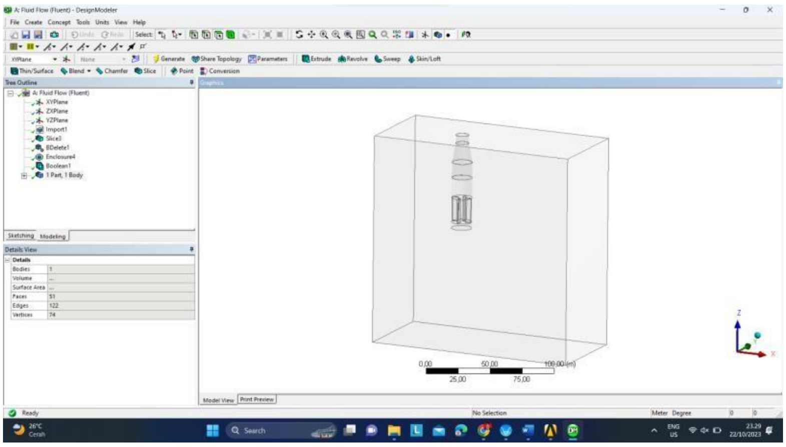

Figure 1. Geometry and model testing domain.

After geometry input, then enter the domain value as a SPAR model testing container with values including; X+ = 100 m, X- = 50m, Y+ = 30m, Y- = 30m, Z+ = 1, Z- = 90. Then enter the setup to input the coefficients in the SPAR test..

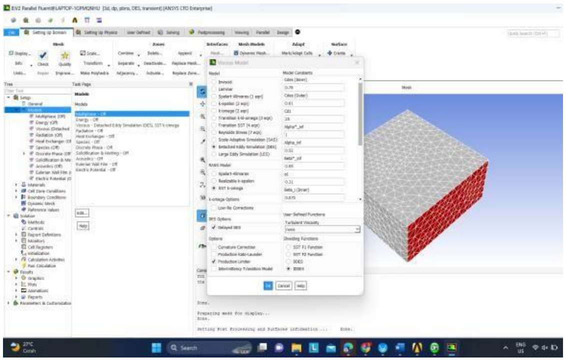

Figure 2. Models Viscous

For the model we use "Detached Eddy Simulation" then for the RANS Model we use "SST k-omega" with Sheilding Functions "IDDES"

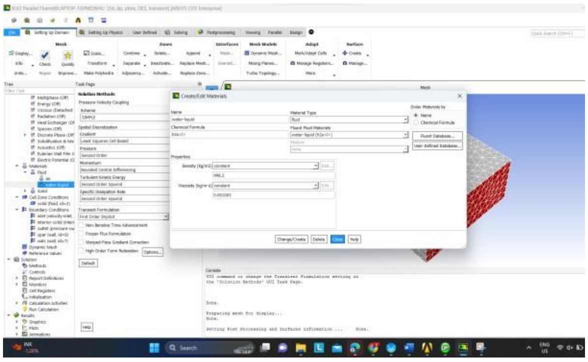

Figure 3. Input materials used

With materials using "Water-Liquid"

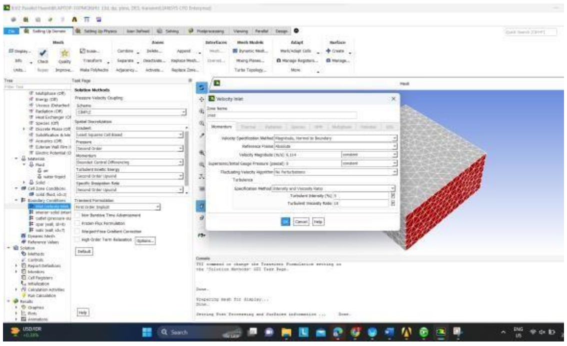

Figure 4. Inlet velocity

Then for Boundary Condotions, input the velocity magnitude with variations in predetermined values as test material, starting from V1 = 0.611, V2 = 6.114, V3 = 61.144, V4= 661.438.

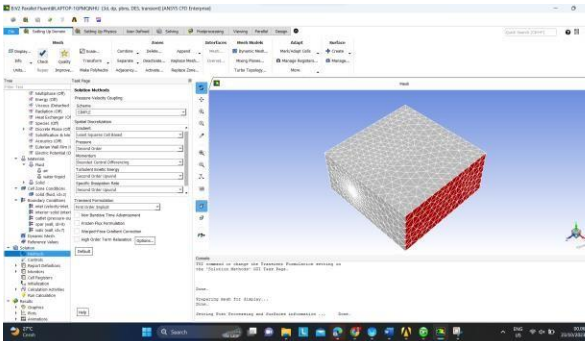

Figure 5. Solution methods

For the method itself, ANSYS Fluend Fluid Flow (Fluent) uses the "Simple" scheme with the Hybrid Initialization method installation.

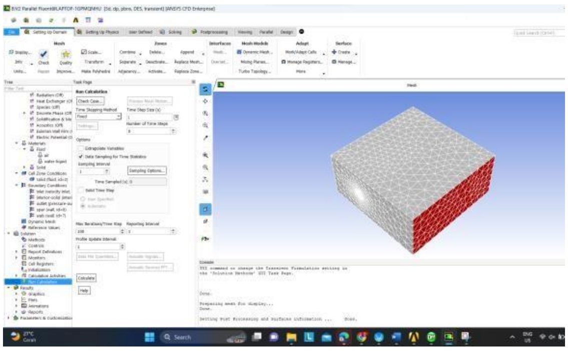

Figure 6. Run calculation

Enter Run Calculation with “Max Iterations/Time Step” using an interval of 100.

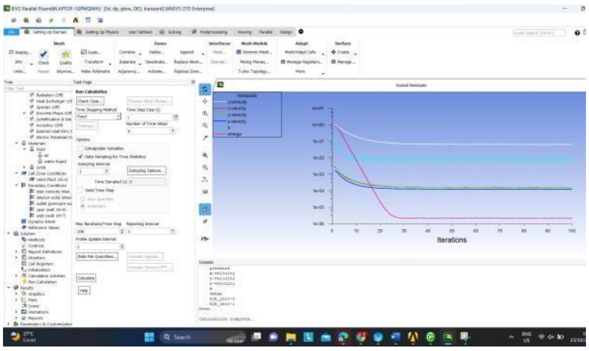

Figure 7. Iteration results

So it produces an iteration graph with an interval of 100, then the test will be the same
This is done to get iterations with different speed variations.

3.2. Flow velocity around the SPAR

The flow conditions after hitting the SPAR model experience changes as well as the flow velocity which changes after passing through the SPAR model where the speeds used are V1, V2, V3, V4, based on the results of speed testing it shows that the addition of vertical plates to the bottom structure of the SPAR can be said to be efficient in reduce the speed of the current hitting the SPAR by looking at the shape of the flow after the SPAR, especially at flow speed V4 = 611.438 m/s.

3.2.1. Speed V1 = 0,611 m/s

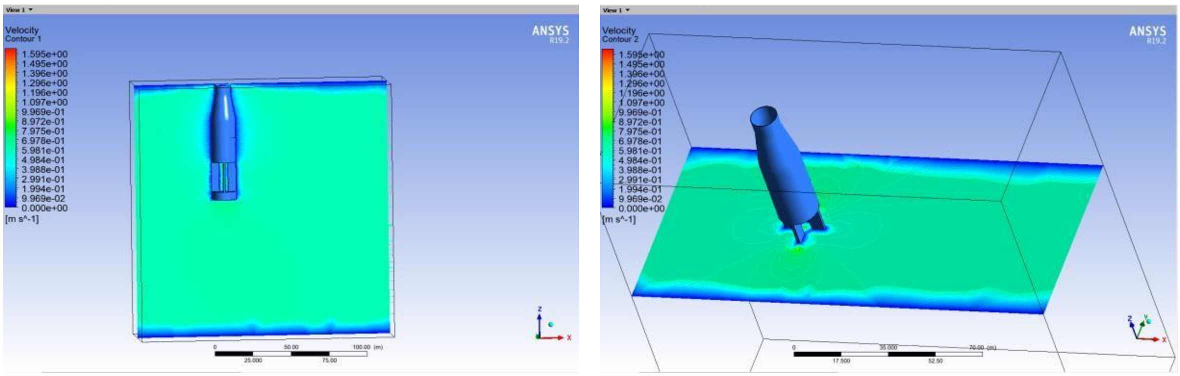

Figure 8. V1 speed test results. Max velocity = 1.64886 m/s

3.2.2. Speed V2 = 6,114 m/s

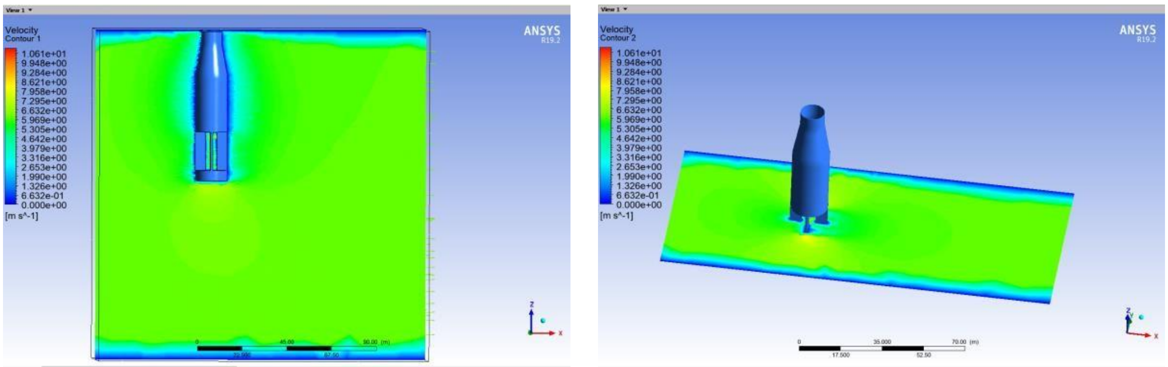

Figure 9. V2 speed test results. Max velocity = 10.9423 m/s

3.2.3. Speed V3 = 61,144 m/s

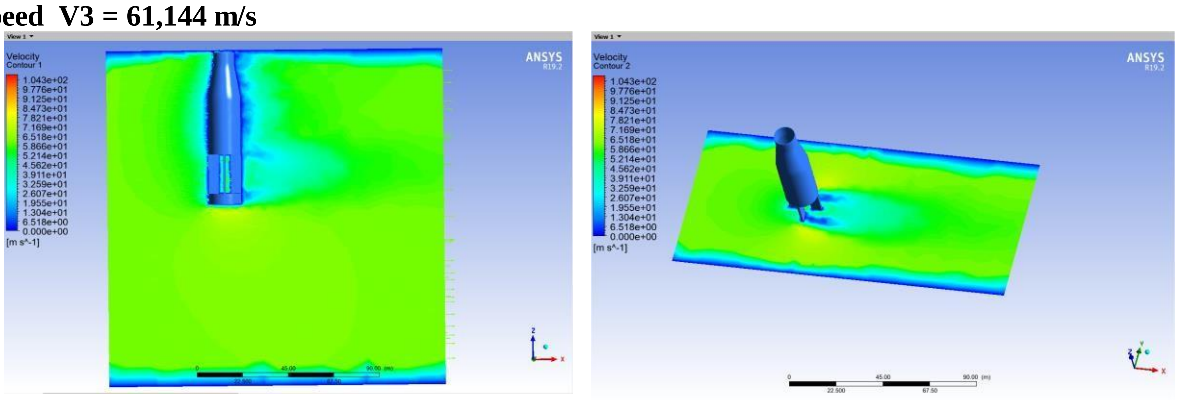

Figure 10. V3 speed test results. Max velocity = 107.541 m/s

3.2.4. Sp

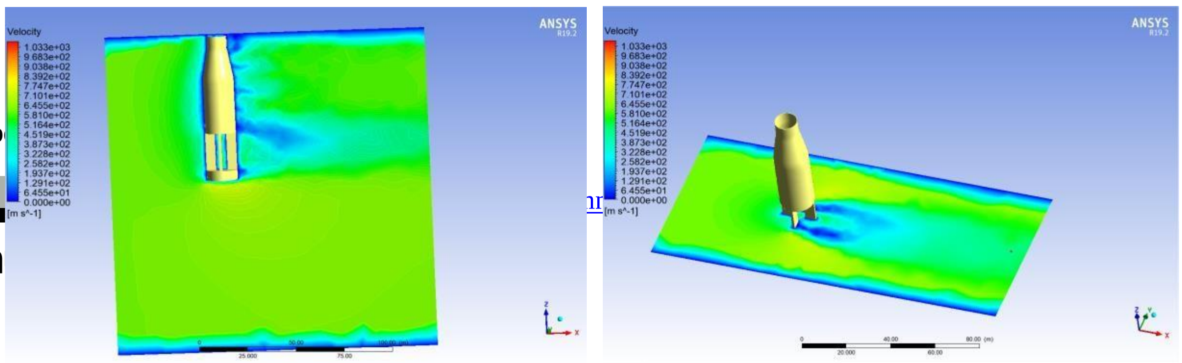

Figure 11. V4 speed test results. Max velocity = 1065.15 m/s

From the test results above, it shows that significant changes in flow are found in the flow velocity test V4 = 611.438 m/s with max velocity = 1065.15m/s, this shows that the addition of vertical plates is able to adjust to the flow speed. So from the simulation results above, the maximum velocity value (max velocity) of the fluid flow around the SPAR is obtained as follows:

Table 1. Maximum velocity value of fluid flow around the SPAR

Current speed (m/s)		Maximum speed (m/s)
V1	0,661	1,648
V2	6,114	10,942
V3	61,144	107,541
V4	611,438	1065,15

3.3. Flow Pressure Around the SPAR

The results of pressure testing on the SPAR model, where we see the SPAR response to flow velocity, show different pressure levels at each speed variation. The location of the pressure can be seen at the top or tubular part of the SPAR, different from the vertical plate, which shows that this area is able to reduce pressure, in other words, this area produces little pressure effect.

3.3.1. Pressure at speed V1 = 0.611 m/s

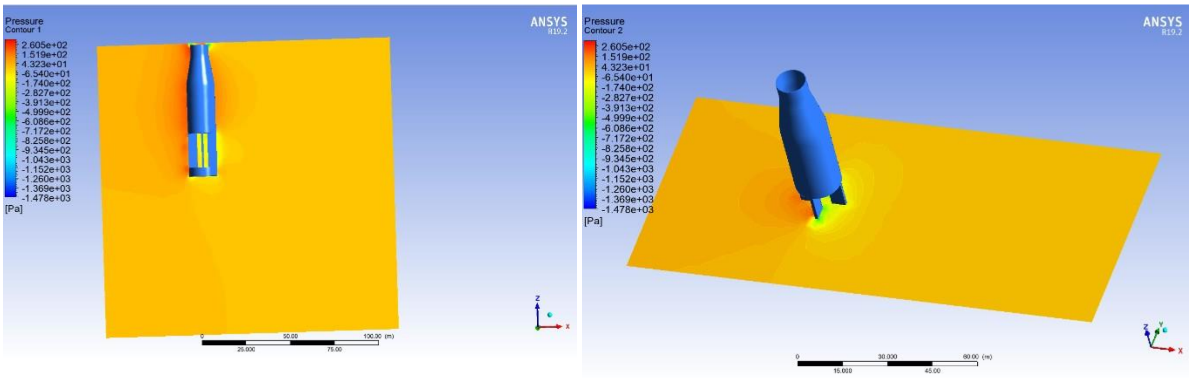

Figure 12. Pressure test results at speed V1.

3.3.2. Pressure at speed V2 = 6.114 m/s

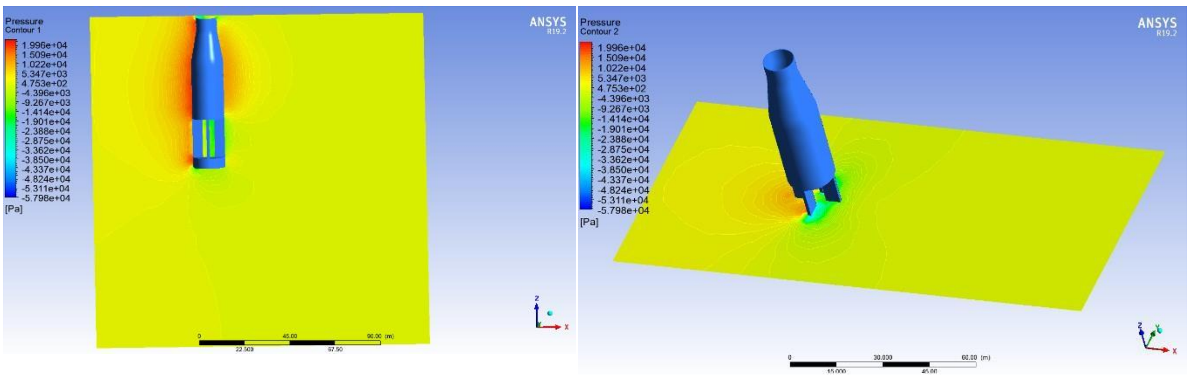

Figure 13. Pressure test results at speed V2.

3.3.3. Pressure at speed V3 = 61.144 m/s

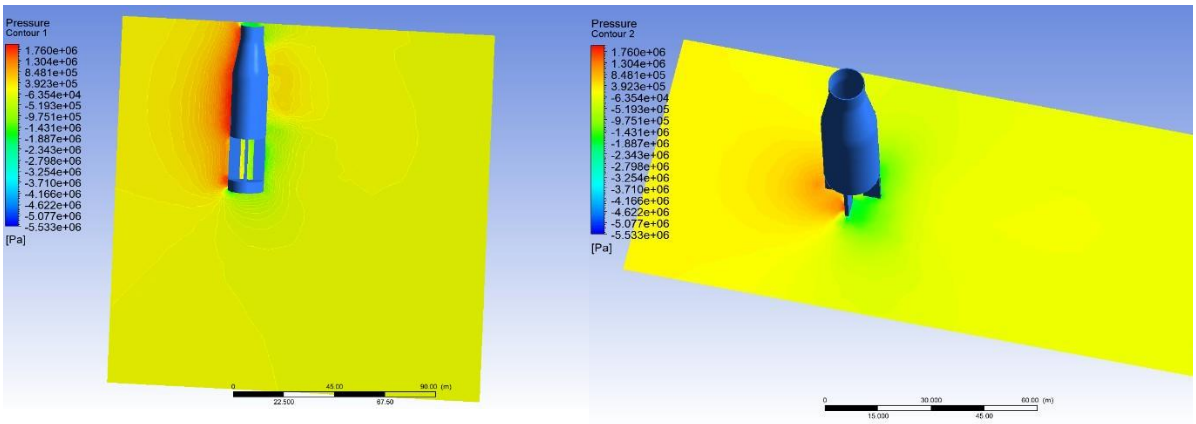

Figure 14. Pressure test results at speed V3.

3.3.4. Pressure at speed V4 = 611.438 m/s

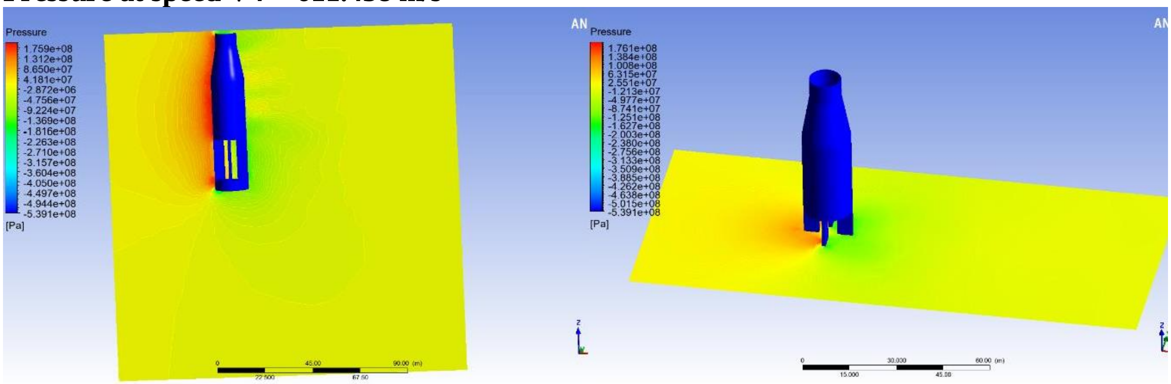

Figure 15. Pressure test results at V4 speed.

From the pressure test results above, we can see that the speed level greatly influences the pressure that impacts the SPAR due to current impact.

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 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] Assidiq, Fuad M., 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.

[2] Assidiq, Fuad M., et. al., "Influence of Vertical Plate on the Pitching Motion of a SPAR WindFloater in Waves", CCORE 2022, SPESS, pp. 30-42, 2023.

[3] Shafiee, Mahmood. 2022. "Failure analysis of spar buoy foating ofshore wind turbine systems". Innovative Infrastructure Solutions, 8-28.

[4] Vernanda, Adytia Z, 2018, "Analisis Respons Dinamis Floating Offshore Wind Turbine Tipe Semi-Sub Dengan Variasi Desain Wind Turbine 1,5MW, 2,5MW DAN 5 MW Untuk Perairan Natuna", InstituteTeknologi Sepuluh November, Surabaya.