Website : jurnal.umj.ac.id/index.php/icecream

Numerical Simulation of Effect the Geometrical Underlip on The Performance of Oscillating Water Column

Azhar Bima Alfarozi[1*] **, Muhammad Agung Bramantya**[1]

[1]Department of Mechanical and Industrial Engineering, Gadjah Mada University, Indonesia
**Email address of corresponding author: azharbima2019@mail.ugm.ac.id*

ABSTRACT

The development of water wave energy conversion especially Oscillating Water Column (OWC) continues to find the most optimal system. Using the CFD method, the research process becomes faster and more effective because one of the advantages of CFDs is the rapid prediction of phenomena with a numerical simulation approach. This research aims to increase efficiency of water wave energy conversion, especially OWC using CFD. Modifications were made to the OWC underlip with the model used in the experiment by Celik (2022). The models are rectangular model (as reference), rounded model, quarter circle model, front semi-circle model, back semi-circle model, and circle model. From the CFD simulation results, an increase in efficiency is obtained for each modification model. The optimal model is the circle model with an efficiency increase of 8.87% from the conventional underlip (rectangular) model. This is because the turbulent kinetic energy in the system can be reduced.

Keywords: Oscilating Water Column, Underlip Geometry, Turbulence Kinetic Energy, Computational Fluid Dynamic

1. Introduction

Ocean wave energy is one of the energies with great potential in the world. However, the utilization of ocean wave energy has not been carried out massively because the conversion device needs to be modified more deeply. Some of the conversion devices currently developed by several researchers are Oscillating Water Column system (OWC). Oscillating water column is one of the water wave energy conversion devices currently being developed massively. This is because OWC has a simple system and potential efficiency that can continue to be developed Çelik [6].

Research on Oscillating Water Column is conducted comprehensively by modifying the conventional model. The conventional model is a simple model of OWC with a simple structure. The simple structure consists of a long beam column with wells turbine above the column. Modification of the conventional model is the focus of researchers in the development of OWC. Modifications made in the form of column wall geometry models, variations in column shape and arrangement, modifications to the conversion system, and other modifications to increase OWC efficiency. Research conducted by Qu et al. [1] modified the front wall of the OWC with an elliptical shape with a certain angle configuration to obtain a difference in efficiency with a conventional shape reference. The front wall varies the angle of the ellipse to get the maximum value of the system. With this modification, a model is obtained that can increase efficiency by 25% from the conventional model. OWC research with column wall modifications was also conducted by Fleming and Macfarlane [2]. The study presents an analysis of 2D PIV model test experiments conducted on a series of forward-facing bent-channel type OWC models with variations in underwater geometry. This complex experiment involved particle imaging velocity (PIV) to compare the performance differences between geometries. The results of the study show that even small changes in geometry shape affect the characteristics of the OWC system in converting energy.

Another OWC shape research was also conducted by Carlo et al. [3]. The study investigated the U-type OWC shape. In this case, the column is formed into a U shape to obtain a higher efficiency. Experimental results show that in some models with a certain damping ratio can increase efficiency. Furthermore, research conducted by Wang and Zhang [4] related to the front-rear dual system (multi chamber OWC) also provides insight into the OWC model. The results show that there is marginal non-uniformity in the vertical airflow velocity through the orifice, making this method based more on-air pressure and airflow velocity reasonable for estimating the capture width ratio. In general, the front-back multi chamber can widen the capture width so the wave energy conversion is greater. In addition to the shape of the OWC geometry, other research on OWC modification focuses on bottom modification to manipulate the wave flow through the bottom passing column Mohapatra et al. [5]. The hydrodynamic performance of the device was investigated with the modification of convex, concave, and inclined ladder-type bottom profile variations. From the study, each bottom profile has its own characteristics that are very different from the conventional case model.

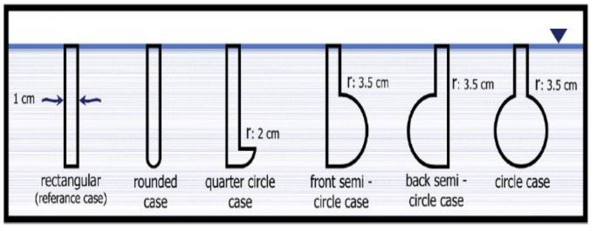
Some experimental results of research on other OWC modifications succeeded in increasing the efficiency only by making small geometry changes. For example, Çelik [6] was able to increase the efficiency only by changing the underlip geometry. Rounded, quarter circle, front semi-circle, back semi-circle, and full circle underlip geometries were studied for their efficiency performance output with a certain damping ratio. The results show that the full circle underlip on the OWC provides a maximum increase of 21.2%. This occurs due to a reduction in vortex shedding and an increase in the amount of wave energy incident into the chamber. Furthermore, a full circle model that can increase efficiency even though it only changes the geometry slightly was studied by Mandev and Altunkaynak [7]. The results showed that the cylindrical front wall entrance geometry prevents or at least reduces the flow separation that occurs due to the sharp bottom of the front wall. Consequently, the reduction in diameter size helps to relieve the shear stress on both sides of the front wall, thereby improving the structural integrity of the OWC.

From these experimental studies, this research focuses on investigating the causes of OWC efficiency increase using CFD method. The reference experimental data is that of Çelik [6]. The Çelik [6] model is used as the basis of the research because changing the underlip geometry can significantly increase efficiency. This is an interesting topic of research, because only small changes in geometry can change the performance characteristics of OWC.

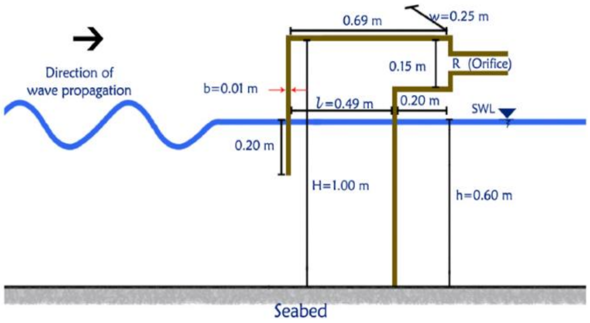
2. Material and Methods

2.1 Description of The Solvers

The research was conducted using a 2-dimensional numerical method. In this case, the simulation used Ansys Fluent software. Modeling begins with creating a 2-dimensional domain of the OWC. The model refers to research conducted by Celik [6] that is shown in Figure 1 and the dimension shown in Figure 2. After the domain is determined, the mesh process is carried out comprehensively with the mesh independence stage. To simplify the case, the model variations are named models 1 to 6 with different underlip geometries. These variations are shown in table 1.



Source: (Çelik, 2022)
Figure 1: Variations Model of Underlip



Source: (Çelik, 2022)
Figure 2: Dimension of OWC

Table 1: Model Underlip

Model	Underlip Geometrical
Model 1	Rectangular
Model 2	Rounded-Case
Model 3	Quarter-Circle Case
Model 4	Front-Semi Circle Case
Model 5	Back-Semi Circle Case
Model 6	Circle Case

The mesh is processed with multizone quad. In this case, the blocking method is used on the mesh so that it is expected that important zones that need accuracy can be resolved accurately. From the mesh independence data, the important part is sized with a maximum sizing zone of 4 mm. Meanwhile, other zones are set with a maximum sizing zone of 25 mm. In the mesh zone in the generation wave section, the sizing size used is 70 mm with hard quad multizone. The setting is done in the generation wave section to avoid errors due to the dynamic mesh applied during wave initiation. Example mesh results are shown in figure 3.

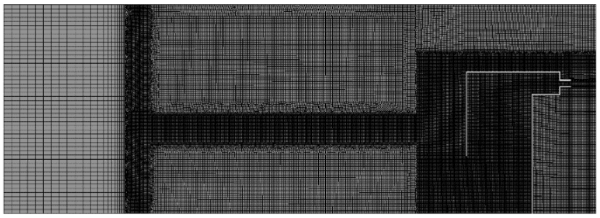


Figure 3: The Mesh Model

OWC modeling uses a pressure-based model with transient. The models used are the 2-phase mixture model and the k-ω shear-stress transport (SST) turbulence model. The setting type used for the 2-phase method is mixture sharp. The selection of the 2-phase method is used according to the case. In this case, water and air interact directly with each other in the column so that in this case it is more appropriate to use mixture sharp, so that both can be seen behavior without losing the contours of each element of different phases. In addition, the surface tension model is used, because to create the behavior of wind and waves, both rub against each other so that in the real-world sea waves can occur. The solution method used is schema simple with spatial discretization first order upwind. This scheme is used to speed up the calculation process with a light calculation load but is quite accurate.

2.2 Wave Generation Method

Wave modeling was initiated with a dynamic mesh. The front part of the mesh is made slightly loose with hard quad multizone to prevent errors. The type of dynamic mesh used is layering. The initiated wave is a transverse wave with a simple wave equation. The wave equation is expressed mathematically as follows.

$$v = \omega . a . \cos(\omega . t) \qquad 2.1$$

Herein, ω is angular velocity, v is velocity and, a and t are the amplitude and time. To create waves in Fluent software, layering dynamic mesh motion is used by using compiled files in the form of user defined functions (UDF) with coding. The calculation process is done in parallel to avoid errors because UDF coding cannot be built into the simulation system. In addition, parallel calculation also makes it easier to manage simulation result data.

2.3 *Data Collective and Validation*

The simulation model consists of 6 geometries with an Orifice ratio of 0.0079 according to the reference experiment conducted by Celik [6]. The model has different underlip geometries. The underlip geometries are rectangular (conventional), rounded case, quarter circle case, front semi-circle case, back semi-circle case, and circle case. Each geometry was tested with the same wave parameters so that they could be compared. The wave characteristics used have a wave height of 0.07 m, with a period of 1 s (for regular transversal wave), and a wave steepness of 0.02.

The simulation data taken is the output velocity and output pressure data at the column outlet. A point is placed on the center axis of the column outlet which can be likened to a sensor. In this case, the data taken is the wind speed and pressure data captured at that point. After obtaining the outlet speed and pressure data, the data is then processed to become the output power generated by the system. In this case, the Bernoulli Equation shown below is used (equation 2.2). Meanwhile, the efficiency is calculated with the equation 2.3. With the wave power comes the calculation with the formula 2.4 formulated by McChormick (2010)[8].

$$P_u = (P_2 - P_0) . v_2 . A_2 \qquad 2.2$$

$$P_W = 0{,}195\, w\, \rho\, g\, h^2\, T \qquad 2.3$$

$$\eta = P_u / P_w \qquad 2.4$$

Where P_u is pressure of air, P_2 is pressure at outlet, P_0 is atmospheric pressure and, $v_2$and A_2 is velocity outlet and the orifice cross-sectional area. P_W is power wave. w is column front wall width. ρ is density of air. h is height of wave. g is gravity and T is periodic of wave. After obtaining the output power data, the efficiency of the system can be calculated. So that the system output review can be compared. Then the data is validated with experimental data from Celik (2023). If the percentage error value is below 5%, then other data is considered valid so that it can be used to determine the characteristics of the system that occurs. In this case, animation is used for the analysis process. The animations used are contour and path line profiles. With contours and path lines, it is expected that the characteristics of the system can be found and explained in detail.

3. Result and Discussion

3.1 Output and Efficiency

Velocity and pressure data at the outlet point are taken transiently over time. The output velocity and pressure of the system are shown on graphs and charts. The data is then used to calculate the output power so that it can later be used to calculate the efficiency of the system.

The efficiency of each system is compared to evaluate the effect of underlip. The efficiency compared is the highest efficiency in each cycle per underlip modification geometry model. This is based on the speed and pressure combination obtained. The speed and pressure that produce maximum power are shown in the diagrams and charts. In this case the maximum efficiency of each system based on the combination of velocity data and pressure data can be seen in Figure 4 and Figure 5. From the figure the changes in velocity and pressure over time form valleys and peaks. This indicates that the process of water goes ups and downs in the column.

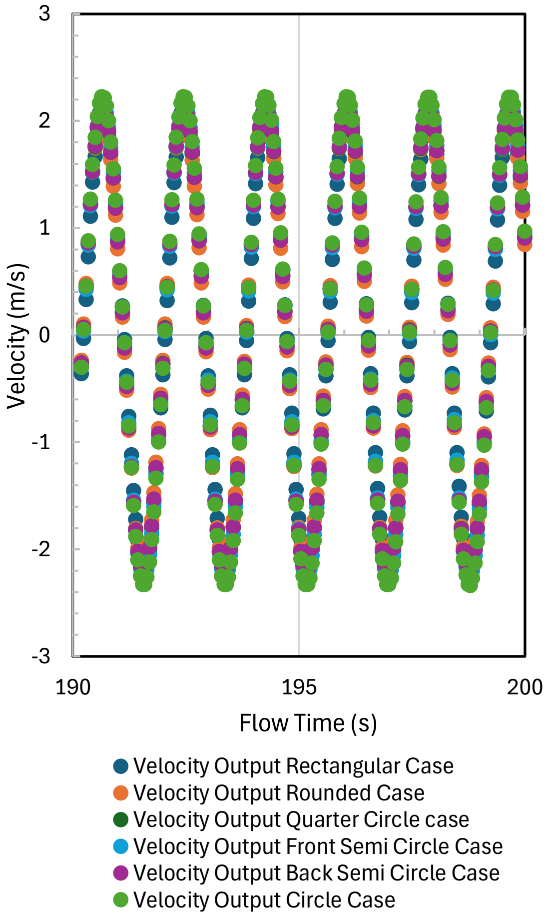


Figure 4: Velocity Air Output

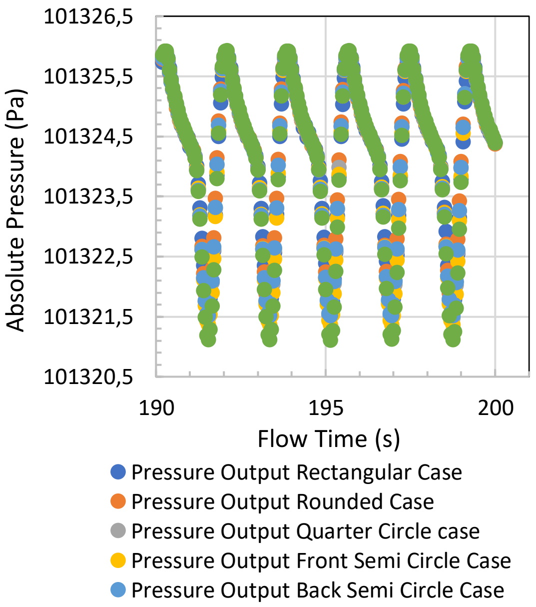


Figure 5: Pressure Air Output

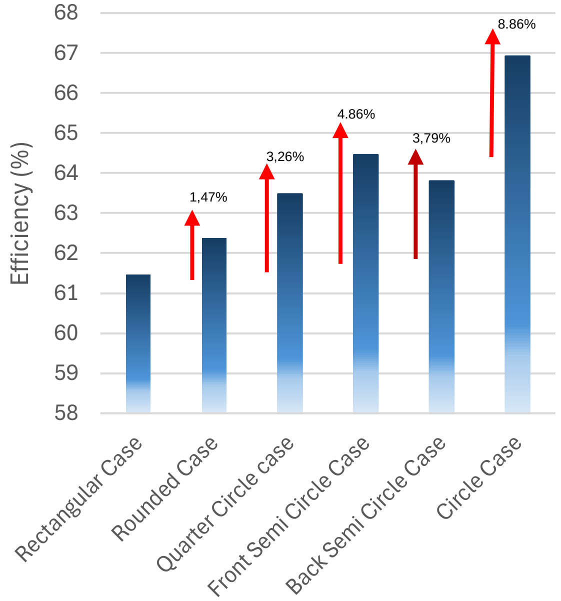


Figure 6: Efficiency

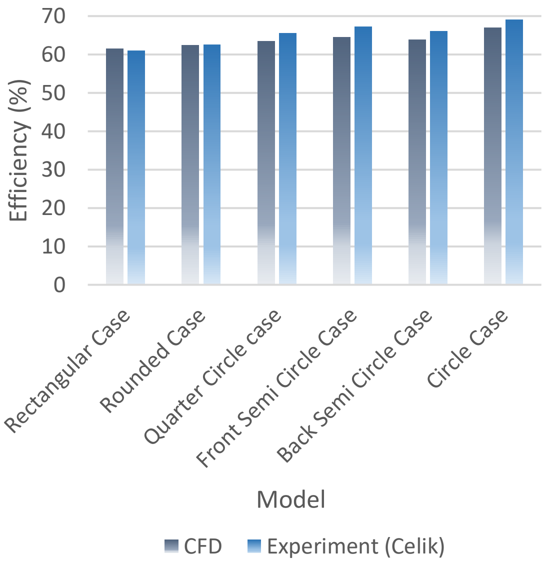


Figure 7: Validation

Meanwhile, the validation is compared with Celik's experiment (2023). The comparison diagram is shown in the diagram. From this data, the maximum error in each model is still below 5%. The maximum error is obtained in the front semi-circle case model with an error of 4.08%. From this comparison, the data is considered valid because it is still in the range below 5%. So that the data obtained can be used to investigate the cause of the increase in efficiency.

3.2 Effect of modification underlip

The efficiency data shows that the underlip modification of the OWC with the research geometry model can increase the efficiency. To know and investigate the cause of the increase in system efficiency, animation solutions are needed in the form of contours and path lines that occur in the system. The contours and path lines used for investigation are phase contours, velocity path lines, and turbulent kinetic energy path lines. The contours and path lines are used to investigate flow patterns, effects, and determine the cause of the increase in efficiency. The contours and path lines are shown in the figure.

In the rectangular underlip model, the CFD path line results inform us that turbulent clumps of kinetic energy are larger than the other models. The turbulent flow clumps occur when the waves come, and the waves go away (the system fluid flow goes up and the system fluid flow goes down). This causes losses to the system because the turbulent flow causes the friction at the column wall to be more significant. With high friction, the energy of the waves cannot be channeled to the outlet end optimally. With the energy that is not channeled, the efficiency of the system decreases. In addition, in the rectangular model, clumpy flow can cause the flow pattern in the system to be disrupted. This will also result in poor resonance between ocean waves and fluid movement in the system. Although the resonance image of the system does not appear to cause turbulent kinetic energy in the contact area of the water and air fluid.

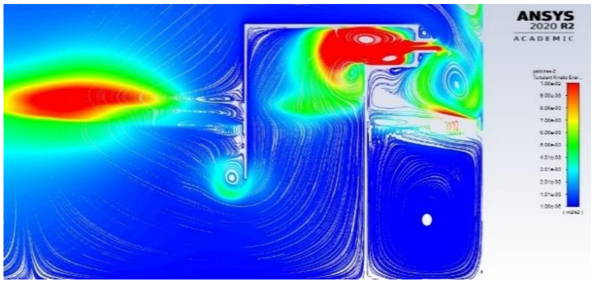


Figure 8: Turbulence Kinetic Energy Rectangular Underlip Wave Back

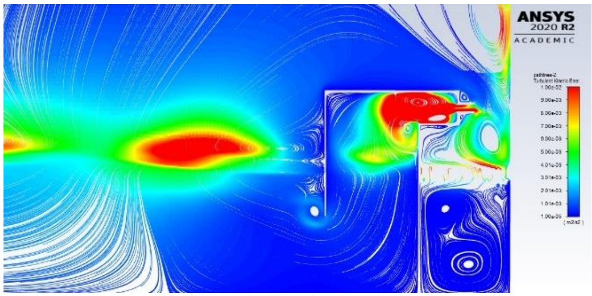


Figure 9: Turbulence Kinetic Energy Rounded Case Underlip Wave Back

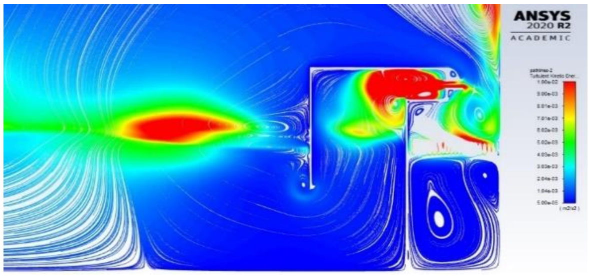


Figure 10: Turbulence Kinetic Energy Quarter-Circle Underlip Wave Back

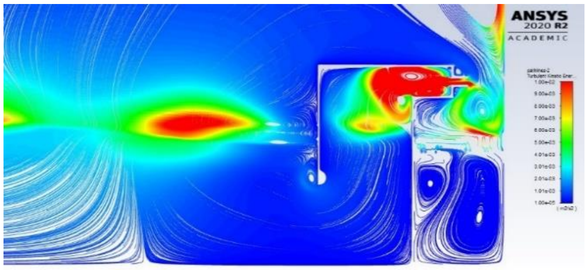


Figure 11: Turbulence Kinetic Energy Front-Circle Underlip Wave Back

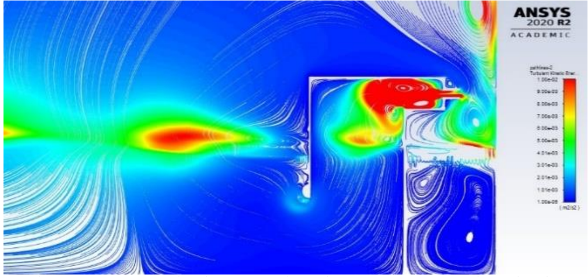


Figure 12: Turbulence Kinetic Energy Back-Circle Underlip Wave Back

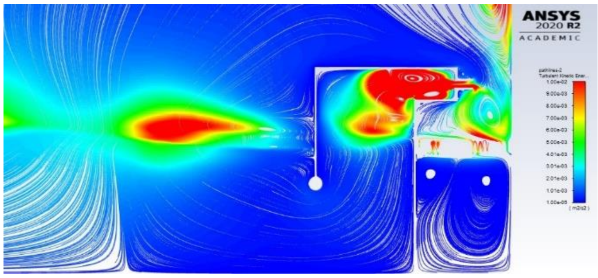


Figure 13: Turbulence Kinetic Energy Circle Underlip Wave Back

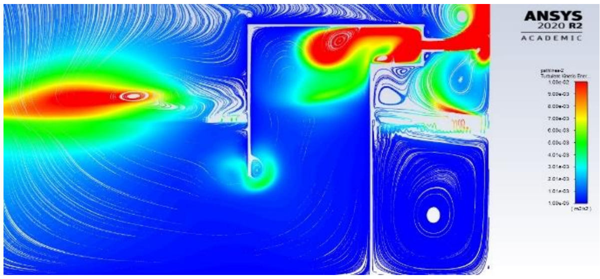


Figure 14: Turbulence Kinetic Energy Rectangular Underlip Wave Come

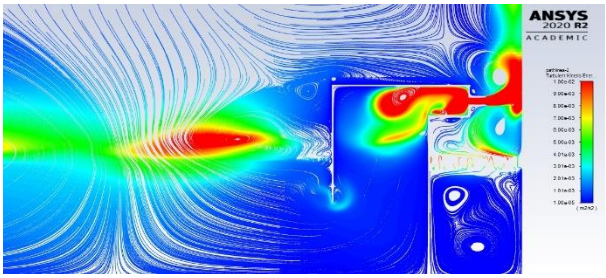


Figure 15: Turbulence Kinetic Energy Rounded Case Underlip Wave Come

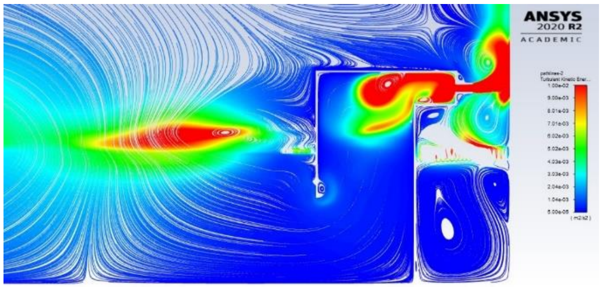


Figure 16: Turbulence Kinetic Energy Quarter-Circle Underlip Wave Come

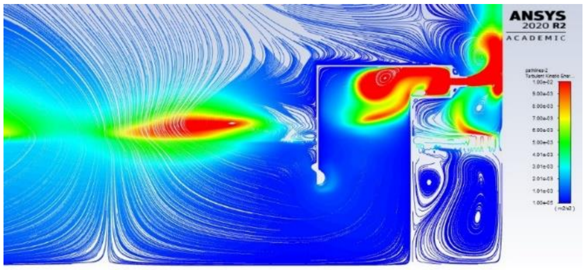


Figure 17: Turbulence Kinetic Energy Front-Semi Circle Underlip Wave Come

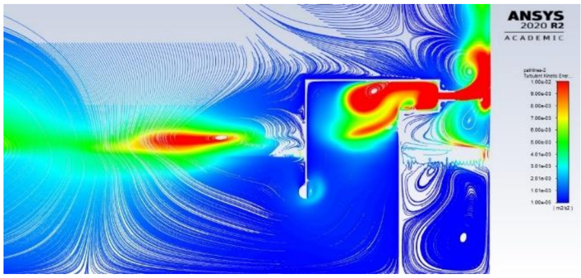


Figure 18: Turbulence Kinetic Energy Back-Semi Circle Underlip Wave Come

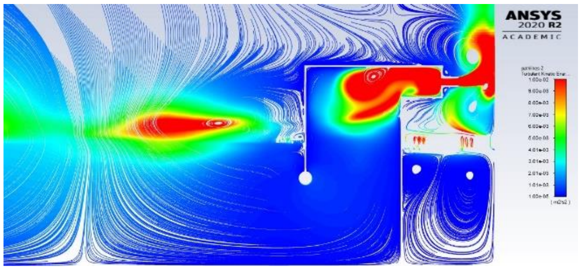


Figure 19: Turbulence Kinetic Energy Circle Underlip Wave Come

Furthermore, in the rounded underlip modification, the CFD path line results inform that there is a turbulent plume of kinetic energy that is larger than the other models but still lower than the rectangular model. The same phenomenon occurs in this model. The difference is that the turbulent kinetic energy that occurs in the system, especially in the underlip area, is not as large as that in the rectangular underlip model. This has a positive impact because the energy channeled becomes slightly larger. Thus, causing the efficiency of the system to increase by 1.47%. However, this happens only when the incoming wave enters the column, when the wave exits the column there is a large clump on the outer side of the column. However, this phenomenon does not interfere with the flow inside the column. This may have more effect on the resonance of the flow because the next incoming wave is slightly disturbed.

In the case of the quarter circle model, the CFD path line results indicate that the flow plume resulting in turbulent kinetic energy shifts to the rear of the quarter circle geometry when waves arrive. This also has a positive impact when compared to the rectangular model. With this modification, there is a reduction in turbulence which can be seen from the color contours of the path line. In this case, the efficiency also increased by 3.26% from the rectangular system. The efficiency is also greater when compared to the system efficiency in the rounded underlip modification. This is because the flow clumps that cause turbulence at the same time are smaller. So that the value of energy that is not channeled is smaller and the efficiency becomes slightly greater.

In the modification of the semi-circular underlip front model, the CFD path line results when the water wave arrives inform that the turbulent flow vortex can be prevented. This phenomenon can significantly reduce the turbulent flow pattern. In this case, due to the reduced turbulence and improved flow pattern, the efficiency increases up to 4.86% of the efficiency of the rectangular underlip model. However, when the wave returns, a vortex flow pattern occurs at the front side of the column which can trigger turbulent kinetic energy. In addition, the poor flow pattern at the front of the column will cause the incoming water waves to not resonate well with the up and down motion of the water in the OWC column. This is the opposite of what happens in the back semi-circle model. In the back semi-circle underlip modification, when the wave arrives, the front side of the column has a good flow pattern. However, in the underlip area inside the column, turbulent kinetic energy is quite significant. This makes the efficiency lower than the front semi-circle underlip type. The efficiency of this model increased by 3.79% from the rectangular model. However, when the ocean waves move away from the column system, the flow pattern becomes slightly better although there is still considerable turbulent kinetic energy.

Meanwhile, the best flow pattern occurs in the underlip circle model. In this model, the flow pattern does not occur turbulence vortex. In addition, the turbulent kinetic that occurs can be prevented significantly. With this positive phenomenon, the underlip circle model has the greatest efficiency value, which has increased by 8.87% from the rectangular model.

4. Conclusion

From the discussion and description of the analysis, it can be concluded that by modifying the conventional underlip model into a rounded underlip case, quarter circle case, front semi-circle case, back semi-circle case, and circle case can increase the efficiency of OWC. In this case, the largest increase in efficiency occurs in the circle underlip model with an increase in efficiency of 8.87% from the rectangular model. This is because losses due to turbulent kinetic energy can be minimized. From the description of the analysis, it can also be concluded that turbulent kinetic energy that occurs in the underlip area affects the efficiency of the OWC system. The greater the turbulent kinetic energy, the greater the conversion loss. So that the greater the turbulent kinetic energy the efficiency of the system decreases.

References

[1] Carlo, L., Iuppa, C., & Faraci, C. (2023). *A numerical-experimental study on the hydrodynamic performance of a U-OWC wave energy converter.* Renewable Energy, 203, 89–101.

[2] Çelik, A. (2022). *An experimental investigation into the effects of front wall geometry on OWC performance for various levels of applied power take off dampings.* Ocean Engineering, 248.

[3] Fleming, A. N., & Macfarlane, G. J. (2017). *Experimental flow field comparison for a series of scale model oscillating water column wave energy converters.* Marine Structures, 52, 108–125.

[4] Mandev, M. B., & Altunkaynak, A. (2023). *Cylindrical frontwall entrance geometry optimization of an oscillating water column for utmost hydrodynamic performance. Energy, 280.*

[5] Mohapatra, P., Vijay, K. G., Bhattacharyya, A., & Sahoo, T. (2023). *Influence of distinct bottom geometries on the hydrodynamic performance of an OWC device.* Energy, 277.

[6] Qu, M., Yu, D., Xu, Z., & Gao, Z. (2022). *The effect of the elliptical front wall on energy conversion performance of the offshore OWC chamber: A numerical study.* Energy, 255.

[7] Wang, C., & Zhang, Y. (2021). *Numerical investigation on the wave power extraction for a 3D dual-chamber oscillating water column system composed of two closely connected circular sub-units.* Applied Energy, 295.

[8] McChormick (1981). *Ocean Wave Energy Conversion*. Annapolis: Maryland: ept of Naval Systems Engineering, U.S Naval Academy