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)
July 20 - 25, 2014, Barcelona, Spain

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

D. Meneses[1], O. López[2] and S. Lain[3]

[1] Graduate Research Assistant. Universidad de los Andes, Cra 1 Nº 18A- 12 Bogotá (Colombia), dp.meneses60@uniandes.edu.co

[2] Assitant Professor. Universidad de los Andes, Cra 1 Nº 18A- 12 Bogotá (Colombia), od.lopez20@uniandes.edu.co

[3] Professor. Universidad Autónoma de Occidente, Calle 25 # 115 - 85, Cali (Colombia), slain@uao.edu.co

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

Global warming is nowadays an important problem that is facing mankind and in consequence renewable energies constitute an important source to substitute the use of fossil fuels. Among them, hydro-power represents an alternative source with a huge potential. Presently, the classical exploitation of hydro-power is on rivers and canals but oceans represent a potential energy source, especially kinetic energy of tidal currents. Kinetic energy can be extracted directly from the tidal stream and this procedure has a low environmental impact.

The study of vertical axis turbines has gained great interest in recent years due to its great potential. Analysis tools such as computational fluid dynamics (CFD) are being used in order to understand the dynamics of the flow around vertical axis turbines. The present study presents 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 blades tangential and normal forces. The domain and mesh was generated using a glyph script in POINTWISE-GRIDGEN. For the simulation a sliding mesh technique [see Figure 1] was used in order to simulate the rotation of the turbine, a shear stress transport k-ω turbulence model was used to model turbulent features of the flow.

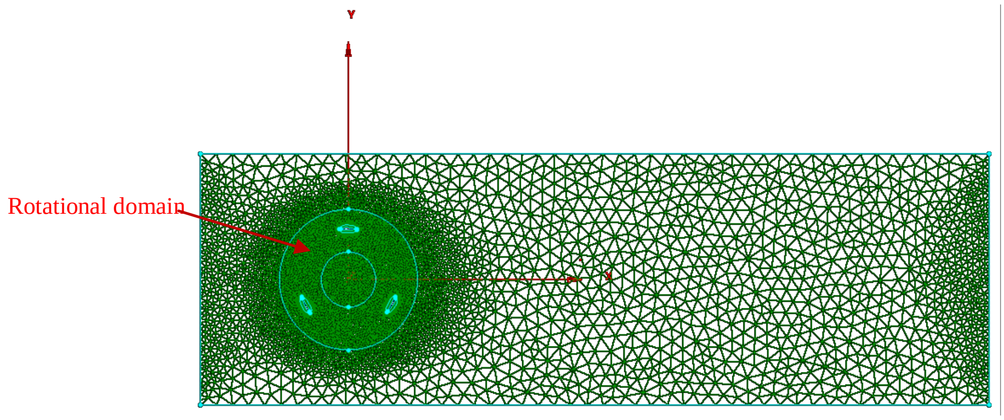

Figure 1. 2D Mesh for Darrieus turbine.

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. This UDF solves the RBD equation of the turbine [Eq. 1] in order to predict the angular velocity of the rotor at every time step. A loose coupling is used between the CFD and the numerical solution of equation 1.

$$\sum T = I\alpha \qquad \text{Eq.1}$$

Numerical results of the simulations show an acceptable performance of the UDF when compared to other numerical and experimental results reported in the literature. The proposed model is a tool that can be used in the future to make a parametric study to determine the influence of other parameters like turbine inertia, frictional torque and turbine geometry in the turbine operational variables.

REFERENCES

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

[2] D. Meneses. Estudio Computacional de la Influencia del Tipo de Alabe en el desempeño Hidrodinámico de una Turbina Tipo Darrieus: Proyecto de grado, Universidad de los Andes, 2012. (In Spanish).

[3] Y. Nabavi. Numerical Study of the Duct Shape Effect on the Performance of a Ducted Vertical Axis Tidal Turbine. s.l. : Msc Thesis, The University of British Columbia. 2008.

[4] C. Simao. The near wake of the VAWT 2D and 3D views of the VAWT aerodynamics. Dissertation, TU Delft, 2009.