International Journal of Research in Engineering and Science (IJRES)
ISSN (Online): 2320-9364, ISSN (Print): 2320-9356
www.ijres.org Volume 10 Issue 5 ‖ 2022 ‖ PP. 13-17

A Review on Offshore Renewable Energies.

M. Harish[1] and M. Mohamed Younus[2]
[1]*Post Graduate student, Department of Civil Engineering, Government College of Engineering, Tirunelveli, Tamilnadu, India.*
[2]*Assistant Professor, Department of Civil Engineering, Government College of Engineering, Tirunelveli, Tamilnadu, India.*
Corresponding Author: M Harish

ABSTRACT:
The global demand for energy is increasing rapidly. As the supply of non-renewable energy sources are limited, renewable energy installations are increasing to meet the energy demand. Renewable energy systems can be installed both onshore and offshore. Some of the challenges of onshore installations are land availability and acquisition, achieving uninterrupted supply of energy, etc. To overcome these problems, renewable energy structure supported on floating structures installed in sea/marine environment is a promising solution. Numerous research activities have been carried out to understand the dynamics of such structures subjected to various environmental loadings. One of the commonly used numerical technique to obtain the dynamic response of the floating structure is the computational fluid dynamics (CFD) technique. CFD helps in understanding the flow field, motion of the floating rigid body arising due to wind, wave and current loads. In the paper, we present critical review of the research work done on the dynamics of floating renewable energy system using CFD technique.
KEY WORDS: *Offshore floating structures, computational fluid dynamics, Offshore wind energy, Offshore solar energy.*

Date of Submission: 24-04-2022 Date of acceptance: 06-05-2022

I. INTRODUCTION

Energy can be extracted from both renewable and non-renewable resources. Non-renewable resources include crude oil, natural gas, coal, and nuclear fuel. These non-renewable resources are transformed into valuable products. Non-renewable resources have a limited supply. Meanwhile, awareness about the ecological consequences, use of fossil fuel and its role in global warming is rising. The emission of greenhouse gases, particularly carbon dioxide and methane, which contribute to climate change, might be the most well-known impact of using non-renewable energy sources. Renewable energy is energy obtained from non-finite or non-exhaustible natural resources, such as wind and sunlight. Renewable energy is a viable alternative to traditional fossil-fuel based energy, and it is generally less harmful to the environment. Renewable energy sources such as solar, wind, hydro, geothermal and biomass are used to generate electricity. Solar, wind and water are abundantly available resources. Solar energy is captured and converted into electricity using solar panels and solar thermal technology. Solar energy is a clean, eco-friendly energy source and an effective technique to lower overall carbon footprint. Based on the structural installation, solar energy is classified into two categories. They are, land-based structure and floating structure. Floating structure is further classified into inland floating structure and offshore floating structure. Wind energy is primarily the use of wind turbines to create electricity. A wind turbine converts wind energy into electricity. Wave energy is a renewable energy source that is generated by the motion of the waves. Wave energy converters (WECs) transform the kinetic and potential energy of a moving ocean wave into mechanical or electrical energy.

II. LITERATURE REVIEW

Hadzic et. al (2005) concentrated on the hydrodynamics of the wave energy converter (WEC) body and its interaction with a wave and investigated a hierarchy of CFD methodologies. Linear theory and fully non-linear numerical approaches were necessary when exploring the hydrodynamics of wave-structure interaction. The viscous effects and surface effects of the wave can be simulated and when the viscous effects were not important, they were neglected. Then Navier-Stokes equations reduced to the Euler equations and it can be discretised into two or three dimensional mesh. Numerical models were finite volume method, finite element method, smoothed particle hydrodynamics and cartesian cut method. And the corresponding software packages used were STAR-CCM+, ANSYS CFX, Numerical approaches such as the well-known Finite Volume (FV) or Finite Element (FE) formulations can be used to compute the fluid velocities and pressures at every mesh cell or node. They have concluded from the findings that CFD is a versatile method capable of handling highly complicated and non-linear wave-structure interaction problems. Each of the CFD approaches evaluated accurately anticipated the wave structure interaction and agreed with experimental results in most cases.

Jan (2011) studied the latter design requirement by investigating the survivability of WECs. The efficiency of wave energy converter device was not an issue in normal sea conditions, but when storms occur, the WEC might be hit by extremely strong waves, which could damage the device. Scale model tank testing and linearised mathematical methodologies were used to examine this issue. Computational fluid dynamics (CFD) could provide an additional tool that could describe fluid-structure interaction in a fully non-linear and full-scale manner, including wave-breaking, over topping, and rigid body motion. This theory concluded that CFD could be coupled with less expensive simpler flow models, such as Boundary Element Methods and potential flow solvers, up to the point where nonlinear effects, such as wave breaking, required the use of more complex methods.

Yi-Hsiang and Ye (2012) investigated the nonlinear effect of the interaction between waves and the FPA system on the power generation performance. They found that hydrodynamics of a two-body FPA system were more complicated when compared to a single-body point absorber system, where the power generation performance was determined by the relative motion of the float and reaction portion. The issues concern the interaction of waves with the FPA system, including the floating and submerged parts, their relative motion, and the effect of the power take-off (PTO) mechanism. A two-body FPA system was analysed using a RANS-based CFD approach. The research shed light on the complex hydrodynamic interactions of a two-body FPA system in heave, and it could be useful for future research, such as determining the best control and tuning of the PTO system in real seas, where nonlinear interactions between waves and the FPA system were more significant.

Markus et. al. (2012) studied wave-current interaction and load analysis using finite volume method (FVM) based numerical wave tank. CFD studies were done in FLUENT software, where governing equations were discretized by the finite volume method. They used fenton model to create the depth varying currents. A simulation strategy that focused on capturing wave-current interaction was introduced and validated with respect to fluid particle kinematics. The results were found to be more successful but they used nonlinear waves with unsteady flow conditions using Unsteady Reynold's Averaged Navier-Stokes (URANS) equations and used stream function wave theory to validated his waves generated in the wave tank and compared the results of non-uniform current with uniform currents.

Jan (2014) investigated that different CFD approaches were used to study the highly complicated dynamics of wave energy converter (WEC) devices, which included multi-body-wave interaction and 6-degree-of-freedom motion. This study focussed on wave structure interaction. The first step was to simulate regular waves interacting with a fixed horizontal cylinder and the wave forces were examined here. Following that, the simulation of a cone-shaped body near the water surface was discussed, in which the interaction between the cone and the water was simplified to act in a single direction. Smooth Particle Hydrodynamics (SPH), a Cartesian Cut Cell method based on an artificial compressibility method with shock capturing for the interface (AMAZON), and two pressure-based Navier Stokes codes, one using a Finite Volume (FV) and the other a control volume Finite Element approach (CVFE), are applied to a fixed cylinder case as well as one motion simulation. The result showed that all the applied computational techniques are very powerful tools. In all experiments, the Cartesian-Cut-Cell method (AMAZON), SPH, and the two Navier-Stokes solvers, i.e., the FV and CV-FE approaches, performed well.

Kim et al (2016) studied wave-current interaction and load analysis using finite volume method (FVM) based numerical wave tank. CFD studies were done in FLUENT software, where governing equations were discretized by the finite volume method. They used 3rdorder Monotone upstream centred scheme (MUSCL) for convection term. The velocity and pressure coupled with Pressure implicit with the splitting of operator (PISO) method. The k-ε model was used for the turbulent stress term in the governing equation and volume of the fluid method with implicit High-resolution interface capturing (HRIC) for the air-water interface.

Zhiyin (2014) this study explained that a short-take-off and landing aircraft operates close to the ground, a complex flow field was formed. The configuration of twin impinging jets in a cross-flow can be used to simulate this flow field. The Reynolds averaged Navier–Stokes (RANS) technique (current industrial practise) makes accurate prediction of this complicated flow field extremely difficult. This paper used the Unsteady-RANS (URANS) technique with an RSM to numerically examine this flow. In terms of the mean flow field, both the SRANS and the URANS performed admirably, displaying a good trend of experimental results as well as good accuracy, with the URANS approach making somewhat superior forecasts. Particularly, for the vertical normal stress predictions where the URANS results not only followed the trend of the actual data well, but also had excellent quantitative agreement with the experimental data.

Nianxin et. al. (2014) studied the hydrodynamic loads of the floating wind turbine platform have been simulated by viscous numerical flume based on volume-of-fluid method. The dynamic responses of the floating wind turbine system could be obtained by solving the dynamic equations of the floating system under typical coupled wind-wave sea cases, and the motion of the floating platform under different time steps would be simulated by advanced sliding mesh technique. As a result, the fluid-structure interactions between the floating wind turbine structure and the coupled wind-wave flow field had been successfully simulated. Furthermore, the availability of the coupled numerical method had been verified by comparing the numerical results with corresponding scale model test data.

Tohid and Ali (2015) Investigated that between offshore structures and fluids in the form of oceanic flows or blast waves were common in offshore engineering applications. In order to build highly durable structures for usage in offshore environments, accurate assessment of the forces induced by these flows is critical. By solving the equations of fluid motion, CFD had been utilized to investigate fluid behavior and its impact on offshore constructions. However, resolving all of the flows spatial and temporal scales necessitates lengthy and expensive CFD calculations. Finally, used coupled CFD structural analysis, the impact of a drop object on a pipeline protection structure was examined.

Ali (2016) studied to explore wave load effects on fixed and free-floating offshore structures, a Numerical Wave Tank (NWT) based on Computational Fluid Dynamics (CFD) was built and proven. The model was based on solving Navier-Stokes equations on a structured grid, tracking the free surface with a level set approach, and studying wave-structure interaction with an immersed boundary method. The purpose of this work was to establish and verify a CFD-based NWT. The numerical results were compared to those predicted by potential flow theory, and experimental data were recorded. The time it takes for generated waves to be disturbed by reflection waves from an offshore structure was predicted theoretically and validated by comparing with CFD results based on wave group velocity. The numerical model was also used to investigate wave forces and moments on a 3D fixed surface penetrating circular cylinder for a wide range of wave frequencies, as well as three-dimensional effects of waves due to structure reflection.

Palm et al (2018) this research investigated that the nonlinear forces acting on a moored point-absorbing wave energy converter (WEC) in resonance at both prototype and model scales (1:16). The linear radiation-diffraction approach, Reynolds Averaged Navier–Stokes (RANS), and the Reynolds Averaged Navier–Stokes (RANS) simulations were employed (linear). The scale effects of the WEC were restricted by the solely viscous contribution (1–4%) for the two waves investigated in Euler simulations, which are proven to be scale-independent. The CFD prototype heave amplitudes were 10% and 24% smaller for W0 and W1, respectively, when compared to the linear simulation, owing to drag forces. Scale-independent generated drag effects have a significant impact on the heave drag force. The driving wave pressure was affected by the vorticity of the flow, which prevents it from acting on the entire region of the buoy, reducing the wave force amplitude.

Hanbin et al (2018) studied that offshore engineering frequently employed floating cylindrical bodies. The advanced CFD software STAR-CCM+ was used to simulate forced heave and surge motion of axisymmetric vertical cylindrical bodies with flat and rounded bases. The overset method was used so that the mesh local to the body moves within a stationary outer mesh. The viscous effects that cause drag as well as those that influence additional mass and radiation damping were studied. Different turbulence closure models were used using the Reynolds-Averaged Navier–Stokes (RANS) equations, and the water surface is captured using the volume of fluid (VOF) approach. The CFD data were fed into a basic dynamical model, which allows the drag, additional mass, and radiation damping factors to be calculated. In the laboratory, for bodies with rounded bases, a free heave decay test was performed without mechanical contact, and the inferred drag and added mass coefficients were extremely close to those obtained from CFD. This study determines how viscous effects influence drag coefficients and modify added mass to inform their magnitude for superimposing in linear diffraction modeling. Although numerical convergence was reached for the overall force with forced heave oscillations, it was not accomplished for the very small shear force.

Windt et al (2019) investigated that Numerical wave tank (CNWT) models based on CFD were a valuable tool for analyzing wave energy converters (WECs). Model validation is critical during the development of a CNWT to ensure that the numerical solution was accurate. The validation of a CNWT model for the 1:5 scale Wave star point-absorber device was presented in this research. The spring stiffness and damping coefficients in the numerical PTO model were determined by fitting the experimental PTO position, velocity, and force data with linear least squares fit. The system dynamics of the experimental setup can be recorded with satisfactory precision by incorporating a linear spring-damper system into the CNWT, with coefficients found via linear least squares regression. Validation of complicated physical systems, such as the one on hand, is a difficult undertaking that necessitates precise knowledge of system characteristics. It was difficult to obtain, yet it is necessary for the construction of assumptions for the numerical model. This might have an impact on the order of accuracy that can be achieved in validation trials.

Wendt et al (2019) studied that two different heaving bodies, they used numerical modeling techniques with varying levels of fidelity in terms of capturing nonlinear hydrodynamic effects. Because the water plane area and wetted surface area of the sphere alter when it was displaced in heave, nonlinear effects are introduced. For the heaving float instance, linear models tend to agree well with weakly and totally nonlinear models. They found that linear and weakly nonlinear codes agreed quite well. Simulating a large number of wave circumstances eventually results in a significant amount of meshing labour, which contributes to the CFD model's intrinsically high computational costs. This study concludes,in small and medium wave circumstances, the outcomes of applying linear, weakly nonlinear, and fully nonlinear models were identical.

Magnus (2019) investigated the vertical hydrodynamic response of a new proposal for a floating solar island. Moss Maritime designed the concept, which consists of a series of hinged similar stiff floating modules. As a result, a preliminary concept research using experimental approaches can be a good place to start. Structural rigidity, relative vertical motion flexibility, and low mooring effects were all achieved. Finally experimental results for regular waves demonstrated good handling when compared to the incident waves that were evaluated. The heave results revealed total wave period flexibility.

Michiel (2020) explained about the Offshore floating wind turbines can only withstand a certain amount of heave before failing. The DeepCwind floater for offshore floating wind turbines was designed to reduce heave motion. This semi-submersible floater was made up of three cylindrical columns, each having a heave plate at the bottom. To begin, a heave decay test was performed. The structure was also subjected to incoming waves. Finally, it was demonstrated that the potential flow model accurately predicts the column's response with and without a heave plate. Despite the fact that viscous flow effects contribute significantly to the damping of both structures, the response was unaffected because the total damping remains low.

Feys (2020) illustrated that, in the inland water bodies, a number of floating solar parks have been established. This inland technology was well-established at the moment. The unlimited availability of surface makes offshore areas appealing. An energy converting system, a position monitoring or mooring system, and a floating support structure are the three physical subsystems of an offshore floating PV. Several theories have been advanced in the literature or by business. Submerged beams form a triangle grid, which is kept together by floating buoys in the buoy and beam construction. Triangle platforms were supported by the buoys, which carry the platforms' corners. The buoys, beams, and platforms all have fully hinged connections. The buoy and beam structure's viability were evaluated.

Davidson and Costello (2020) this study expected that many million hours of operation must be simulated to develop and optimize a WEC, with one million hours of WEC simulation per year of the R&D program. Finally, there are a range of nonlinear PF methods existing or in development, with different levels of complexity. However, there does not appear to be a universal method for robustly adding viscous effects in PF-based simulations. Domain decomposition was a potential strategy that was gaining traction, in which lower-fidelity models are utilized for the majority of the simulation and are connected to more computationally expensive high-fidelity models that were restricted to the domains of interest.

Mikael and Max (2021) computational fluid dynamics were a quick and inexpensive approach to test alternative designs and environments. This paper compared and contrasted two different software programmes in order to determine their capabilities and limits. FINE^{TM}/Marine and OpenFOAM were the software used in this study. Three alternative meshing methods were examined in this research to discover how efficient and precise. Overset, sliding, and a deforming mesh technique were the different meshing techniques used. A sphere with a radius of 0.15 metre is dropped from 0.15 metre above the sea surface and the motion of the sphere was studied for six seconds. Experimental data was then used to validate the results of the various meshing approaches.

Kramer et al (2021) this paper explained about simulating the motions of floating bodies in the ocean, numerical models with complicated fluid structure interactions were frequently constructed. These models can be used to evaluate the effectiveness of wave energy devices. Heave degradation experiments on a 300mm diameter sphere were extremely accurate and exact. When at rest in calm water, the sphere was ballasted to half submersion, allowing it to float with the waterline at the equator. The sphere was held motionless and dropped from three different drop heights that was a short drop height, which can be regarded a linear example, a moderately nonlinear case, and a highly nonlinear instance, where the entire sphere was initially above the water. Finally, this paper concludes that the physical test findings had a high degree of precision, which was measured using time-varying systematic and random errors in the heave time series.

III. CONCLUSION

Following are the major conclusions derived from the literature study,

- The end results have to rely upon both experimental and analytical data.
- The physical test results were quite precise, as evidenced by time-varying systematic and random errors in the heave time series.
- The movements of floating bodies were predicted using the Navier-Stokes equations and body dynamics equations at the same time.
- In wave energy converter, domain decomposition is a prospective method that is gaining interest, in which lower-fidelity models are used for the majority of the simulation and are coupled to higher-fidelity models that are more computationally expensive.

REFERENCES

[1]. Bechkramer M., Anderson J., Thomas S., Buusbendixen F., Bingham H., Read R., Holk N., Ransley E., Brown S., Yu Y., Toantran T., Davidson J., Horvath C., Janson C., Nielsen K. and Eskilsson C. (2021), 'Highly Accurate Experimental Heave Decay Tests with a Floating Sphere: A Public Benchmark Dataset for Model Validation of Fluid–Structure Interaction', Energies 2021,14,269.

[2]. Bengtsson M. and Delvret M. (2021) 'Motion Decay of a floating object', Master's thesis in Sustainable Energy Systems, Innovative and Sustainable Chemical Engineering, Gothenburg, Swedan.

[3]. Darvishzadeh T. and Sari A. (2015), 'CFD Applications in Offshore Engineering', Offshore Technology Conference held in Houston, Texas, USA, May 4th -7th 2015.

[4]. Davidson J. and Costello R. (2020), 'Efficient Nonlinear Hydrodynamic models for wave energy converter Design - A scoping study', Journal of Marine science and Engineering, Vol.8, No.35, pp.1-65.

[5]. Feys A.B.Z. (2020), 'Support structure for offshore solar: The proposal of a new concept', Master of Science at the Delft University of Technology, to be defended publicly on June 26th 2020.

[6]. Gu H., Stansby P., Stallard T. and Carpinteromoreno E. (2018), 'Drag, added mass and radiation damping of oscillating vertical cylindrical bodies in heave and surge in Stillwater', Journal of fluids and structures, Vol.82, pp.343-356.

[7]. Hadzic I., Hennig J., Peric M. and Xingkaeding Y. (2005), 'Computation of flow-induced motion of floating bodies', Vol.29, pp.1196-1210.

[8]. Kim N., Kurian V. J. and Shahir L. (2011), "Wave Current interaction on Offshore Spar Platform", National Postgraduate Conference.

[9]. Markus D., Hojjat M., Wuchner R. and Bletzinger U. (2012), "A Numerical wave channel for the Design of Offshore Structure with Consideration of Wave-Current Interaction", Twenty-second International Offshore and Polar Engineering Conference, Rhodes, Greece.

[10]. Nematbakhsh A. (2016), 'Benchmarking of a CFD Based Numerical Wave tank for studying wave load effects on fixed and floating offshore structures', Journal of Offshore Mechanics and Arctic Engineering.

[11]. Nianxin R., Yugang L., and Jinping O. (2014), "Coupled wind-wave time domain analysis of floating offshore wind turbine based on computational fluid dynamics method", Journal of Renewable and Sustainable Energy, Volume: 6.

[12]. Onsrud M. (2019), 'An Experimental Study on the Wave-Induced Vertical Response of an Articulated Multi-Module Floating Solar Island', Master's thesis in Marine Technology, Norwegian University of Science and Technology.

[13]. Palm J., Eskilsson C., Bergdahl L. and Bensow R.E. (2018), 'Assessment of scale Effects, viscous forces and induced drag on a point - Absorbing wave energy converted by CFD simulations', Journal of Marine science and Engineering, Vol.6, No.124.

[14]. Wendt F., Nielson K., Yu Y., Bingham H., Eskilsson C., Kramer M., Babarit A., Bunnik T., Costello R., Crowley S., Gendron B., Giorgi G., Giorgi S., Girardin S., Greaves D., Heras P., Hoffman J., Islam H., Jakobsen K., Janson C., Jansson J., Yulkim H., Kim J., Kim K., Kurniawan A., Leoni M., Mathai T., Nam B., Park S., Rajagopalan K., Ransley E., Read R., Ringwood J.V., Miguelrodrigues J., Rosenthal B., Roy A., Ruehl K., Schofield P., Sheng W., Shiri A., Thomas S., Touzon I. and Yasutaka I. (2019), 'Ocean energy systems wave energy Modelling task: Modelling, Verification and Validation of wave energy converter', Journal of Marine science and Engineering, Energies 2019,7,379.

[15]. Westphalen J. (2011), 'Extreme wave loading on Offshore wave energy Devices using CFD', A thesis submitted to the University of Plymouth in partial fulfilment of the requirements for the degree of Doctor of Philosophy June 20th 2011.

[16]. Westphalen J., Greaves D.M., Raby A., Hu Z.Z., Causon D.M., Mingham C.G., Omidvar P., Stansby P.K. and Rogers B.D. (2014), 'Investigation of Wave-structure Interaction using state of the Art CFD Techniques', Vol.4, pp.18-43.

[17]. Williemtheodoor M. (2020), 'Comparison of potential flow and CFD for a column with Heave plate', European Wind Energy Master Offshore Engineering, Master of Science Thesis defence at the 24th of September 2020.

[18]. Windt C., Davidson J., Ransley E.J., Greaves D., Jakobsen M., Kramer M. and Ringwood J.V. (2019), 'Validation of a CFD based numerical wave tank model for the power production assessment of the wavestar ocean wave energy converter', International Journal of Renewable energy, ISSN.0960-1481, Vol.19, pp.1-53.

[19]. Yang Z. (2014), 'Assessment of unsteady-RANS approach against steady-RANS approach for predicting twin impinging jets in a crossflow', Cogent Engineering, ISSN.2331-1916.

[20]. Yu Y. and Li Y. (2012), 'Reynolds-Averaged Navier-stokes simulation of the heave performance of a two-body floating-point absorbed wave energy system', vol.73, pp.104-114.