11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
E. Oñate, J. Oliver and A. Huerta (Eds)

COMPUTATIONAL STUDY OF THE INTERACTION BETWEEN HYDRODYNAMICS AND RIGID BODY DYNAMICS OF A DARRIEUS TYPE H TURBINE

Diana Meneses[1], Omar López[2] and Santiago Lain[3]

[1] Graduate Research Assistant, Department of Mechanical Engineering, Universidad de los Andes
Cra 1 Este Nº 19A- 40 Bogotá (Colombia)
Email: dp.meneses60@uniandes.edu.co
[2] Assitant Professor. Department of Mechanical Engineering, Universidad de los Andes
Cra 1 Este Nº 19A- 40 Bogotá (Colombia)
Email: od.lopez20@uniandes.edu.co
[3] Professor, Department of Energetics and Mechanics, Universidad Autónoma de Occidente
Calle 25 # 115 - 85, Cali (Colombia)
Email: slain@uao.edu.co

Key Words: *Computational fluid dynamics (CFD), vertical-axis water turbine (VAWT), Darrieus turbine.*

Abstract. The present study discusses two-dimensional numerical simulations of a cross-flow vertical-axis marine (Water) turbine (straight-bladed Darrieus type) with particular emphasis on the hydrodynamic of the flow. Numerical investigations of a model turbine were undertaken using developed computational models. The turbine model was implemented in a commercial solver (ANSYS-FLUENT v14.5). The primary turbine operational variables of interest were torque, power and runaway speed. The domain and mesh were generated using a glyph script in POINTWISE-GRIDGEN. For the simulation, a sliding mesh technique was used in order to model the rotation of the turbine; a shear stress transport k-ω turbulence model was used to model the turbulent features of the flow. In order to simulate the interaction between the dynamics of the flow and the Rigid Body Dynamics (RBD) of the turbine a User Define Function (UDF) was generated. The proposed model has the capabilities to test several operation and transient conditions of the turbine, so it constitutes an interesting design tool.

1. INTRODUCTION

Industrialization and overpopulation have generated excessive use of no-renewable energy sources especially oil, but this is coming to an end for two reasons: first the scarcity of these resources and second and most important is the awareness of environmental problems such as global warming. This has intensified the search for new and better energy sources, which must be more environmentally friendly. The kinetic energy of sea water contained in marine current is one of the key elements in this quest with a very high potential of success.

Turbines used in marine aplicattions can be classified into two types: the first is the horizontal axial turbine, which has the axis of rotation in the direction of flow (e.g: Tidal turbines), the second type is the vertical axis turbine which has the axis of rotation perpendicular to the water flow. The latter type is ideal when the current can change direction as they are designed so they do not need a yaw mechanism to take advantage of the current.

The study of flow dynamics in vertical turbines has had a great interest in recent years by the scientific community, particularly in relation to improving efficiencies and understanding of their performance. In the specific case of the Darrieus type turbine, which was patented in 1927, still requires an understanding especially for hydrodynamic applications. Computational Fluid Dynamics (CFD) has proven to be a useful tool for flow analysis around these devices. For example, Lain et al [1] shows numerical results for the moment coefficient (Cm) using CFD, validation of these results with experimental data is quite satisfactory. Simao [3] shows and discusses an experimental - computational study on the aerodynamics and performance of a small scale vertical axis turbine. The authors obtained two important conclusions: Experimentally, it was observed that the roughness of the surface finish blades has significant effects on the turbine performance and computationally it was observed that the predictions found in the 3D model are significantly lower than in 2D by the presence of tip vortices. Nabavi [4] computationally studied the influence of various changes of the test duct in a turbine, in order to obtain the optimal shape to increase the power generated by the turbine. Lee [7] made a computationally study about a Savonius turbine performance when the angular velocity was not constant but with the RBD equation solution. Lee also implemented an equation for the resistive torque for direct-drive generator, the result for the study was a tool for the turbines design and research.

Given the need that still exists in the scientific community by the application of CFD models in the study of vertical turbines, the present work aims to study the interaction between the hydrodynamics and the rigid body dynamics of a Darrieus type H turbine using a two-dimensional CFD model. The objective for this project was to analyze the turbine performance when the solid rigid dynamic equation was implemented in the simulation. This project is part of a sequence of projects to design a turbine.

1 Methodology

1.1 Base Case

The present work uses the turbine model of reference [1], which has been previously studied both computationally and experimentally [2]. The parameters of this base case are specified in Table 1.

Table 1. Parameters of case base turbine [1]

Parameter	***Value***
Radio (R)	0,45 m
Number of blades	3
Airfoil	NACA0025
Chord	132,75 mm
Tip speed ratio (λ)	1.745
Angular velocity	6.28 rad/s
Solidity	0.89
Blade length (wingspan)	0,7 m
Reference Area	0,63 m²
Fluid	Water @ 20°C

The computational domain used in reference [1] for the simulation of the turbine is shown in Figure 1. The domain is subdivided in two: one domain near the turbine which is rotational and another far from the turbine which is fixed. Between the two domains exits an interface that allows continuous flux of mass and momentum. The rotational domain has the following dimensions De=2.8R and Di=1.1R, while the fixed domain has h=5R y l=16R.

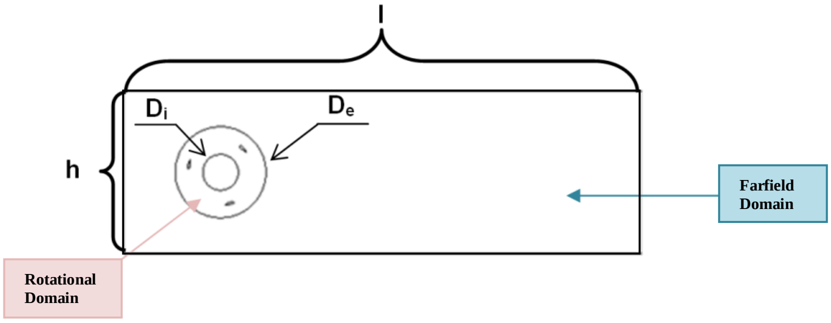

Figure 1. Computational domain for the base case

Figure 2 shows some details of the generated mesh in which it is observed the characteristics of the mesh close to the airfoil surface. In this area a hyperbolic mesh generation scheme was used, in which the size of the first element and the growth rate of the mesh in the normal direction to the surface was controlled. In this part of the domain quadrilateral elements were used, while for the rotational domain triangular elements were used. The transition between the structured and the unstructured mesh was generated as smooth as possible. Finally, the outer domain was meshed ensuring a smooth transition at the interface. Through a convergence analysis performed in reference [1] it was concluded that a mesh of about 100k elements was sufficient to ensure that the results are independent of the mesh size at a moderate computational cost.

Figure 3 shows the different boundary conditions for the computational domain that were set as follows:

- Velocity inlet: The freestream velocity is 1.62 m / s.

- Slip walls: Top and bottom planes of the domain have translational motion r of 1.62 m / s.
- Pressure outlet: Boundary condition for the flow at the outlet of the domain, with a value equal to the atmospheric pressure.
- Wall: zero velocity (no slip) at the surfaces of the turbine blades.

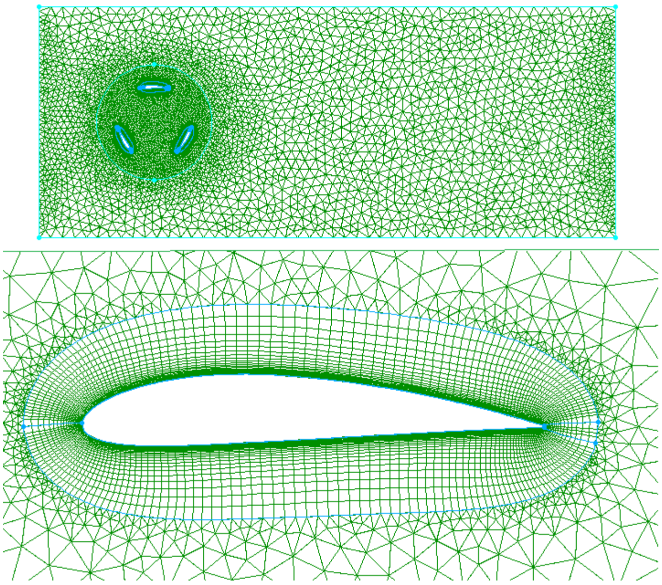

Figure 2. Mesh details.

Figure 3. Boundary conditions.

The commercial software ANSYS / FLUENT v14 was used as a flow solver , in which a scheme of sliding mesh between fixed and rotating domains was used . An incompressible, Newtonian and turbulent flow is assumed. The turbulence model selected for the base case simulation was k-ω SST, the choice was made based on previous work. Spatial and temporal discretizations of first order for all equations and the SIMPLE (Semi-Implicit Method for Pressure Linked Equations) scheme for the solution of the governing equations were used. To advance in time, a time step (Δt) of 0.005 s was used and 40 iterations were performed per time step.

1.2 Output Variables

The interpretation of the results was performed by comparing quantitative and qualitative global variable. Relevant to the present study quantitative variables were: coefficients of moment (Cm), tangential force (Ct) and normal force (Cn). Cm is a dimensionless number that is related to the torque output of the turbine. This is the torque that rotates the rotor in response to the force exerted by the fluid on the blades. Equation (1) shows its definition, where ρ is the fluid density, A_{ref} is the frontal area of the turbine 2RH (H is the wingspan) and V is the velocity of the incident flow.

$$C_m = \frac{M}{\frac{1}{2} \cdot \rho \cdot V^2 \cdot R \cdot A_{ref}} \tag{1}$$

The runaway speed is other important concept in the study because it is the speed in which the turbine runs at the maximum possible speed according to the conditions in which it is involved.

These global variables not only allow us to have a clearer idea of some key variables in the correct operation of the turbine torque and power. Qualitative variables provide information about the dynamics of turbulent flow around the turbine, so velocity; pressure, vorticity and turbulent viscosity fields were also analyzed in the computational domain.

1.3 RBD and CFD coupling.

To solve the Rigid Body Dynamics of the turbine, the solution of equation (4) has to be coupled with the flow solver. This coupling was implemented in a loose way using an UDF, which is a function that is programmed in C++ and can be coupled to the FLUENT solver. UDFs are defined using macros DEFINE which has FLUENT predesigned for different actions to be executed. Two macros already defined in FLUENT were used DEFINE_EXECUTE_AT_END and DEFINE_CG_MOTION. The first one is executed at the end of each time step and it is used to calculate the instantaneous aerodynamic torque of the turbine and to numerically solve equation 4 with an explicit Euler scheme. DEFINE_CG_MOTION macro is used to update de angular velocity and to control the instantaneous motion of the rotating domain. Figure 4 shows the model algorithm implemented.

$$\sum T = I \frac{d\omega}{dt} \tag{4}$$

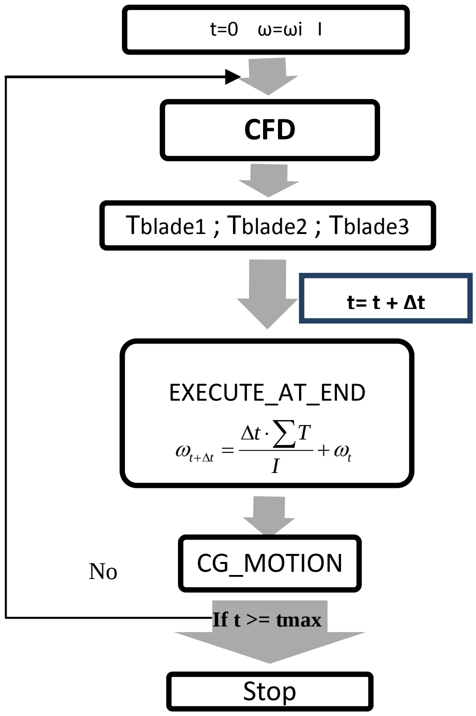

Figure 4. Algorithm implemented

2 Results

2.1 Runaway angular velocity for the base case.

The first test performed in the implemented model was the estimation of the runaway velocity of the turbine of the base case. It was assume that the blades were made of polyester resin so that the moment of inertia of the turbine relative to its center of rotation was 1.3 kg m2. The initial condition of the flow was impulsively started and at the same time the turbine started to rotated from rest. Figure 5 shows the evolution of the aerodynamic torque, it is clear that during the first second there is a transient evolution of the turbine torque until it reaches a periodic behavior. The averaged output moment is zero because there is not a resistance torque (runaway condition) but the instantaneous torque coefficient is approximately 0.1.

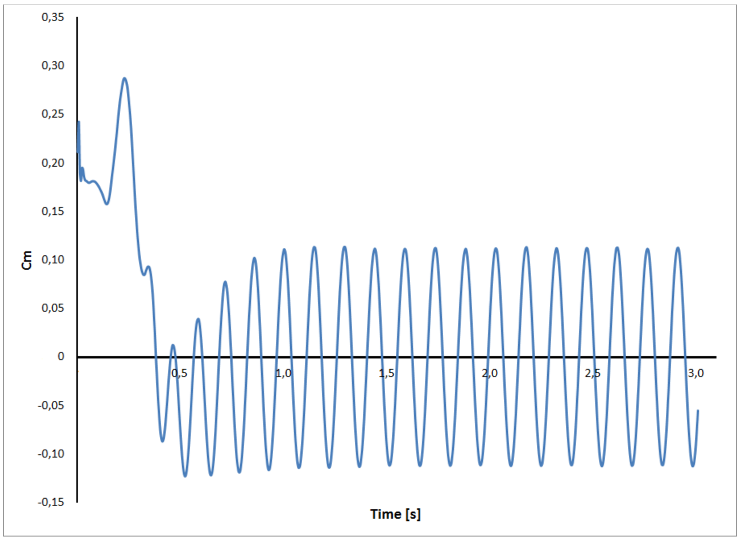

Figure 5. Time evolution-Cm.

Figure 6 shows the evolution of the angular velocity of the turbine. It is observed that the angular speed starts at rest (initial condition) and that the angular speed increases very rapidly, so that in less than 0.5 s it reaches a maximum angular speed of 19 rad/s. An overshoot in the evolution of the runaway angular velocity of the turbine is appreciated. Between 0.5 s and 1 s the angular velocity decreases in a transient evolution until it reaches a periodic behavior. The average runaway angular velocity of the turbine is approximately 14.3 rad/seg.

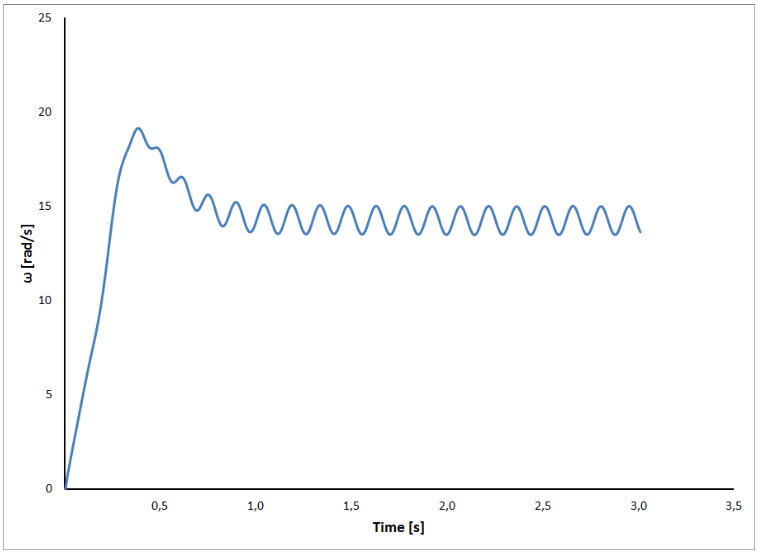

Figure 6. Angular velocity vs time for base case.

2.2 Influence of the freestream velocity in the runaway angular velocity

In order to explore the turbine performance it was decided to test 4 different freestream velocities (1, 1.62, 2 and 3 m/seg) corresponding to tip velocity ratio of XX, YY, ZZ and WW. Figures 7 and 8 show that the runaway angular velocity increases as the free stream velocity inlet increases.

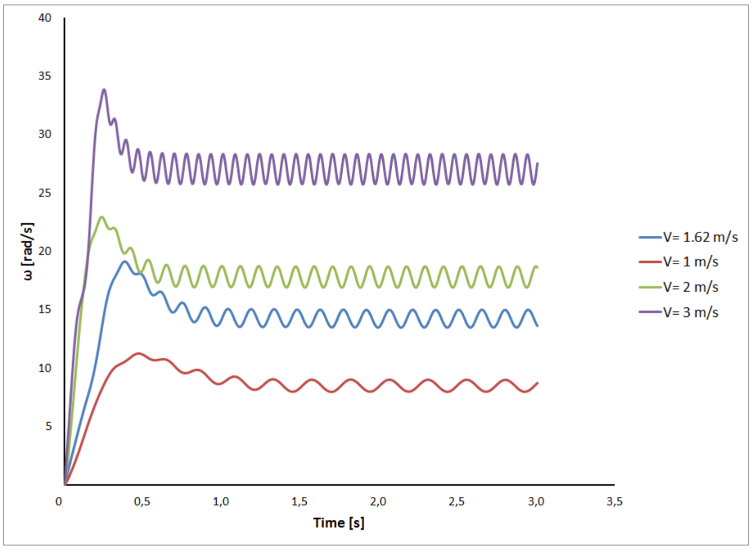

Figure 7. Angular velocity vs time for different velocity inlets.

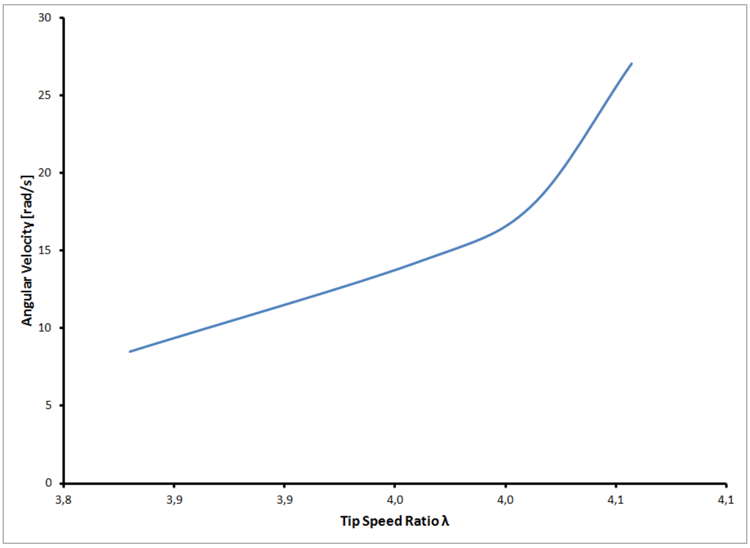

Figure 8. Angular velocity vs tip speed ratio.

2.3 Influence of the turbine moment of inertia in the turbine performance

Figure 9 shows the evolution of the angular velocity for 4 different moments of inertia of the turbine (blades made of 4 different materials). It is clear that as the moment of inertia increases the settling time (Figure 10) increases, but the average runaway angular velocity is the same for the 4 different turbines. It is also observed that as the moment of inertia increases the overshoot in the evolution of the angular velocity decreases and the rotation of the turbine is smoother (amplitude of the periodic behavior of the angular velocity decreases).

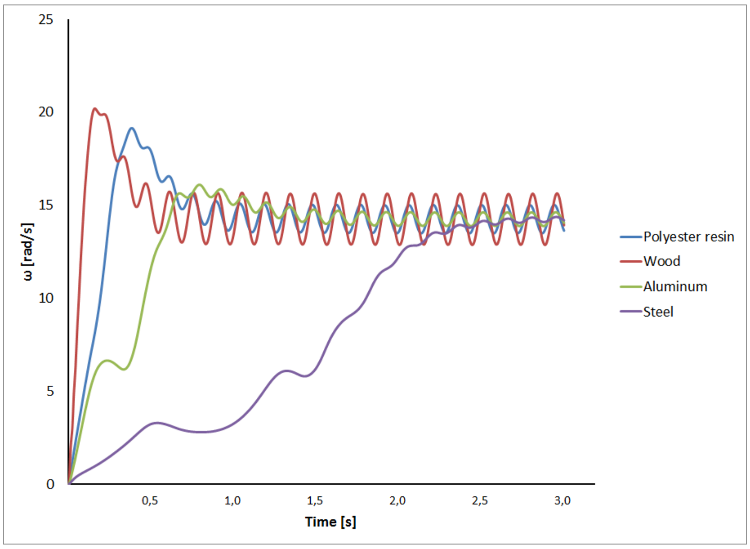

Figure 9. Angular velocity vs time for different materials.

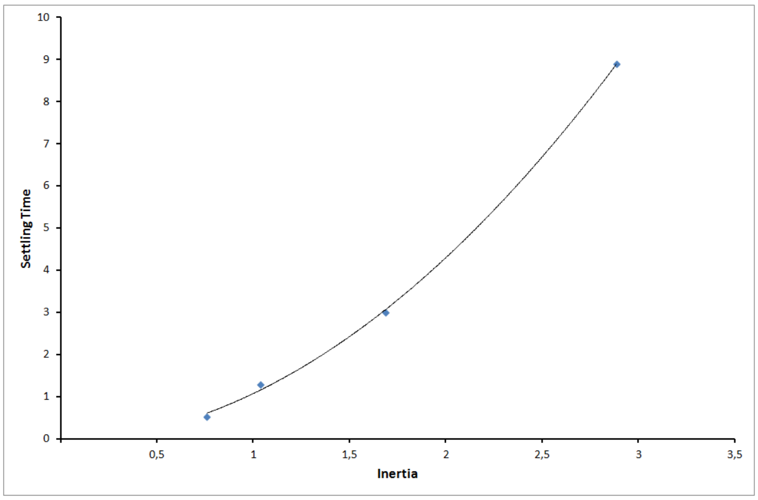

Figure 10. Settling time vs inertia

3 Conclusions

A computational study using an UDF to implement rigid body dynamics in two-dimensional solid modeling Darrieus type turbine was presented and discussed. The UDF was used as a design tool to explore the influence of the freestream velocity and the inertia on the turbine performance. Numerical results show that when the freestream velocity is increased, the runaway angular speed of the turbine increases. The increased inertia of the turbine does not have influence on the average value of the angular velocity but causes an increase in the time taken in stabilizing the turbine, the trend for the setting time is quadratic (Figure 9). The results for the study are important because it was possible to demonstrate that the inertia is not a factor which can increase the output power so it is important because the designer does not need to overdimension the turbine as a preventive measure. Also the study shows the importance of the freestream velocity and this parameter is essential for the design because depends on the operation conditions the designer need to design according to the runaway speed to have a good final prototype depends on the place in which it going to be install.

4 Acknowledgment

This project was sponsored by the Young Researchers program from the Colombian administrative department of science, technology and innovation (Colciencias).

REFERENCES

[1] Meneses, D. “Estudio Computacional de la Influencia del Tipo de Alabe en el desempeño Hidrodinámico de una Turbina Tipo Darrieus”. Undergraduate thesis. Universidad de los Andes, 2012.(In spanish)

[2] Lain, S y Osorio, C. “Simulation and evaluation of a straight-bladed Darrieus-type cross flow marine turbine” Journal of Scientific & Industrial Research, pp. 906-912. 2010

[3] Simao, C. “The near wake of the VAWT 2D and 3D views of the VAWT aerodynamics.” PhD thesis. TU Delft, Netherlands, 2009.

[4] Nabavi, Y. “Numerical Study of the Duct Shape Effect on the Performance of a Ducted Vertical Axis Tidal Turbine.” Master of Applied Science thesis. s.l. : The University of British Columbia, April of 2008

[5] Launder, B. E., Reece, G. J. and Rodi, W. (1975). Progress in the Development of a Reynolds-stress Turbulence Closure, J. Fluid Mech., Vol. 68, Pt 3, pp. 537–566.

[6] T.-H. Shih, W. W. Liou, A. Shabbir, and J. Zhu. A New k-ε Eddy-Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation. Computers Fluids, 24(3):227-238, 1995.