## ENE 503 – Computational Fluid Dynamics

# WEEK 11: ARTICLE ASSIGNMENT 3

***ARTICLE ASSIGNMENT 3:***

- **Computational modelling of a Savonius type wave energy converter system:**

➢ Summary

Please summarize the numerical study conducted in the article

➢ Numerical modelling and simulation

In this section briefly discuss the following points

*- How is the flow geometry created?*
*- Specify the boundary conditions defined on the surfaces.*
*- Compare compatibilities for computational modeling and simulation.*
*- What kind of computational nodes are used in the study?*
*- Is the mesh independence test performed? If it is so, analyze the results comparatively.*
*- Is there a validation study for computational modeling and simulation?*
*- Is a parametric study performed? If so, what are the results?*
*- What are the effects of inflow turbulence? What are the best way of representing inflow turbulence on the flow of interest? Discuss.*

➢ Results of the article study

*- Summarize the results of the study. Are there qualitative and quantitative results both presented in this study? What do these results indicate? What are*

*the challenges of modelling of wake flow, vortex formation and their elongation in the wake region?*

# A study of energy conversion efficiency of a Savonius type wave energy converter system

**Mustafa Tutar**

Department of Mechanical and Manufacture, Mondragon Goi Eskola Politeknikoa, Loramendi 4, 20500, Arrasate, Spain. IKERBASQUE, Basque Foundation for Science,48011,Bilbao, Spain. E-mail: mtutar@mondragon.edu

**Ceyhan Erdem**

Department of Aerospace Engineering, Middle East Technical University, 06531, Ankara, Turkey. E-mail: ceyhanerdem@gmail.com

**Abstract** In the present study, two-dimensional, two-phase and turbulent flow around a horizontal axis 3-bladed Savonius rotor is considered. Numerical wave tank (NWT) simulations based on FVM/FDM technique in association with volume of fluid (VOF) element method are performed for specified values of wave heights for no-rotor flow case. Once validated against the theoretical data, the numerical simulations are extended to investigate the overall performance of the turbine over a very large range of wave height conditions for the rotor-flow case.

**Key words:** Wave energy conversion (WEC); Savonius rotor; VOF method; turbulence model.

## INTRODUCTION

Savonius type hydraulic turbines, in which the fluid energy is captured through a hydraulic mechanism rather than an aerodynamic mechanism, are considered to be simple, efficient with good starting capabilities and to operate at relatively low rotational speeds. There have been both experimental and computational studies [1-3] of energy efficiency and power performance analysis of Savonius turbines, which spin due to differential drag on the curved surface [2]. The net driving force, which can be attributed to the differential drag reproduced between the advancing blade(s) and the returning blade(s) can be increased by either reducing the reverse force on the returning blade(s) or increasing the positive force on the advancing blade(s).

The motion of the waves usually sets the water particles in orbital motion which can be considered to a combination of both longitudinal and traverse motions of water waves. In longitudinal motion, the water particles oscillate back and forth parallel to the wave propagation direction while in transverse motion, these particles oscillate up and down in their position. These two motions are later combined together to reproduce the overall orbital motion. The kinetic energy of the water particle´s orbital motion can be used to rotate the blades [2]. The relative power performance of the Savonius rotor is usually determined with the shape and size of the orbital motion with respect to the rotor diameter and their shapes are subject to change depending on the wave length to water depth aspect ratio. Therefore, the study of orbital motion of the mechanic waves and their characteristics with respect to size and positioning of the rotor is essential for a better optimization of design of such devices.

The present work aims at studying the planar regular wave propagation and its interaction with a horizontal Savonius rotor using numerical methods in a numerical wave tank (NWT) at different wave heights. These studies construct a basis for further investigation of the effect of different governing parameters on the performance of such ocean wave energy conversion devices for further application.

## EQUATION OF FLUID MOTION

The governing flow equations for the present 2-D turbulent flow behaviour are continuity, momentum i.e. Reynolds averaged Navier-Stokes (RANS) equations and turbulence transport equations. These conservation equations in non-linear differential form of vector notation for incompressible, viscous fluid flow conditions can be summarized below.

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \tag{2.1}$$

$$\rho \frac{\partial \bar{u}_i}{\partial t} + \rho \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{\partial \bar{P}}{\partial x_i} + \frac{\partial}{\partial x_j}\left( \mu \frac{\partial \bar{u}_i}{\partial x_j} - \rho \overline{u'_i u'_j} \right) + \rho g_i \tag{2.2}$$

In the above equations, $\rho$ is the fluid density, $\bar{u}_i$ is the time averaged velocity, $x_i$ is the coordinate direction, $u'_i$ is the deviation from the time averaged velocity, $\bar{P}$ is the time averaged pressure, $g_i$ is gravity acceleration, $\mu$ is the dynamic viscosity of the fluid, - $\rho \overline{u'_i u'_j}$ is the Reynold's stress tensor which is required to be modelled using a turbulence model for closure of RANS equations. The temporal and spatial co-ordinates correspond to *t* and $x_i$, respectively. In 2-D Cartesian co-ordinates the continuity and RANS equations can be re-defined by simply dropping the over bar for brevity for the time averaged quantity. In eddy viscosity based k-ε turbulence models, Reynolds stress tensor is related to the mean flow straining field for incompressible flow as below:

$$-\rho\overline{u_i'u_j'}=\frac{2}{3}\rho k\delta_{ij}+2\mu_t\overline{S}_{ij} \tag{2.3}$$

Where $k$ denotes the turbulence kinetic energy, $\mu_t$ is the eddy viscosity related to turbulence kinetic energy, $k$ and its dissipation rate,$\varepsilon$, and $\overline{S}_{ij}$ is the time averaged strain rate tensor related to mean velocity gradient in the flow. The turbulent kinetic energy, $k$ and its dissipation rate,$\varepsilon$ for isothermal are then defined by the turbulence transport equations to determine the eddy viscosity term, which is used to calculate the Reynolds stress term to closure the RANS equations for the present RNG $k$- $\varepsilon$ model.

For the present two-phase flow, the volume of fluid (VOF) element method initially proposed by Hirt and Nichols [4] is used for free surface tracking a surface in a fixed Eulerian mesh. In the VOF method, a single set of momentum equations is shared by the fluids and the volume fractions of each of the fluids in each computational cell are tracked through domain. Interface tracking where variables and properties in any given cell are purely represented by either one of phases or mixture of phases depending upon the volume fraction values:

$$\begin{cases} f_i = 0 & \text{The cell is empty (of } i^{th} \text{ cell)} \\ f_i = 1 & \text{The cell is full (of } i^{th} \text{ cell)} \\ 0 < f_i < 1 & \text{The cell contains the interface} \end{cases} \tag{2.4}$$

The fractional volume function is governed by a transport equation and this equation determines the movement of interface position such that:

$$\frac{\partial f}{\partial t}+v\cdot\nabla f=0 \tag{2.5}$$

## COMPUTATIONAL DETAILS

A 2-D schematic diagram of a numerical wave tank (NWT) which is constructed as a representation of the experimental wave tank (EWT) study of Hindasageri et al. [5] is shown in figure 1. The Savonius rotor, which is placed at water submerged level of z = 0 m in the NWT, is generated with the same geometric dimensions in accordance with that used in the experiment. The boundaries are also illustrated in figure 1. At the left boundary, inflow boundary conditions are imposed to generate a wave train coming from the x = 0 m. A sinusoidal wave boundary condition with a Second-order Stokes wave formulation is implemented here.

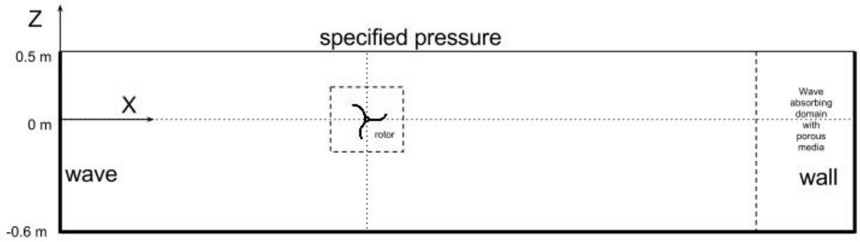


**Fig.1.** A schematic of NWT generated in the present numerical study

Based on this formulation, at the inflow boundary, the following below velocity components are employed as inflow velocity boundary conditions:

$$u=\frac{H}{2}\frac{gk}{\omega}\frac{\cosh k(h+z)}{\cosh kh}\cos(kx-\omega t)+\frac{3H^2\omega k}{16}\frac{\cosh 2k(h+z)}{\sinh^4(kh)}\cos 2(kx-\omega t) \tag{2.6}$$

$$w=\frac{H}{2}\frac{gk}{\omega}\frac{\sinh k(h+z)}{\cosh kh}\sin(kx-\omega t)+\frac{3H^2\omega k}{16}\frac{\sinh 2k(h+z)}{\sinh^4(kh)}\sin 2(kx-\omega t) \tag{2-7}$$

Where $H$ is the wave height, ω is the wave frequency, $k$ is the wave number, and h is the mean wave depth. The wave absorbing domain- porous media on the other hand is defined in the flow exit zone to prevent the wave reflection into computational domain. No-slip boundary conditions at the bottom surface and the flow exit domain i.e. outflow boundary are also imposed. At the free surface kinematic and dynamic boundary conditions with the specified atmospheric pressure condition is imposed to make sure that no transport equations are resolved in the air region (i.e. air is not treated as a fluid but rather as avoid).

The multi-block meshing modeling is utilized for more efficient use of computational resources and fast flow solution. The computational domain is constructed with non-uniformly spaced 2-D quadrilateral Cartesian mesh elements (figure 2) with fine resolution near wall surfaces and interface between air and water to successfully resolve the air-water surface movement due to wave propagation and to improve the numerical accuracy for measuring velocity and pressure gradients.

A FDM/FVM based numerical flow modelling approach of FLOW-3D [6] is used to solve the partial differential equations (continuity, momentum and energy equations) governing rotor movement and surface tracking here. To increase the convergence rate, momentum equations and the pressure based continuity equation are also coupled with a pressure-velocity coupling scheme of Generalized Minimal Residual Solver (GMRES) scheme [7] and the first order upwind scheme for discretization of the momentum equation. The one fluid VOF model is chosen for free surface, the Fractional Areas/Volumes Obstacle Representation (FAVOR) is chosen for efficient geometry definition.

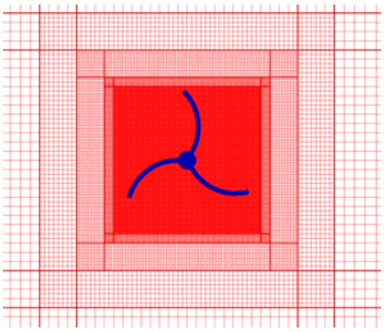

**Fig. 2**. The multi block meshes employed in the computational domain

## THE RESULTS and DISCUSSION

Initial computations are performed for no-rotor flow problem with three different mesh resolutions (maximum number of cells is around 115,000) at wave height of H = 100 mm for wave period of 2.1 s, water height of 0.6 m, and a wave length of 4.62785 m. As seen in figure 3, the numerical data corresponds well with the theory for all mesh resolution with a better agreement with the final mesh configuration of around 115,000 cells, which is found to be in very good correspondence with the theory. The shape of the surface waves is found to be almost sinusoidal throughout the domain.

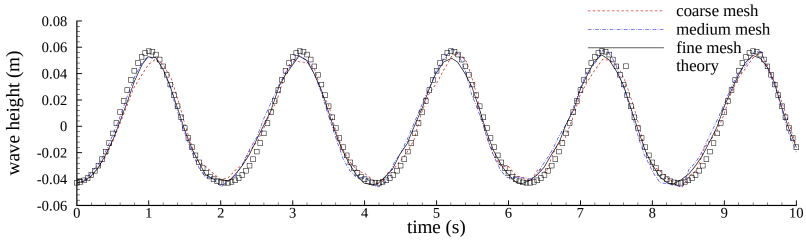


**Fig. 3**. Wave elevation history at a probe position of x =30 m for a wave height of *H* = 100 mm

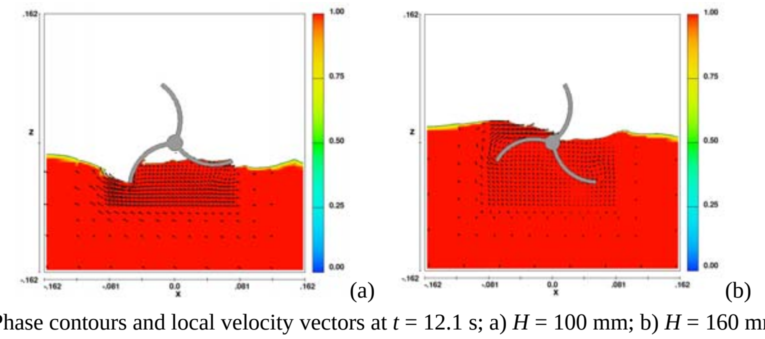


**Fig. 4**. Phase contours and local velocity vectors at *t* = 12.1 s; a) *H* = 100 mm; b) *H* = 160 mm

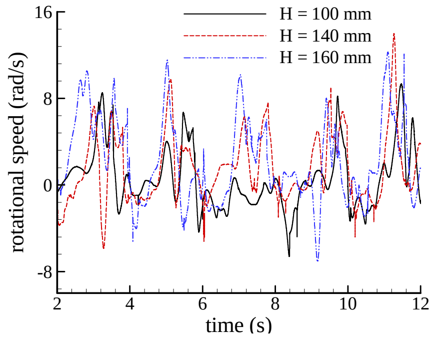


**Fig. 5.** Time evolution of rotational speed at different wave height values.

Distinguished phase contours and corresponding velocity vectors obtained for each case as in figure 4 can be attributed to differing wave propagation conditions and energy conversion rate with varying wave height. The maximum rotational speed obtained in the clock wise direction and slightly higher rotational speed is obtained as the wave height increases as an indication of positive effect of wave height increase on the rotational torque as shown in figure 5.

## CONCLUSIONS

The present numerical model successfully reproduces planar mechanic wave propagation. The higher wave heights the higher rotational speed of the Savonius rotor. Non-continuous flow through the rotor causes fluctuating rotational motion.

**Acknowledgments**

This work was funded under the project of the Gobierno Vasco- Basica y Aplicada- PI 2011-8.

REFERENCES

[1] Nakajima M., Lio S., and Ikeda, T.: Performance of Double-Step Savonius Rotor for Environmentally Friendly Hydraulic Turbine. *J. of Fluid Science and Technology* 3(3), 410-419 (2008)
[2] Faizal M., Ahmed M.R., and Lee Y-H.: On Utilizing the Orbital Motion in Water Waves to Drive a Savonius Rotor. *Renewable Energy*, 35,164-169 (2010)
[3] Zullah M.A., Lee Y-H.: Performance Evaluation of a Direct Drive Wave Energy Converter Using CFD. *Renewable Energy*,1-5 (2012)
[4] Hirt C.W. and Nichols B.D.: Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries. *Journal of Computational Physics* 39,201 (1981)
[5] Hindasageri V., Ramesh H., and Kattimani S.C.: Performance of Savonius rotors for Utilizing the Orbital Motion of Ocean Waves in Shallow Waters. *J. of Sustainable Energy & Environment* 2, 117-119 (2012)
[6] FLOW 3D, User Manual Version 10.0.1, Flow Science. Inc., Santa Fe (2012)
[7] Saad Y., and Schultz M.H.: GMRES: A Generalized Minimal Residual Algorithm for Solving Nonsymmetric Linear Systems. *SIAM J. Sci. Stat. Comput.*7,856-869 (1986).