Hydrodynamic Analysis and Design of Marine Current Turbine Blades

Ching-Yeh Hsin\textsuperscript{1}* , Shih-Yun Wang\textsuperscript{1}* , Chiung-Chih Lin\textsuperscript{1}* , Jiahn-Horng Chen\textsuperscript{1}* and Forng-Chen Chiu\textsuperscript{2}* 

\begin{abstract}
In this paper, two design procedures of the marine current turbine blade geometries are presented. The first one is similar to the propeller design method, the lifting line method is used for the design of the optimum circulation distribution, and the lifting surface method is then adopted for the blade geometry design. The second design procedure uses Genetic Algorithm method to design the turbine blade geometry. Hydrodynamic performances of the marine current turbine are then computed and analyzed by the potential flow boundary element method and the viscous flow RANS method. Two design examples, including a 20kw floating type current turbine, are demonstrated in the paper, and the design results show the geometries designed by both methods satisfy the design goal; however, the geometry designed by Genetic Algorithm method has a better result. It is believed that both methods are applicable for the current turbine blade designs.
\end{abstract}

\textbf{Keywords:} Renewable energy — Current Turbine — Turbine Blade Design — Boundary Element Method — RANS

\textsuperscript{1}Department of Naval Architecture and Systems Engineering, National Taiwan Ocean University, Keelung, Taiwan
\textsuperscript{2}Department of Engineering Science and Ocean Engineering, National Taiwan University
\textsuperscript{*}Corresponding author: hsin@mail.ntou.edu.tw

\section*{INTRODUCTION}

When extracting the ocean current energy, two most common devices are the horizontal axis current turbine and the vertical axis current turbine. For the horizontal axis current turbine design, most people use the methods for wind turbine blade designs. However, the current turbines operate in the water, and their physical behaviors are more like marine propellers. In this paper, two turbine blade design procedures are presented, and the design cases 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 confirm the satisfaction of designed forces [8]. The other design procedure is to use Genetic Algorithm [9] 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 ([10]&[11]) will be applied to the analysis of the performances of the current turbines for the final check of the designs.

\section*{1. 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 commercial software STAR CCM+ is used for the viscous flow computations.

The Boundary element Method (BEM) used in this paper is a perturbation potential based boundary element method, and the governing equation is:

\begin{equation}
2\pi\phi(p) = \int_{S_{B}} (\phi(q) \frac{\partial}{\partial n_{q}} r(p:q) - \frac{1}{r(p:q)} \frac{\partial \phi(q)}{\partial n_{q}}) dS + \int_{S_{W}} \Delta \phi(q) \frac{\partial}{\partial n_{q}} \frac{1}{r(p:q)} dS
\end{equation}

In equation (1), $S_{B}$ denotes the body surface, and $S_{W}$ denotes the wake surface. $\phi$ is the strength of perturbation potentials, or equivalent to the dipole strength, and $\partial \phi / \partial n$ is the source strength. $r(p:q)$ is the distance between the panel point $q$ and the induced point $p$. The term $1 / r$ is the potential induced by a unit strength source, and $\partial (1 / r) / \partial n$ is the potential induced by a unit strength dipole. $\Delta \phi$ is the dipole...
strength in the wake from the Kutta condition, and the source strength in the wake is zero since the wake has no thickness. The discretized form of the equation (1) is:

$$
\sum_{j=1}^{N_p} a_{i,j} \phi_j = \sum_{j=1}^{N_p} b_{i,j} \frac{\partial \phi_j}{\partial n} - \sum_{m=1}^{M} \sum_{j=1}^{N_p} W_{i,m,j} \Delta \phi_{m,j} \quad i = 1, N_p
$$

(2)

In equation (2), $\phi_j$ and $\sigma_j$ represent the discrete forms of $\phi$ and $\partial \phi / \partial n$, and $a_{i,j}$, $b_{i,j}$ represent the discrete forms of the integrations of $\partial (1/r) / \partial n$ and $1/r$ over a panel. $w$ represents the discrete forms of the integration of $\partial (1/r) / \partial n$ over a wake panel. A wake alignment scheme is used in this boundary element method to correct the turbine blade wake geometry based on the induced velocities downstream, and a simple viscous correction is used for including the viscous effect. The turbine performances including the axial forces, torques, powers, pressure distributions and the circulation distributions can be obtained from the computations, and so are the pressure distributions and the circulation distributions.

The commercial software STAR CCM+ is used for the viscous flow computations. It solves the RANS equations by a finite volume method. The computational domain and boundary conditions are shown in Figure 1, and the grid topology is shown in Figure 2. Approximately 4 million polyhedral grids are normally used for the computations. A very dense mesh is adopted at tip of the blades in order to capture the phenomenon of the complex tip flow. Three turbulence models have been used for investigation, and they are “Realizable k-ε two layer”, “Standard k-ε” and “k-ω SST”. The computational results show that using three different turbulence models are equally good, and the STAR-CCM+ default turbulence model “Realizable k-ε two layer” has been chosen for subsequent computations.

We will first verify two computational methods. For both the BEM and RANS computations, the computational conditions and the geometric specification for the verification case are listed in Table 1, and the computer depiction of this marine current turbine is shown in Figure 3. For this case, the inflow is assumed to be axis-symmetrical, and the flow is assumed to be steady. Four different inflow velocities: 1.5, 2.0, 2.5, 3.0 m/s are used to investigate the performance of the marine current turbine at different tip speed ratios (TSR).

<table>
<thead>
<tr>
<th>Foil Geometry</th>
<th>NACA66/a=0.8meanline</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of blade</td>
<td>3</td>
</tr>
<tr>
<td>Radius(m)</td>
<td>0.4</td>
</tr>
<tr>
<td>Rotational Speed(rps)</td>
<td>4.13</td>
</tr>
<tr>
<td>Inflow velocities(m/s)</td>
<td>1.50, 2.00, 2.50, 3.00</td>
</tr>
<tr>
<td>Tip Speed Ratio(TSR)</td>
<td>6.92, 5.19, 4.15, 3.46</td>
</tr>
</tbody>
</table>
To verify the computational results of different methods, we first introduce several important coefficients related to the marine current turbine performance. These coefficients are the tip speed ratio (TSR), axial force coefficient, torque coefficient and power coefficient. They are defined as follows:

\[
\text{TSR} = \frac{2\pi R n}{V_\infty} \tag{3}
\]

In equation (3), \(R\) is the radius of turbine in meter, \(n\) is the rotational speed of turbine in rps, and \(V_\infty\) is the inflow velocity in m/s.

\[
C_a = \frac{F_x}{\frac{1}{2} \rho AV_\infty^2}, \quad C_Q = \frac{Q}{\frac{1}{2} \rho AV_\infty^2 R}
\]

\[
C_p = \frac{P_w}{\frac{1}{2} \rho AV_\infty^2} = \text{TSR} \times C_Q \tag{4}
\]

In equation (4), \(F_x\) is the axial force working on turbine in \textit{Newton}, \(Q\) is the torque working on turbine in \textit{Newton-meter}, \(P_w\) is the power of current turbine, and \(P_w = 2\pi n Q\).

The case we use for verification is the designed turbine blade later described in section 3.1. The computational axial force coefficients, torque coefficient and power coefficients are shown in Figure 4. The comparisons show that both methods predict similar results near the design TSR (5.19). Torque coefficient decreases as the value of tip speed ratio increases. However, the axial force coefficients for different tip speed ratios do not vary much. Figure 5 shows the comparisons of pressure distributions computed by the BEM and RANS methods at TSR=5.19, and we can see that the pressure distributions from BEM and RANS are different near the leading edge and trailing edge. These differences may be due to the viscous effects, and thus result in the discrepancies in the computational forces shown in Figure 4. In general, the discrepancies of the computational results between the BEM and RANS are acceptable, and the trends predicted by two methods are the same.

We will then validate two computational methods by the experimental data. The experiment conducted by Bahaj et. Al. [12] is used here. In their experiment, a three-blade turbine with 0.8 meter diameter is tested at the inflow speed of 1.73m/s. Figure 6 and Figure 7 show the comparisons of computational results from both BEM and RANS methods and the experimental data for both the axial force and power coefficients. From two figures, we can see that the RANS method shows a better prediction for the axial force, and both methods have similar accuracies for the power prediction. The errors are around 5% to 10%, and the trends are predicted correctly. From the verification and validation of two computational methods, we think both methods are reliable with acceptable accuracies.

![Figure 4. Force and power coefficients computed by two different computational methods](image1)

![Figure 5. The Pressure distributions computed by different computational methods at r/R=0.5 and r/R=0.7 for TSR=5.19](image2)
2. DESIGN PROCEDURES

2.1 The Lagrange Multiplier Method

The first design approach is established based on the propeller design method. That is, the lifting line method is first used to obtain the optimum circulation distribution, or, the loading distribution. In the lifting line method, we can obtain the forces of the current turbine blade by Kutta-Joukowski Law:

\[ \mathbf{F} = \rho \mathbf{U} \times \mathbf{\Gamma} \]  

(5)

In equation (5), \( \mathbf{U} \) is the resultant inflow, and \( \mathbf{\Gamma} \) is the circulation distribution. Therefore, we can obtain the following equation:

\[
\begin{bmatrix}
\hat{e}_a \\
\hat{e}_t \\
\end{bmatrix} = \begin{bmatrix}
\hat{\omega}_a \\
\hat{\omega}_t \\
\end{bmatrix} + \begin{bmatrix}
\hat{\omega}_r \\
\hat{\omega}_i \\
\end{bmatrix} \\
\begin{bmatrix}
\mathbf{0} \\
\mathbf{0} \\
\end{bmatrix} + \mathbf{\Gamma} \\
\mathbf{\Gamma}
\end{bmatrix}
\]

\[
= \mathbf{F}_a \hat{\omega}_a + \mathbf{F}_t \hat{\omega}_t
\]

\[\mathbf{U} \times \mathbf{\Gamma} = \begin{bmatrix}
\hat{\omega}_a \\
\hat{\omega}_t \\
\end{bmatrix} + \begin{bmatrix}
\hat{\omega}_r \\
\hat{\omega}_i \\
\end{bmatrix} - \begin{bmatrix}
\mathbf{V}_a \times \mathbf{r} \\
\mathbf{V}_r \times \mathbf{r} \\
\end{bmatrix} \mathbf{\Gamma}
\]

\[= \begin{bmatrix}
\mathbf{0} \\
\mathbf{0} \\
\end{bmatrix} + \mathbf{\Gamma}
\]

Where \( \omega \) is the rotational speed, \( V_a \) and \( V_r \) are axial and tangential inflow velocities, and \( u_a^* \) and \( u_t^* \) are axial and tangential induced velocities. \( F_a \) and \( F_t \) are axial and tangential forces which contribute as thrust and torque. Figure 8 shows this force-velocity diagram. Since the induced velocities depend on the magnitude of the circulation, we can express the forces as functions of discrete circulation distributions. We can then optimize the circulation distribution to obtain the best power coefficient.

The Lagrange Multiplier Method is used to obtain the optimum circulation distribution. The Lagrange Multiplier Method can transfer a constrained problem to a non-constrained problem by introducing the Lagrange multiplier \( \lambda \). For the current turbine blade design, the constrained optimization problem is to find a circulation distribution which provides the best power coefficient \( (C_{PW}) \), that is, the minimum axial force coefficient \( (C_X) \) with a given torque coefficient \( (C_Q) \). Therefore, the constraint is \( C_Q = C_Q^* \), and \( C_Q^* \) is the objective torque coefficient. The design problem thus can be stated as:

\[
\begin{cases}
\min \quad C_X \\
\text{subject to} \quad C_Q - C_Q^* = 0
\end{cases}
\]  

(7)

As described earlier, the torque coefficient and the axial force coefficient are functions of circulation distribution:
\begin{align}
C_c &= C_c(\Gamma) \\
C_0 &= C_0(\Gamma) \\
\Gamma &= [\Gamma_1, \Gamma_2, \ldots, \Gamma_N] 
\end{align} \tag{8}

We thus can define the Lagrangian of this optimization problem as:

\[ L(\Gamma, \lambda) = C_x + \lambda (C_0 - C_0^*) \] \tag{9}

To get the minimum value of \( C_x \), we take the gradient of equation (9), and \( \nabla L \) can be expressed as:

\[ \nabla L = \nabla C_x + \lambda \nabla C_0 = 0 \\
C_0 - C_0^* = 0 \] \tag{10}

We can further derive to obtain:

\[ \frac{\partial C_x}{\partial \Gamma_i} + \lambda \frac{\partial C_0}{\partial \Gamma_i} = 0, \quad \text{for } i = 1, \ldots, M \\
C_0 - C_0^* = 0 \] \tag{11}

The optimum circulation distribution thus can be obtained by solving the equation, \( G(X) = 0 \), and

\[ G = \begin{bmatrix} \nabla L \\ C_0 - C_0^* \end{bmatrix}, \quad X = [\Gamma] \] \tag{12}

Once the optimum circulation distribution is obtained, the blade geometry, namely, the pitch and the camber distributions can be obtained by the lifting surface vortex lattice method by distributing the radial circulation distribution to the chord-wise direction. The potential flow boundary element method is then used to confirm the satisfaction of designed forces.

### 2.2 Genetic Algorithm Method

The second design procedure we used is based on the Genetic Algorithm method. This optimization method was inspired by Darwin’s theory of evolution. The individual whose gene fit the environment better will have better potential for survival. After eliminating unsuitable individual for several generations, the remaining individuals indicate they might be the answer that we are looking for. Our objective is to design a new geometry of the current turbine that has a bigger \( C_Q \) and smaller \( C_T \) than the current turbine designed by the first method described in section 2.1. In this procedure, the optimization method is the Genetic Algorithm, and the computations are carried out by the BEM since it is much more efficient than the RANS method. We considered the pitch angle distributions as the gene in the individual. We designed a procedure that uses Genetic Algorithm to adjust the gene (pitch angle) to reach our objective (bigger torque and smaller axial force). The following is the entire process of the method:

1. Generate the first generation individual randomly.
2. Calculate the objective function of each individual. Objective function \( (F_i) \) is a score we design to describe how close this individual is to our objective. It takes several times testing to find a proper objective function.
3. Calculate the fitness function of each individual.
4. Do crossover and mutation to generate the individual of the next generation.
5. Repeat step 3, 4 until the amount of the individual of the next generation is the same as the amount of the first generation.
6. Repeat step 2, 5 until there is an individual reach the objective we decided or reach the generation number we decided.

The designs from this procedure will finally be double checked by the RANS method.

### 3. DESIGN RESULTS

Two design cases are shown in this paper, and the first one is to redesign the current turbine shown in Table 1, and the second case is a 20kW current turbine.

#### 3.1 Design Case 1

For the first design case, the design point is at inflow speed 2.0m/s, and TSR=5.19 as in Table 1, and the designed \( C_Q=0.0754 \). Two different designs are listed as follows:

- The first one is that we use the current turbine lifting line method with the Lagrange multiplier method to obtain the “optimum circulation distribution”, and use the current turbine lifting surface method to design the pitch and camber distributions. The first design is named as “original design”.
- Based on the same required torque, the circulation
distribution is then adjusted to tip unloading from the optimum circulation distribution. The current turbine lifting surface method is then adopted to design the pitch and camber distributions from the adjusted circulation distribution. This design is named as "new design".

The circulation distributions of these two designed turbines are shown in Figure 9, and one can see that the circulation distribution of the "new design" is tip unloaded, that is, less loading near the tip. Figure 10 shows the pitch and camber distributions of two designs, and it is interesting to see that the pitch distributions of two turbines are quite close; however, the camber distributions show that the "new design" is a tip unloaded current turbine since it has a relatively low camber values near the tip.

The performances of these two turbines computed by the BEM method are shown in Table 2, and the results show that both the torque and thrust are very close for two designs. In order to investigate the differences of two designs, we then look into the pressure distributions of two designs. Figures 11a and 11b show the pressure distributions computed by BEM of two designs. We can see that the "new design" has a higher loading at mid-chord near the hub (r/R=0.30), and this is due to the tip unloading. The pressure distributions of two designs are very close at r/R=0.51, and this reflects the circulation distributions in Figure 9. The "new design" shows lower loadings at mid-chord near the tip (r/R=0.81 and 0.90), and it also shows a flatter pressure distributions near the leading edge near tip. Therefore, from the pressure distributions point of view, "new design" seems to be a better design.

Table 2. The performances of two designs computed by the BEM method

<table>
<thead>
<tr>
<th></th>
<th>TSR=5.19</th>
<th>Cₓ</th>
<th>Cᵧ</th>
<th>Cₚₓ</th>
</tr>
</thead>
<tbody>
<tr>
<td>&quot;Original Design&quot;</td>
<td>0.7464</td>
<td>0.07554</td>
<td>0.39205</td>
<td></td>
</tr>
<tr>
<td>&quot;New Design&quot;</td>
<td>0.7436</td>
<td>0.07538</td>
<td>0.39122</td>
<td></td>
</tr>
</tbody>
</table>

Figure 9. The circulation distributions of two designs

Figure 10. The pitch distributions (upper) and camber distributions (lower) of two designs

Figure 11a. The pressure distributions at radii r/R=0.30 and r/R=0.51 of two designed turbines
The GA method is then used to design this turbine, and comparisons of the calculated axial force coefficients and torque coefficients using two design methods are shown in Table 3. The comparisons show that both Lagrange method and Genetic Algorithm methods have similar design results.

In Table 3, GA(2) is to decrease the geometry smooth factor, and adjust objective function from 50% to 20%. Table 3 shows the Comparisons of three design results, and it shows that GA (2) performance is better than the designs from both GA (1) and Lagrange method. We then remove GA (1) design result since its performance is not as good as GA (2). Table 4 is the comparison of the design results from Lagrange method and Genetic Algorithm method. The computational force coefficients shown in the table are from two computational methods. The comparisons show that BEM method and RANS method predict similar trends. However, the RANS method gives a larger forces and power.

We then observed the pressure distributions of the geometry designed by two different design procedures as in Figure 12. Since the Lagrange Multiplier method use the lifting line and lifting surface methods for the geometry designs, we can see that its design results have more physical considerations. For example, the pressure distributions in Figure 12 show that the pitch angles of the Lagrange Multiplier method are designed at angles of attack closer to the ideal angles of attack than those from the Genetic Algorithm method.

### Table 3. The axial force coefficients $C_X$ and torque coefficients $C_Q$ computed by the BEM method for the designed geometries at the inflow velocity $V=2\text{m/s}$

<table>
<thead>
<tr>
<th>Method</th>
<th>$C_X$</th>
<th>$C_Q$</th>
</tr>
</thead>
<tbody>
<tr>
<td>BEM</td>
<td>0.7436</td>
<td>0.0754</td>
</tr>
<tr>
<td>GA(1)</td>
<td>0.7403</td>
<td>0.0757</td>
</tr>
<tr>
<td>GA (2)</td>
<td>0.6923</td>
<td>0.0756</td>
</tr>
</tbody>
</table>

### Table 4. The forces and power coefficients computed by the BEM and the RANS methods and the differences between two design methods

<table>
<thead>
<tr>
<th>Method</th>
<th>$C_X$</th>
<th>$C_Q$</th>
<th>$C_{PW}$</th>
<th>Differences</th>
</tr>
</thead>
<tbody>
<tr>
<td>BEM</td>
<td>0.7436</td>
<td>0.0754</td>
<td>0.3913</td>
<td>-6.899%</td>
</tr>
<tr>
<td>RANS</td>
<td>0.6923</td>
<td>0.0756</td>
<td>0.3924</td>
<td>-3.969%</td>
</tr>
<tr>
<td>GA (2)</td>
<td>0.7509</td>
<td>0.0768</td>
<td>0.3986</td>
<td>+0.265%</td>
</tr>
<tr>
<td>GA (2)</td>
<td>0.7211</td>
<td>0.0779</td>
<td>0.4043</td>
<td>+1.432%</td>
</tr>
</tbody>
</table>

**Figure 11b.** The pressure distributions at radii $r/R=0.81$ and $r/R=0.90$ of two designed turbines

**Figure 12.** The pressure distributions of the geometries designed by two design methods at $r/R=0.5$ and 0.9
Table 5. The geometric parameters of the 20kw floating type current turbine

<table>
<thead>
<tr>
<th>Foil Geometry</th>
<th>NACA66/a=0.8meanline</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of blade</td>
<td>3</td>
</tr>
<tr>
<td>Radius(m)</td>
<td>5.0</td>
</tr>
<tr>
<td>Rotational Speed(rps)</td>
<td>0.5</td>
</tr>
<tr>
<td>Inflow velocities(m/s)</td>
<td>1.50</td>
</tr>
</tbody>
</table>

Table 6. The force and power coefficients of the designed 20kw floating type current turbine

<table>
<thead>
<tr>
<th></th>
<th>$C_x$</th>
<th>$C_Q^{*}10$</th>
<th>$C_w$</th>
</tr>
</thead>
<tbody>
<tr>
<td>BEM</td>
<td>0.7194</td>
<td>0.08073</td>
<td>0.4227</td>
</tr>
<tr>
<td>RANS</td>
<td>0.7700</td>
<td>0.08680</td>
<td>0.4545</td>
</tr>
</tbody>
</table>

3.2 Design Case 2

The second design case demonstrated here is a 20kw floating type current turbine. There are two sets of turbines for this 20kw current turbine, and each turbine should provide at least 10kw power. The geometric parameters are shown in Table 5, and the design torque is to have at least 4070 N·m ($C_Q=0.07388$). For this design case, we are going to set the design goal to be the maximum power since we can have larger axial forces for the floating type current turbine. Therefore, the first procedure will be used for an initial design, and the GA method will then be used for the final designs.

Table 6 shows the force and power coefficients of the designed geometry computed by the BEM and RANS methods. We can see that the torque coefficients are larger than the required one which means this design fulfills the design goal. The BEM is the computational method used in the GA method, and the RANS method is the method to double check the design. Since the predicted power by RANS method is larger than the BEM, it means that both computational methods predict this design reaches the design goal. Figure 13 shows the performance curves computed by RANS method, and Figure 14 shows the power curve computed by the RANS method. In Figure 14, the line with circles is the rated power of the electricity generator at different inflow speeds. Both Figures 13 and 14 are important in the operation of the current turbine.

4. CONCLUSIONS

We have developed two different design procedures based on the Lagrange Multiplier method and Genetic Algorithm method respectively. Two design cases, including a 20kw floating type current turbine, are demonstrated in the paper, and the design results show the geometries designed by both methods satisfy the design goal. For the analysis, two Computational methods, RANS and BEM methods, are used to predict the performances of the current turbines, and both methods predict the similar trend. It is believed that both design procedures presented in this paper are applicable, and they can be used for the current turbine blade designs in the future. To continue this research, the blade element momentum method commonly used for the wind turbine analysis will be developed to confirm two computational methods presented in this paper. Also, the current turbine performance in the unsteady inflows will be computed and investigated.

ACKNOWLEDGMENTS

This research project is funded by the Ministry of Science and Technology under National Energy Program, Phase II, Taiwan, R.O.C. The authors would like to thank for their support.
REFERENCES


[12] Bahaj, A.S., Molland, A.F., Chaplin, J.R., Batten, W.M.J., “Power and thrust measurements of marine current turbines under various hydrodynamic flow conditions in a cavitation tunnel and a towing tank”, University of Southampton, UK, 2006