TiMER
Trends in Mechanical Engineering Research

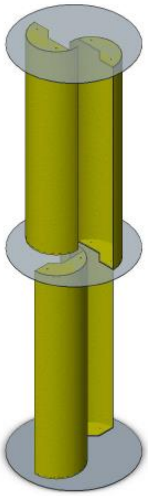
REVIEW OF COMPUTATIONAL FLUID DYNAMIC (CFD) OF SAVONIUS ROTOR FOR TURBINE RENEWABLE ENERGY

Alwan Habibie[a], Hadi Wahyudi[a], Dedy Triawan Suprayogi[a]*

[a] Department of Mechanical Engineering, Faculty of Engineering, Universitas Sultan Ageng Tirtayasa, Cilegon, Banten, Indonesia

*Corresponding author: dedy.ts@untirta.ac.id

Graphical abstract



Article history
Received
11 December 2023
Received in revised form
13 December 2023
Accepted
20 December 2023
Published
30 December 2023

Abstract

This research aims to establish the accuracy foundation of Computational Fluid Dynamics (CFD) in simulating the utilization of Savonius Rotor in marine currents, particularly in the vast potential waters of Indonesia. The methodology involved a comprehensive review of journal articles, evaluating studies on the application of CFD in simulating the performance of Savonius turbines in marine current environments. The journal review findings indicate the effective use of CFD in replicating Savonius Rotor behavior, particularly in Indonesia's marine current context. The evaluation of CFD simulation accuracy in this study reveals reliable and accurate results, aligning well with experimental data. These findings suggest that CFD is a dependable tool for effectively simulating Savonius Rotor performance in Indonesian marine current conditions, providing a strong foundation for further research on harnessing marine current potential as a renewable energy source in Indonesia.

Keywords: Computational Fluid Dynamics (CFD),Curent Sea, Savonius Rotor

Abstrak

Penelitian ini bertujuan untuk membentuk dasar akurasi Computational Fluid Dynamics (CFD) dalam mensimulasikan penggunaan Savonius Rotor pada arus laut, khususnya di perairan potensial luas di Indonesia. Metode yang digunakan melibatkan tinjauan komprehensif terhadap artikel jurnal, mengevaluasi studi tentang penerapan CFD dalam mensimulasikan kinerja turbin Savonius di lingkungan arus laut. Temuan dari tinjauan jurnal menunjukkan bahwa CFD telah digunakan secara efektif dalam mereplikasi perilaku Savonius Rotor, terutama dalam konteks arus laut di Indonesia. Evaluasi akurasi simulasi CFD dalam penelitian ini mengungkapkan hasil yang andal dan akurat, sejalan dengan data eksperimental. Temuan ini mengindikasikan bahwa CFD merupakan alat yang dapat diandalkan untuk mensimulasikan kinerja Savonius Rotor secara efektif dalam kondisi arus laut di Indonesia, memberikan dasar yang kuat untuk penelitian lebih lanjut mengenai pemanfaatan potensi arus laut sebagai sumber energi terbarukan di Indonesia.

Kata kunci: Arus Laut, Computational Fluid Dynamics (CFD), Rotor Savonius

DOI: http://dx.doi.org/10.62870/timer.v1i2

1.0 INTRODUCTION

Indonesia, known as the country of a thousand islands, is also a maritime nation, which means that Indonesia has a territory in the form of seas and waters that are wider than land. In this case, Indonesia has a wide potential to generate renewable energy in marine areas [1]. These marine potentials include ocean currents, tidal effects, sea heat, salinity differences, and waves. Tidal energy is the exploitation of energy that arises when water masses move due to changes in tides that are influenced by the rotation of the Earth and the gravitational pull of the moon [1] And ocean currents are the movement of ocean water masses from one location to another, both vertically and horizontally [2]

In Indonesia, there are many straits that have the potential for current speeds of around 2.0 m/s or even more. This speed is sufficient to be utilized as electricity [2]. Among the straits that have a sufficient potential for ocean currents is the Sunda Strait. According to a study by [3], the Sunda Strait has an average ocean current potential of 0.53 m/s and a maximum current speed of 0.87 m/s in the middle of the strait.

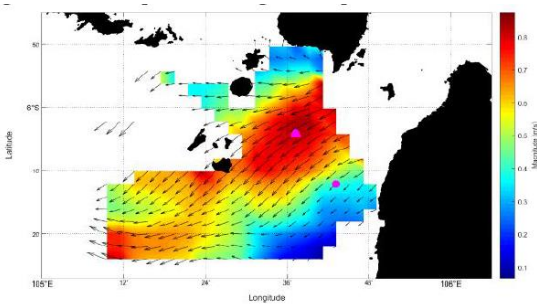


Figure 1. The Potential of Ocean Currents in the Sunda Strait [2].

Together with the target set by the Government of the Republic of Indonesia through the Ministry of Economic Affairs, there is a serious focus on the Renewable Energy (RE) industry, one of which involves reducing dependence on fossil energy. The target is to achieve 23% by 2025 and progressively increase it to 31% by 2050, compared to the current level of around 15% [4]. With Indonesia's potential for ocean currents, particularly in the Sunda Strait, which can be effectively utilized, one method is through the use of ocean current turbines, such as the Savonius Turbine. This turbine is advantageous in low-speed currents, as indicated in the study by [5], where a current of 0.269 m/s produced a torque exceeding 30 Nm.

Computational Fluid Dynamics (CFD) is a comprehensive system analysis method that considers phenomena related to fluid flow, heat transfer, and other associated phenomena, including chemical reactions, through computer-based simulations utilizing a numerical approach [6]. One notable tool for CFD simulations is Fluent, which is part of the ANSYS suite of software. Fluent is widely used for its robust capabilities in modeling and analyzing fluid flow, heat transfer, and various related phenomena in diverse engineering applications.

Computational Fluid Dynamics (CFD) is a computer tool that has been widely utilized to investigate and simulate various topics, including the Savonius Rotor. Some of the studies employing this tool include the following journal [7], which conducted wind turbine Savonius Rotor simulations to understand the performance improvement by transforming the conventional Savonius shape into a Bezier Curve. Therefore, this research will delve deeper into studies utilizing CFD tools, particularly in the realm of Savonius Rotors.

Renewable Energy

Renewable energy, as explained in Law Number 30 of 2007 on energy, refers to energy sources that are renewable, including geothermal, wind, bioenergy, solar radiation, water flow and falls, as well as the movement and temperature differences of the ocean layers. This renewable energy harnesses environmentally friendly resources, does not cause pollution, and does not contribute to climate change or global warming. The concept arises because the energy produced comes from natural processes that occur sustainably, such as solar heat, wind, water, biofuels, and geothermal heat [8].

The significance of renewable energy is not only related to the availability of resources but also to its positive impact on the environment and ecology. Energy generated from renewable resources does not harm the environment, which is a primary reason why Renewable Energy is closely linked to environmental and ecological issues.

Table 1. Energy Resources [8].

Types of Renewable Energy	Potential (MW)
Wind	950
Solar Power	11.000
Hydro Power	75.000
Biomassa	32
Biofuel	32
Ocean	60.000
Geothermal	29.000

The development and potential outlined in the table above regarding renewable energy in Indonesia represent a strategic step toward achieving sustainability in the utilization of energy resources. Indonesia, as a country with significant potential in the development of renewable energy, has the opportunity to utilize these resources optimally.

Consequently, the country can not only meet its energy needs but also reduce negative impacts on the environment.

Savonius Rotor

The Savonius Rotor is a common type of rotor often used in wind power known for its operation based on the principle of drag force, which generates mechanical energy from the wind thrust on the buckets or blades that rotate the rotor. The basic construction of the Savonius Rotor involves two to three blades arranged in a specific pattern, giving it an appearance like the letter "S" when viewed from above. As the wind passes through these blades, the pressure generated by the wind flow pushes the blades, causing the rotor to rotate.

A crucial parameter for the aerodynamic performance of the Savonius rotor is the aspect ratio representing the rotor height (H) relative to the diameter (D). A high aspect ratio value (a) significantly enhances efficiency [9], as illustrated in the image below:

$$\alpha = \frac{H}{D}$$

Where:
α = Aspect Ratio
H = Rotor Height
D = Rotor Diameter

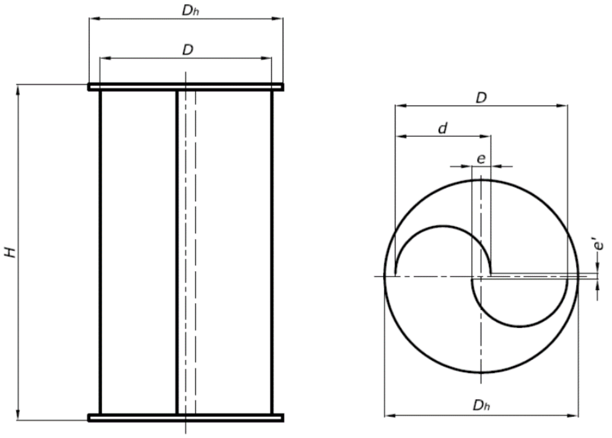


Figure 2. Dimension Of Savonius Rotor [9].

Another parameter is the overlap ratio (β), which affects the overlap (e) and paddle diameter (d). The overlap ratio (β), considered as a variation in the study, is given by the following equation [9].

$$\beta = \frac{e}{d}$$

Where:
β : Overlap Ratio
e : Overlap Between Paddles
d : Paddles Diameter

This overlap ratio will influence the results or efficiency of the tested turbine. In a study on Savonius Rotor in marine currents conducted by [10]), the optimal value for the overlap ratio was found to be 0.21, resulting in a torque of 0.387 Nm. In contrast, a study conducted by [11], found the best overlap ratio to be 0.15, with an average power coefficient of 0.3161 for a tip speed ratio of 1.25 in a wind Savonius turbine.

Due to its distinctive characteristics, the Savonius Rotor has a wide range of applications in the field of renewable energy, such as wave breakers or in small-scale power generation. It provides an interesting alternative in the effort to harness renewable energy resources [12].

Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems involving fluid flow. CFD provides a powerful tool for simulating and understanding the behavior of fluids in various engineering applications. The process of CFD involves several key steps [13]:

1. Mathematical Formulation. The first step in CFD involves formulating the mathematical equations that describe the behavior of fluid flow. These equations are typically partial differential equations that govern the conservation of mass, momentum, and energy within the fluid domain.

2. Discretization. The continuous mathematical equations are then iscretized to create a numerical representation of the fluid flow problem. This involves dividing the fluid domain into small grid cells or elements, allowing the equations to be solved at discrete points within the domain.

3. Boundary Conditions. Boundary conditions, which define the behavior of the fluid at the boundaries of the domain, are specified as part of the problem setup. These conditions are essential for accurately simulating the interaction between the fluid and its surroundings.

4. Solution Methods. Various solution methods, such as finite difference, finite element, or finite volume methods, are employed to solve the discretized equations and obtain numerical solutions for the fluid flow variables (e.g., velocity, pressure) within the domain.

5. Pre-processing. The pre-processing stage involves setting up the problem geometry, generating the grid, and defining the flow parameters and boundary conditions. This step prepares the problem for solution within the CFD software.

6. Flow Solver. The flow solver is the core component of the CFD process, responsible for solving the discretized equations of fluid flow subject to the specified conditions. It utilizes numerical algorithms to compute the behavior of the fluid within the domain.

7. Post-processing. Once the numerical solution is obtained, post-processing involves analyzing and visualizing the results in a format that is easy to interpret. This step may include generating graphical representations of the flow field, extracting relevant data, and assessing the accuracy of the simulation results.

CFD has diverse applications across various engineering disciplines, including aerospace, automotive, environmental, and civil engineering. It is used to predict fluid flow patterns, heat transfer, mass transfer, and other related phenomena, making it an indispensable tool for design, analysis, and optimization of engineering systems involving fluid dynamics.

2.0 METHODOLOGY

The researcher conducted a comprehensive literature review by gathering information from reputable journals and websites accessible to a wide audience. This method allowed for an in-depth exploration of topics related to renewable energy, Savonius Rotor, and Computational Fluid Dynamics (CFD). The analysis aimed to extract meaningful insights and conclusions regarding the utilization of CFD in optimizing Savonius Rotor performance.

3.0 RESULTS AND DISCUSSION

Several studies related to CFD simulations, especially on Savonius Rotor, include research [14]. The study discussed in this research involves an analysis using Computational Fluid Dynamics (CFD) to understand the behavior of Savonius wind turbines. CFD is a numerical method used to model and analyze fluid flow, such as air through wind turbines. In this context, CFD is employed to predict how airflow interacts with the shape and motion of the Savonius turbine.

In this study, CFD simulations were conducted using the open-source software OpenFOAM. The simulations encompassed both 2D and 3D models to comprehend turbine performance under various conditions. The 2D model was used to reduce computation time, while the 3D model was utilized for result validation against experimental data.

In CFD simulations, the airflow domain around the turbine is modeled into small cells forming a grid. Subsequently, the fundamental equations describing fluid flow are numerically solved to obtain the airflow solution around the turbine. From this solution, various parameters such as aerodynamic forces, torque, and power coefficients can be calculated.

The results of CFD simulations are utilized to compare turbine performance under different conditions, such as variations in rotational speed and blade tip speed ratio. Thus, this study provides an in-depth understanding of how design and operational parameters influence the performance of Savonius wind turbines.

By employing CFD, researchers can visualize airflow around the turbine, predict aerodynamic forces acting on the turbine, and optimize turbine design to enhance energy efficiency. This serves as an example of how CFD is used to comprehend and improve the performance of renewable energy technologies, such as wind turbines.

In that study, the CFD error values were obtained by comparing the CFD simulation results with available experimental data. This comparison provided information about how well the CFD model replicates the turbine's performance observed in the experimental testing. The results indicated that the CFD model could replicate the performance characteristics of the turbine observed in the experimental testing, with error values lower than the experimental uncertainty. This suggests that the CFD simulation results have a high level of agreement with experimental data, making them reliable for understanding the performance of Savonius wind turbines.

Therefore, the findings from the study demonstrate that the CFD model used has provided valid and accurate results in replicating the performance of Savonius wind turbines. This instills confidence that CFD simulation results can be used to comprehend airflow around the turbine, predict turbine performance under various operational conditions, and optimize turbine design to enhance energy efficiency.

Similar to the research conducted by [15], they employed Computational Fluid Dynamics (CFD) methodology to study the behavior of vertical-axis Savonius wind turbines. They utilized a 3D model to simulate the airflow through Savonius wind turbines. Due to the operating principles and continuous variation of the flow angle towards the blades, strong unstable effects, including separation and vortex formation, were observed. Therefore, turbulence modeling using k-Z and DES allowed for obtaining favorable results compared to experiments.

They also explained that they utilized Particle Image Velocimetry (PIV) for experimental investigations in a wind tunnel. This investigation enabled them to determine the real flow structure and assess the quality of numerical simulations. The experimental results were then used to validate the simulation outcomes obtained through CFD.

Furthermore, they elaborated on the 3D simulation domain they employed. They applied "velocity inlet" boundary conditions at the upstream rotor and "pressure outlet" at the downstream rotor. They also used 15 structured blocks with 50.10^3 cells and 2 unstructured blocks with 30.10^3 cells for the internal grid section. Additionally, they employed Detached

Eddy Simulation (DES) technique to enhance airflow simulation.

The CFD simulation results produced by Dobrev and Massouh indicated that the 3D model with DES yielded results closer to experimental data compared to the 2D model or the 3D model with k-Z. This suggests that the use of DES in CFD simulations can provide more accurate results in representing separated and dynamic airflow.

4.0 CONCLUSION

From several studies conducted by us, it can be concluded that CFD simulations are accurate and well-suited for use as references. The results obtained through CFD simulations in our research demonstrate reliability in replicating the performance of Savonius wind turbines, providing valuable insights into airflow dynamics and optimizing turbine design for enhanced energy efficiency. The consistent agreement between CFD results and experimental data indicates the effectiveness of CFD as a reliable tool in understanding and predicting the behavior of Savonius turbines under various operational conditions..

.

Acknowledgement

This research is independent work. The authors fully acknowledged to Mechanical Engineering Dept, Universitas Sultan Ageng Tirtayasa.

References

[1] M. Buwana, N. Royyana, U. Budiarto, and R. Good, "ANALISA BENTUK OSCILLATING WATER COLUMN UNTUK PEMANFAATAN GELOMBANG LAUT SEBAGAI SUMBER ENERGI TERBARUKAN DENGAN METODE COMPUTATIONAL FLUID DYNAMIC (CFD)," 2015.

[2] A. Kasharjanto, D. Rahuna, and D. Rina, "KAJIAN PEMANFAATAN ENERGI ARUS LAUT DI INDONESIA Study on the Implementation Of Marine Current Energy In Indonesia," 2017.

[3] C. M. Serodja, A. Ismanto, A. R. Hakim, and D. A. Ramdhani, "Pengaruh Angin Monsoon Timur Terhadap Arus Permukaan Berdasarkan Data HF Radar di Perairan Selat Sunda," 2022. [Online]. Available: https://ejournal2.undip.ac.id/index.php/ijoce

[4] Menko Perekonomian, "Miliki Perhatian Serius pada Energi Baru Terbarukan, Pemerintah Berkomitmen Menjaga Ketersediaan Energi Nasional Berkelanjutan," Kementrian Kordinator Bidang Perekonomian Republik Indonesia. Accessed: Nov. 16, 2023. [Online]. Available: https://www.ekon.go.id/publikasi/detail/5250/miliki-perhatian-serius-pada-energi-baru-terbarukan-pemerintah-berkomitmen-menjaga-ketersediaan-energi-nasional-berkelanjutan

[5] E. Pranatal and M. M. Z. Beu, "Analisa CFD Penggunaan Duct pada Turbin Kombinasi Darrieus-Savonius," *J. IPTEK*, vol. 22, no. 1, p. 63, May 2018, doi: 10.31284/j.iptek.2018.v22i1.239.

[6] S. Z. N. I. Akmal, "Analisa Profil Aliran Fluida Cair dan Pressure Drop pada Pipa L Menggunakan Metode Simulasi Computational Fluid Dynamic (CFD)," *Teknol. Kim. Unimal*, pp. 97–108, 2019.

[7] M. Zemamou, A. Toumi, K. Mrigua, Y. Lahlou, and M. Aggour, "A novel blade design for Savonius wind turbine based on polynomial bezier curves for aerodynamic performance enhancement," *Int. J. Green Energy*, vol. 17, no. 11, pp. 652–665, Sep. 2020, doi: 10.1080/15435075.2020.1779077.

[8] P. H. Lumbangaol, "Energi Terbarukan Untuk Pengembangunan Berkelanjutan di Indonesia," *J. Energi*, pp. 1–14, 2016.

[9] J. L. Menet, "A double-step Savonius rotor for local production of electricity: A design study," *Renew. Energy*, vol. 29, no. 11, pp. 1843–1862, Sep. 2004, doi: 10.1016/j.renene.2004.02.011.

[10] O. B. Yaakob, D. T. Suprayogi, M. P. Abdul Ghani, and K. B. Tawi, "Experimental studies on savonius-type vertical axis turbine for low marine current velocity," *Int. J. Eng. Trans. A Basics*, vol. 26, no. 1, pp. 91–98, 2013, doi: 10.5829/idosi.ije.2013.26.01a.12.

[11] J. V. Akwa, G. Alves Da Silva Júnior, and A. P. Petry, "Discussion on the verification of the overlap ratio influence on performance coefficients of a Savonius wind rotor using computational fluid dynamics," *Renew. Energy*, vol. 38, no. 1, pp. 141–149, 2012, doi: 10.1016/j.renene.2011.07.013.

[12] M. Latif, "Eisiensi Prototipe Turbin Savonius pada Kecepatan Angin Rendah," 2013.

[13] M. H. Zawawi *et al.*, "A review: Fundamentals of computational fluid dynamics (CFD)," *AIP Conf. Proc.*, vol. 2030, no. November, 2018, doi: 10.1063/1.5066893.

[14] G. Ferrari, D. Federici, P. Schito, F. Inzoli, and R. Mereu, "CFD study of Savonius wind turbine: 3D model validation and parametric analysis," *Renew. Energy*, vol. 105, pp. 722–734, 2017, doi: 10.1016/j.renene.2016.12.077.

[15] I. Dobrev and F. Massouh, "CFD and PIV investigation of unsteady flow through Savonius wind turbine," *Energy Procedia*, vol. 6, pp. 711–720, 2011, doi: 10.1016/j.egypro.2011.05.081.