Research Journal of Applied Sciences, Engineering and Technology 5(1): 233-238, 2013
DOI:10.19026/rjaset.5.5111
ISSN: 2040-7459; e-ISSN: 2040-7467

Submitted: May 30, 2012 Accepted: June 29, 2012 Published: January 01, 2013

Research Article

CFD Simulation of Twin Vertical Axis Tidal Turbines System

Syed Shah Khalid, Zhang Liang and Sheng Qi-hu
Deep Water Engineering Research Centre, Harbin Engineering University, China

Abstract: As concerns about rising fossil-fuel prices, energy security and climate-change increase, renewable energy can play a vital role in producing local, clean and inexhaustible energy to supply world rising demand for electricity. In this study, hydrodynamic analysis of vertical axis tidal turbine operating side-by-side is numerically analyzed. Two-dimensional numerical modeling of the unsteady flow through the blades of the turbine is performed using ANSYS CFX, hereafter CFX; this is based on a Reynolds-Averaged Navier-Stokes (RANS) model. The purpose is to find an optimal distance between the turbines where interaction effect is minimal and constructive, where the turbines operate more efficiently than stand alone turbine. A transient simulation is done on Vertical Axis Tidal Turbine (VATT) using the Shear Stress Transport Turbulence (SST) model. Main hydrodynamic parameters like torque T, coefficients of performance C_P and coefficient of torque C_T are investigated. The gap spacing between the turbines has an important role in performance improvement and also in vortex shedding suppression for the flows around two counters rotating systems. The simulation results are validated with Ye and Calisal data. The results of this study prove that the total power output of a twin-turbine system with an optimal layout can be about 24% higher than two times that of a stand-alone turbine. We conclude that the optimally configured counter-rotating twin turbines should be a side-by-side arrangement.

Keywords: CFD simulation, hydrodynamic interaction, renewable energy, tidal energy, twin turbine, vertical axis tidal turbine

INTRODUCTION

Tidal energy is one of the most important forms of Renewable Energy (RE) which can be used without depleting natural resources such has fossil fuels and wood with minimal environmental effects. It does not rely on the burning of a fossil fuel to create power. Because there are little or no fuel costs associated with generating electricity from renewable sources, more researchers are looking to resources like wind, geothermal, hydropower, tides, waves, solar and biomass to hedge against the price volatility of natural gas and diesel. RE systems generally require less maintenance than traditional generators and engines. Their fuel being derived from natural and available resources reduce the costs of operation. RE projects can also bring economic benefits to many regional areas, as most projects are located away from large urban centers and suburbs of the capital cities.

So, it's easy to generate it locally as to avoid huge cost if supplying from the main distributing power house. For more detail on renewable-energy resources see the review study Rourke *et al*. (2009) and Charlier (2003). Regarding harnessing the tidal energy two main sources are horizontal and vertical axis turbines. This division is based on turbine axis with respect to flow direction. When the flow direction is parallel to the axis of rotation, the turbine is horizontal axis turbine and when the flow direction is perpendicular to the axis of rotation, the turbine is a vertical axis. For more detail on the difference between vertical axis and horizontal axis turbine concern the review studies Batten *et al.* (2007) and Shikha (2005).

Many investigations of stand-alone turbines have been reported some of them, (Wang *et al*., 2009; Ye *et al*., 2010; Antheaume *et al*., 2006). Mostly stand alone system produce energies in Kilowatts, to get energy in megawatts the concept of tidal fence and tidal farm were introduced. In which more than one turbine can be arranged in suitable configuration. A little contribution is done on this side and especially on the twin turbine system. Some work on Twin turbine with horizontal arrangement was done by marine current turbine Ltd. the world's first twin-turbine system with horizontal axis was deployed by Marine Current Turbine in 2008. Furthermore, Clarke *et al*. (2007) design and experimentally tested the horizontal axis twin turbines. A systematic study on VATT with Achard model was done by Georgescu *et al*. (2010). However, the Achard turbine consists of delta shape blades not straight as shown in Fig. 1.

A detailed study on the twin turbine system was done by Ye and Calisal (2010b) for both co-rotating and counter rotating arrangements. The numerical

Corresponding Author: Syed Shah Khalid, Deep Water Engineering Research Centre, Harbin Engineering University

Fig. 1: Tidal farm concept by Achard turbines

computation was done using a programme known as DVM-UBC. The present effort is done to simulate the twin turbine system using commercial available software CFX. This study is one part of the research on the design and development of double VATT platform.

Modeling and simulation of single turbine: First, we simulate a single turbine using ANSYS ICEM CFX. The main design parameters are: number of blades (N) = 4, Blade profile NACA 0018, chord length C = 0.6 m, turbine solidity ($\sigma = NC/\pi D$) = 0.1912. A rotor with four blades is simulated. ICEM CFD is used to model rotor and stator field. Multi-domain grid scheme is used in which inner domain rotates and the outer domain is stationary. The hydrofoil mesh is done by grid stretching. Moving mesh method, which is used for unsteady conditions, is employed. Unstructured hexahedral volume mesh is created both for rotor and stator field.

Inlet Boundary Condition (BC) is applied at the left side of outer domain with uniform velocity of water V = 1.2-3.5 m/sec with turbulence option of medium intensity 5%. In the outlet section, right-hand side of the domain, the pressure is considered to be zero. Further, the outlet BC designated as opening and symmetry at the three sides. For inner rotating domain each blade is separately considered and analyzed with mass and momentum option of no slip at the wall. The no-slip condition is by default and it indicates that the fluid sticks to the wall & moves with the same velocity as the wall, as in this case, blade rotates. The rotor domain rotates with angular velocity ω. The interface is applied where the outer domain and rotating domain meet each other with discretization type General Grid Interface (GGI), GGI option will consider: if the nodes on the two sides of the connection are not aligned k-ω Shear Stress Transport (SST) turbulence model is considered, so as closed to the real scenario of turbulence field around the turbine domain. The above double precision solver chosen is based on the basic CFD codes using the pressure-velocity coupling scheme. To obtain a good discretisation of the continuity, momentum and turbulence model equations and second-order discretisation schemes is used.

Moreover, SST modeling is based on Menter (1994) which is composed of k-ω and k-ε model and also resolves the shortcoming of these two models. To accuratly model the viscous boundary-layer effect upon the wall (y+), a turbulent law of the wall parameter is used. Since the velocity next to a no-slip wall boundary change rapidly from a value of zero at the wall to the free stream value a short distance away from the wall. This layer of a high-velocity gradient is known as the boundary layer. Meshes not refine sufficiently near the wall to accurately grasp the velocity profile in boundary layer. y+ phenomenon is used so that the first node to the wall is not far or closed enough. For both the outer domain and rotating domain the y+ value are less than 8, i.e., $y+ \approx 5$-7 (ANSYS CFX Release 13.0, year and Menter, 1994). To refine the mesh grid further to obtain $y+ \leq 1$ for all cells in both domains, will ensure a mesh independent solution, but a finer mesh will increase the computational time. This is not the scope of study. The aim is to get the numerical results as quick as possible within acceptable accuracy.

Expressions are defined for torque, power and coefficient of performance in CFX Command Language (CEL) to get the output in the desired form. The timestep size is defined such that in 1sec, it takes 10 coefficient iteration loops to run and rotor domain rotates about 3°. Transient analysis is performed at different tip speed ratio i.e., λ = 0.5-5. TSR ($\lambda = \omega R/V$) where, ω is the angular velocity, R is the radius of rotor. A plot is drawn between coefficients of performance $C_P = P/(0.5\ \rho V3A)$, coefficient of torque $C_T = T/\ (0.5\ \rho V2AR)$ against the tip speed ratio λ for single turbine, in these equations P is the power, T is the torque, ρ is the density of water and A is the area of a rotor. We observed that maximum (CoP for single turbine) C_{PS} = 46.6% and C_{TS} = 22.6% are observed at λ = 2 at V = 3.5 m/sec as shown in Fig. 2. The maximum tip speed ratio achieved is based on the pitch of the rotor blades is 2 i.e., $\lambda = \tan\theta$: where, θ is the blade azimuth angle. Correspondingly the rotor inclination is 63.4° using the theory of axial flow meters (Robinson and Byrne, 2008). Moreover, Fig. 3, illustrate the coefficient of performance & torque with respect to blade position (Azimuth angle) at λ = 2.

The same simulation is run again at V = 1.2 m/sec, i.e., actual velocity of flow, for λ = 0.5-4 and for this case maximum CoP for single turbine C_{PS} = 45.2 is observed at λ = 2 and similar graphs for C_{PS} and λ is drawn as shown in Fig. 4.

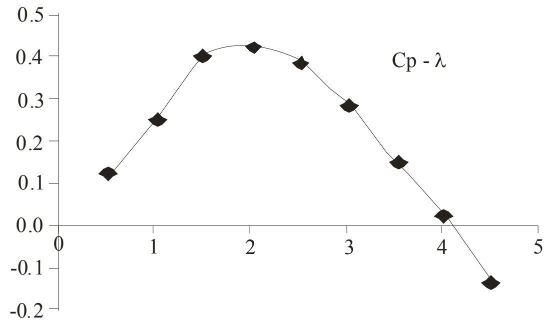

Fig. 2: Coeff. of performance vs. tip speed ratio

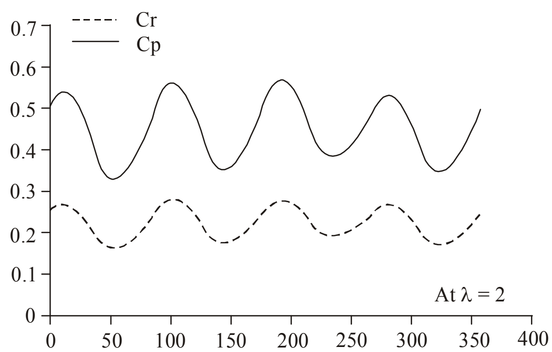

Fig. 3: Coefficient of (performance and toque) vs. azimuth angle for single turbine

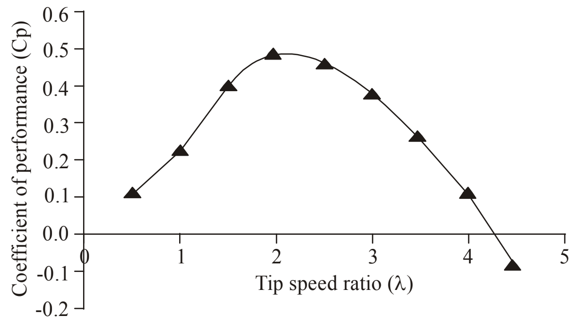

Fig. 4: Coeff. of performance vs. tip speed ratio

Modeling and simulation of twin turbine system: The inner rotating domain is the same as for the single turbine system, but the outer domain is changed. The spacing between the two rotating domain is varied from 0.5, 1, 1.5, 2, 2.5, 3 and 3.5 m, respectively. One rotor is named upper rotor and the second down rotor and the outer domain containing these two rotating domains. The upper turbine is rotating clock wise with angular velocity (ω) while, lower turbine is rotating counter clockwise with angular velocity (-ω).

BCs are the same as discussed above for the single rotating domain. Velocity at Inlet BC is defined at left side, sides and outlet BCs is defined as opening. As shown in Fig. 5. The Aim of this simulation is to find

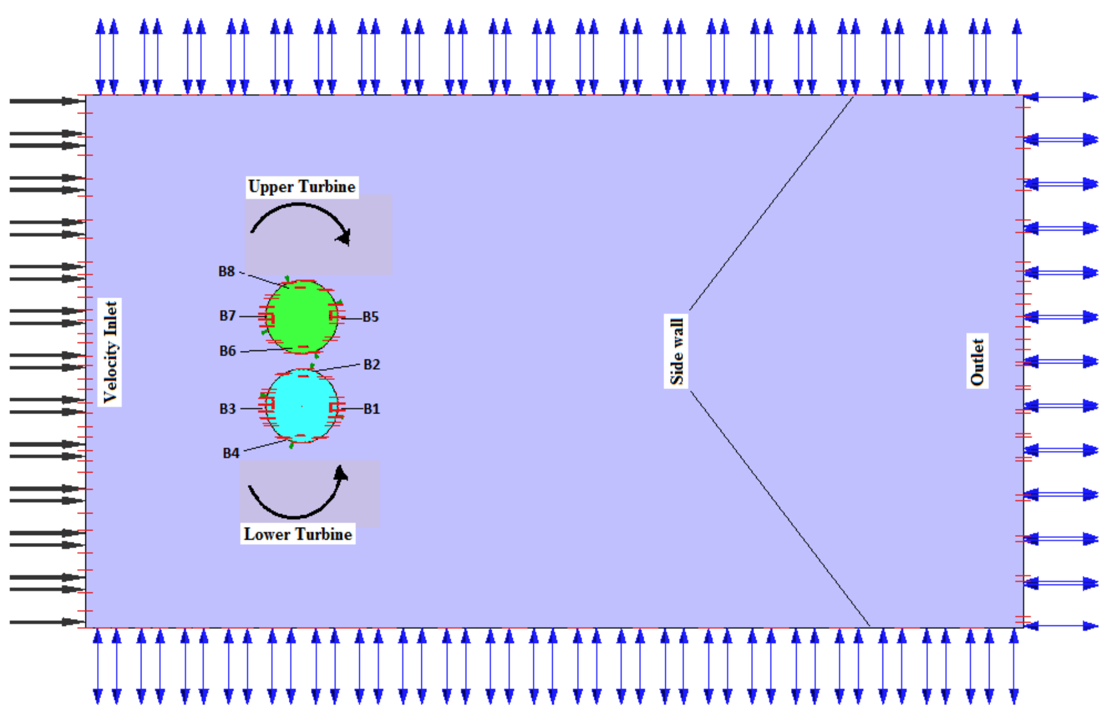

Fig. 5: Twin turbine model with BC’s

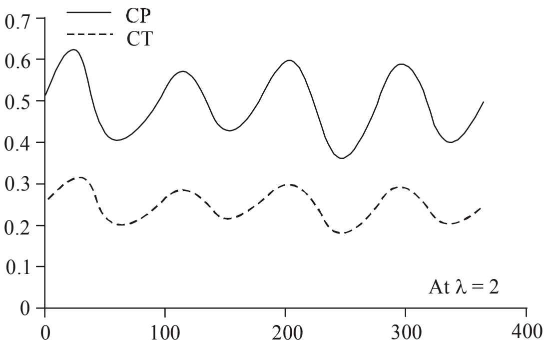

Fig. 6: Coefficient of (performance and toque) vs. azimuth angle for twin turbine

an optimal distance between two turbines so that hydrodynamic interaction and vortex shedding affect is minimum and to get maximum efficiency out of the twin system. For each case transient, simulation is run at different tip speed ratios (0.5-4) at V = 1.2 m/sec. The time steps are defined such that after 10 revolutions, the solution is converged. From Fig. 3 and 6, the difference between CoP and CoT is evidently observed and have higher values at the same conditions and same tip speed ratio. With the help of CEL expression's language, Coefficient of Performance (CoP) and torque for upper turbine (C_{PD}, C_{TD}) and for lower turbine (C_{PU}, C_{TU}) are evaluated and drawn for the tip speed ratio λ as shown in Fig. 7. The curve shows that combine CoP ($C_{PD} + C_{PU}$) of twin turbine is higher than a single turbine. From calculation, we observed that the sum of CoP of upper turbine plus lower turbine ($C_{PD} + C_{PU} = 114\%$) in a twin turbine system is greater than twice the coefficient of performance of single turbine ($C_{PS} = 2*45\%$) as shown in Fig. 8. i.e.:

$$(C_{PD} + C_{PU}) > 2C_{PS} \quad (1)$$

From calculation we observed that CoP of twin turbine system is 24% more than twice the CoP of single turbine system. These computations are done with following assumptions:

- 2-Dimentional numerical simulation modeling is done
- Connecting arm and shaft are not considered in simulation modeling
- Free surface effects and bottom effects are not considered
- Three dimensional affects are not considered

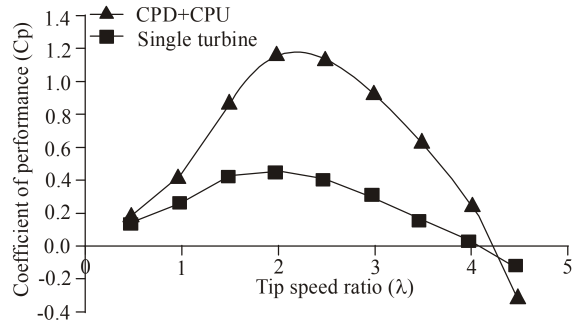

Fig. 7: Coefficient of performance of twin turbine and single turbine vs. Tip speed ratio

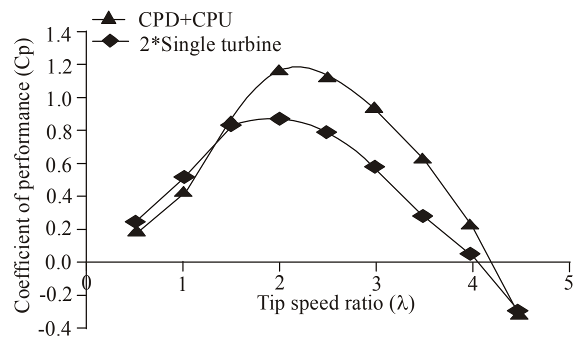

Fig. 8: CoP of twin turbine and twice of CoP of single turbine vs. Tip speed ratio

After considering the above assumptions in simulation the CoP will definitely be less than the above calculated value. After all the CoP of twin turbine system will be more than single turbine, when the distance between turbines is such that there is minimum interaction between them.

VALIDATION OF NUMERICAL SIMULATION RESULTS

Due to complex hydrodynamics of VATT, no systematic study has been done or proposed except the one by Ye and Calisal (2010a). They use DVM-UBC method to simulate the twin turbine system. DVM-UBC is based on Discrete Vortex Method (DVM), which is a potential flow method. They modify this traditional method and incorporate the viscous effects using perturbation theory. Further detail on this method is available in Ye (2007). They validated their twin turbine simulation model with experimental results. The experiment is carried on in towing tank of Canadian Research Council (Institute of Ocean Technology). The details of experimental setup and towing tank dimensions are available in Robinson and Byrne (2008). Some of the specifications of the prototype tested were: blade profile was NACA 63-021; solidity 0.435, with the Reynolds number was 160,000. The results were compared at TSR 2.5 and 2.75. They claimed that deviations of numerical simulation's results from experimental values are mostly less than 10% for the majority of cases. The simulation model specifications were: Three blade turbine with NACA 0015 profile, solidity 0.375 and Reynolds number was 160,000.

Besides this model, no other study as a twin system is available, so the present numeric simulation is validated by comparison with Li Ye model. Using Li Ye specification and using the above numerical method presented is simulated in ANSYS CFX with three blade geometry, we observed that CoP of a twin turbine system $C_{PD} + C_{PU}$ = 116% and twice of a single turbine system is 89%, so we get 27% more power from a twin turbine system. When comparing with Li Ye model the deviation is 16%, while Li Ye results deviate relatively 10% from experimental value. Therefore, we can say that numerical simulation method adopted is quite effective and CFX results are useful and in an acceptable range.

DISCUSSION AND CONCLUSION

In this study, two dimensional numerical simulation model of a twin turbine system is analyzed. The main idea behind this study is one step toward tidal turbine farm/fence modeling, taking advantage of hydrodynamic interaction between two turbines. As the VATT is lift based so the turbine can produce more power by increasing the tangential velocity component which enhanced the lift of the turbine. The lift can be improved either by reducing the turbulence vortex shedding between the turbine or by enhancing the local induce velocity on the blade. Furthermore, the hydrodynamic interaction between twin turbine systems depends upon following main parameters: relative distance between them, the incoming flow angle, the tip speed ratio λ, the relative rotating direction (contra rotating or co-rotating) and L/R ratio. Where, *L* is the distance between two turbines and *R* is the radius of turbine.

From CFX analysis, we can say that turbine with the given specification have maximum CoP when the relative distance L = 0. 5 m. This distance is between two rotating domain not from the centers of turbines. Less than or greater than this distance have produce fewer CoPs, because at smaller distance wake vortices interaction will be more dominant and have destructive effect. At this distance Maximum, CoP is observed at λ = 2. For V = 1.2 m/sec. As the distance between the turbines is increased the CoP is gradually decreased and at certain distance, they operate and behave like independent turbine system. Moreover, if the inlet velocity or tip speed ratio is increased at this distance, wake vortices will play a role because the swirling effect increases due to turbidity behind the turbine. The flow pattern for twin system at λ = 2 is shown in Fig. 9. Figure shows that velocity particle flows smoothly in-stream lines and doesn't intersect each other.

At the end we can conclude that the numerical simulation results are in agreements with the Li Ye model with certain error, but after all we achieve a certain level of satisfaction. This study supports the Li Ye finding firstly: about the twin VATT system, that the co-efficient of performance improves when the gap between turbines is suitably adjusted and we can make the interaction positive when the turbines are at a distance where velocity augmentation take place. Secondly, this study also shows that we can use ANSYS CFX software for CFD analysis, keeping in mind about the deviation and error of CFX numerical

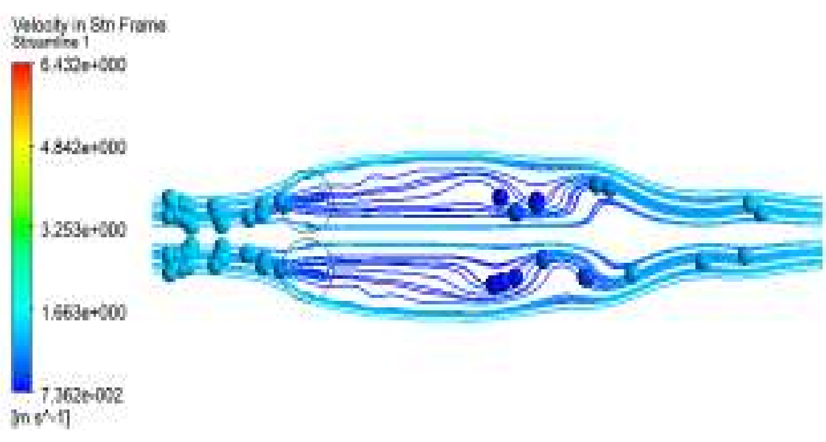

Fig. 9: Stream lines flow behavior for twin turbine

results from experiments data. Also twin system produce 24% more power than twice of single turbine at an optimal gap between two turbines. These numerical results are not generalize for all twin system but specific to the model considered.

ACKNOWLEDGMENT

This research is financially supported by National S&T Programme (grants 2007AA05Z450, 2008BAA15B04, 2008BAA15B06, 2008C14039), Zhejiang province S&T Programme (2008C14039), National Natural Science Foundation (grant no. 51106034) and '111'Project foundation from State Administration of Foreign Experts Affairs of China and MOE of China (grant B07019).

REFERENCES

ANSYS CFX Release 13.0 helps (7.7.6). Mechanical User Guide. Documentation ANSYS Ltd., Europe.

Antheaume, S., T. Maitre and C. Buvat, 2006. Abonnel. Optimization of a vertical axis water turbine farm. International Conference-from Innovation to Industry, Bremerhaven, Germany.

Batten, W.M.J., A.S. Bahaj, A.F. Molland and J.R. Chaplin, 2007. Experimentally validated numerical method for the hydrodynamic design of horizontal axis tidal turbines. J. Ocean Eng., 34(7): 1013-1020.

Charlier, R.H., 2003. A ''sleeper'' awakes: Tidal current power. J. Renew. Sustain. Energ. Rev., 7(6): 515-529.

Clarke, J., G. Connor, A. Grant and C. Johnstone, 2007. Design and testing of a contra-rotating tidal current turbine. Proceedings of the Institution of Mechanical Engineers, Part A. J. Power Energy, 221(2): 171-179.

Georgescu, A.M., S.C. Georgescu, C.I. Cosoiu, N. Alboiu and Al Hamzu, 2010. Velocity field in the wake of a hydropower farm equipped with Achard turbines. IOP Conf. Ser.: Earth Environ. Sci., 12, DOI: 10.1088/1755-1315/12/1/012108.

Menter, F.R., 1994. Two-equation eddy-viscosity turbulence models for engineering applications. AIAA-J., 32(8): 1598-1605.

Robinson, E. and G. Byrne, 2008. Methodology for modeling tidal turbine characteristics. Oceans, pp: 1-5.

Rourke, F.O., F. Boyle and A. Reynolds, 2009. Renewable energy resources and technologies applicable to Ireland. J. Renew. Sustain. Energ. Rev., 13(8): 1975-1984.

Shikha, S., T.S. Bhatti and D.P. Kothari, 2005. Early development of modern vertical and horizontal axis wind turbines: A review. J. Wind Eng., 29(3): 287-300.

Wang, L.B., L. Zhang and N.D. Zeng, 2009. A potential flow 2-D vortex panel model: Applications to vertical axis straight blade tidal turbine. J. Energ. Conver. Manage., 48(2): 454-461.

Ye, L., 2007. Note on Hydrodynamic Interaction between Two Turbines. Technical Note, Naval Architecture and Offshore Engineering Laboratory, the University of British Columbia.

Ye, L. and S.M. Calisal, 2010a. Numerical analysis of the characteristics of a vertical axis water current turbine. J. Renew. Energ., 35: 435-442.

Ye, L. and S.M. Calisal, 2010b. Modeling of twin-turbine systems with vertical axis tidal current turbines: Part I-Power output. J. Ocean Eng., 37(7): 627-637.