HYDRODYNAMIC DESIGN AND ANALYSIS OF MARINE CURRENT TURBINE BLADES

Ching-Yeh Hsin, Shih-Yun Wang, Chiung-Chih Lin and Jiahn-Horng Chen
Department of Naval Architecture and Systems Engineering,
National Taiwan Ocean University, Keelung, Taiwan

Long Abstract

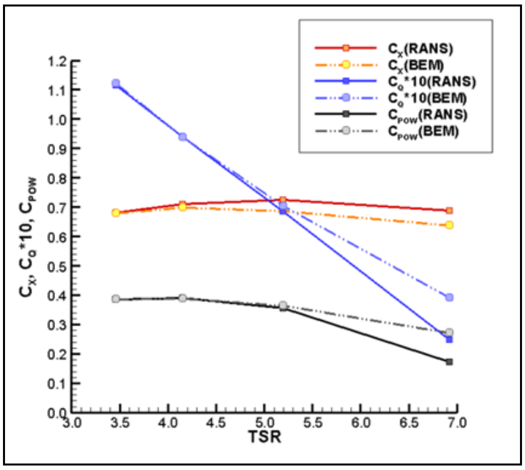
Introduction

In this paper, two turbine blade design procedure are presented, and the design examples are demonstrated. One design procedure is similar to the propeller designs, a lifting line method ([1]~[4]) for the current turbine is first developed to obtain the optimum circulation distribution, and the blade geometry is then designed by the lifting surface vortex lattice method modified from the propeller lifting surface design method ([5]~[7]). Finally the potential flow boundary element method is used to verify the force computations [8]. The other design procedure is to use Genetic Algorithm to adjust the pitch angle and camber distributions of the blade, and the objective is to find a geometry which can provide the maximum torque and the minimum thrust. After completing the designs, both the boundary element method and the viscous flow RANS method ([9]~[10]) will be applied to the analysis of the performances of the current turbines for the final check of the designs. The self-developed, perturbation potential based boundary element method is the potential flow method we have used, and the commercial software STAR- CCM+ is used for the viscous flow computations..

Analysis Methods

Two computational methods are used for the current blade performance analysis, and they are the potential flow boundary element method (BEM) and viscous flow RANS method. The potential flow boundary element method used for computing the horizontal axis current turbines is a perturbation potential based method, and a wake alignment numerical scheme is established for the current turbine. Though the flow around the turbine is assumed to be steady, inviscid, irrotational, and incompressible, viscous corrections are used for predicting the forces. The turbine performances including the torques, axial forces and powers can be obtained from the computations, so are the pressure distributions and the circulation distributions along the span-wise direction. The commercial software STAR CCM+ is used for the viscous flow computations. It solves the RANS equations by a finite volume method, and here the realizable two-layer turbulence model is adopted in the computations. Approximately 4 million polyhedral elements are used for the computations, and several layers of meshes are arranged near the blade region. A very dense mesh is adopted at tip of the blades in order to capture the phenomenon of the complex tip flow.

Comparisons of the predicted axial force coefficients, torque coefficients and power coefficients at different TSR's for two different methods are shown in Figure 1. The comparisons show that both methods predict similar results at low TSR's. In general, the torque coefficients decrease with increasing TSR, on the other hand, the change of the axial force coefficients is not significant



$$TSR = \frac{2\pi Rn}{V_\infty}$$

$$C_x = \frac{F_x}{\frac{1}{2}\rho A V_\infty^2}$$

$$C_Q = \frac{Q}{\frac{1}{2}\rho A V_\infty^2 R}$$

$$C_{P_w} = \frac{P_w}{\frac{1}{2}\rho A V_\infty^3}$$

Figure 1. The predicted current turbine performances from BEM and RANS methods

Design Methods

The first design procedure is established based on the propeller design method. That is, the lifting line method is used for the design of the circulation distribution, and the lifting surface method is then adopted for the blade geometry design. An optimization method using the Lagrange Multiplier method is developed with the lifting line method, and the optimum circulation can be obtained by this method. For the current turbine blade designs, the constrained optimization problem is to find a circulation distribution which provides the best power coefficient by assuming that both the torque coefficient and the axial force coefficient are functions of the circulation distributions.

The other design procedure is to use Genetic Algorithm method, and our objective is to design a new geometry of the current turbine that has a bigger torque coefficient and smaller axial force coefficient than the one designed by the Lagrange Multiplier method. In this case, we considered the pitch angle and camber distributions as the gene in the individual. We then designed a procedure that uses Genetic Algorithm to adjust the pitch angle and camber distributions to reach our objective (bigger torque and smaller axial force). Table 1 shows the design results, and Figure 2 shows the pressure distributions of the geometries designed by Genetic Algorithm and Lagrange Multiplier method. Although the Genetic Algorithm method hasn't obtained blade geometries with better performance comparing to the Lagrange Multiplier method, it has a larger flexibility which can design other geometry parameters, such as chord length distribution.

	Lagrange Method	GA Method
C_X	0.7436	0.7403
C_Q	0.0754	0.0757

Table 1. Comparisons of the calculated axial force coefficients and torque coefficients using two design methods

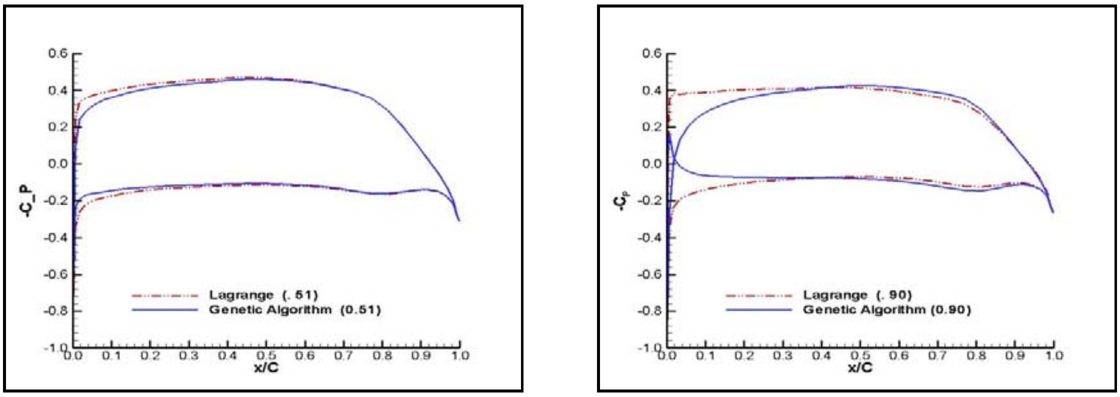


Figure 2. The pressure distributions of geometries designed by two design methods computed by the BEM at r/R=0.5 and 0.9

References

[1] S. Goldstein, "On the vortex theory of screw propellers", Proc. R. Soc. London Ser. A 123 :440-65.

[2] H.W. Lerbs, "Moderately loaded propellers with a finite number of blades and an arbitrary distribution of circulation", SNAME Trans. Vol. 60, 1952.

[3] C.-Y. Hsin, "Efficient Computational Methods for Multi-Component Lifting Line Calculations", Master Thesis, MIT, Dept. of Ocean Engineering, 1986

[4] J.E. Kerwin, J.E., W.B. Coney, W.B., and C.-Y. Hsin, "Optimum Circulation Distributions for Single and Multi-Component Propulsors", ATTC conference, 1986.

[5] P.C. Pien, "The calculation of marine propellers based on lifting surface theory", J. of Ship Research, Vol.5, No. 2

[6] J.E. Kerwin, "The solution of propeller lifting surface problems by vortex lattice methods", report, Dept. of Ocean Eng., M.I.T.

[7] D.S. Greeley, and J.E. Kerwin, "Numerical methods for propeller design and analysis in steady flow", SNAME Trans. Vol. 90, 1982

[8] Y.-S. Luo "Investigation of the Current Turbine Performance by Computational Methods", Master thesis, National Taiwan Ocean University, 2013

[9] C.-Y. Hsin, C. Ko, A.-C. Kuo and Y.-L. Tsai "Applying Different Computational Methods to the Simulation of Flow Field around a Horizontal Axis Marine Current Turbine", The 12th International Conference on Fluid Control, Measurements and Visualization (FLUCOME 2013), Nara, Japan.

[10] C.-Y. Hsin, Y.-S. Luo and C.-C. Lin "Applying the Boundary Element Method to the Analysis of Flow Around Horizontal Axis Marine Current Turbines" Proceedings of the Eighth International Workshop on Ship Hydrodynamics, September, 2013.