Effect of buoyancy force on buoyancy waterwheel efficiency using numerical flow simulation

Fadhlurrahman Zaki, Dan Mugisidi*

* Mechanical Engineering, Faculty of Industrial Technology and Information Technology, Muhammadiyah University Prof. Dr. Hamka, Indonesia, 13830. Jl. Tanah Merdeka No.6, RT.10/RW.5 13830 Region of Jakarta, Indonesia
* dan.mugisidi@uhamka.ac.id
Submitted: 30/12/2023 Revised: 15/01/2024 Accepted: 07/02/2024

ABSTRACT

In this work, the performance of a buoyant waterwheel to produce hydrokinetic power is investigated through analytical theory and computational fluid dynamics simulation. The impact of the buoyancy wheel is investigated by establishing the performance parameters through the use of a moving mesh approach and a realizable k-ε turbulence model. Transient simulation is required to comprehend the flow of physical processes. Using moving mesh as a transient methodology of the buoyancy waterwheel, numerical flow simulations and theoretical analytical methods are used in this study to assess the effect of buoyant force generated on the performance of the buoyancy wheel. The buoyancy waterwheel that will be put to the test has eight straight blades and a diameter of one meter. The pinwheel force and torque created in the numerical flow simulation (CFD) are 414.96 N and 207.48 Nm, respectively, whereas in the theoretical calculation, they are 449.06 N and 224.53 Nm, according to the research findings. It is possible to compute the buoyancy wheel's power output and efficiency mathematically, yielding values of 1619.35 W and 68.07%. The buoyancy wheel's power output and efficiency, as determined by numerical flow simulation, are 1495.95 W and 62.88%, respectively. Based on theoretical and CFD study results, the buoyancy wheel generates a standard deviation of 7.62%. Thus, for the buoyancy wheel, a temporary method that takes advantage of the moving mesh characteristic is advised. This method can also be applied as a future alternative energy source for the Piko hydro turbine.

Keywords: Renewable energy; waterwheel; CFD; moving mesh; hydropower

1. INTRODUCTION

Year after year, Indonesia’s requirement for electrical energy keeps growing [1]. Because of the depletion of non-renewable natural resources, research is being done on the potential of renewable energy to provide electricity [2],[3]. Indonesia create renewable energy sources like hydro energy because it is a tropical nation [4],[5], wind energy [6]–[10], geothermal energy [11], biogas energy [12], and biomass energy [13]. Hydropower is regarded as a clean, sustainable energy source with enormous growth potential [14]. By 2025, the Indonesian government wants to use 23% renewable energy, particularly in the country’s outlying regions [15]. Pico hydro is the term for water energy, which falls under the lowest size category and has a maximum production of 5 kW [16]. The process of operating a pico hydro water turbine involves using the turbine's rotation to generate mechanical energy [17]. An electric generator uses a revolving turbine to transform mechanical energy into electrical energy. Numerous varieties of hydraulic turbines exist: Reaction turbines (like Francis and Kaplan) employ water pressure, impulse turbines (like Pelton, Turgo, and Banki) use water kinetic energy, and gravity turbines (like water wheels and Archimedes screws) rely mostly on water weight [18],[19].
A great deal of research has been done to enhance turbine performance to solve this issue. William and Samson believe that a pico hydro turbine would be a wise decision [20], the feasibility analysis of self-sufficient electricity production for outlying areas. The effectiveness of waterwheels in connection to the ratio of immersion radius to paddle count was examined by Tevata and Inprasit. Their research indicates a significant relationship between the torque load of a waterwheel running at maximum power and the dipping radius ratio [21]. As stated by Quaranta [22]–[24], The efficacy of the breastshot waterwheel can be increased by numerically and experimentally altering the overshot waterwheel's geometry with a circular wall encircling the wheel. Warjito also examined the quantity of distinct buckets to evaluate the flow field's performance and quality. The efficiency of water wheels is expected to be significantly impacted by the design of the radial blades [25]. Next, Adanta assessed Warjito's proposal [25]. Each wheel's moment of inertia is determined using the CFD approach with 6-DoF characteristics. Thus, 20 blades get the best results, but 8 blades yield the most consistent results [26].

In the realm of renewable energy, the combination of buoyancy and water wheel is particularly promising for hydroelectric power plants [27]. In fluid mechanics, the upward force a fluid exerts on a submerged object is known as the buoyant force [28]. Archimedes' principle states that this force is equal to the weight of the fluid displaced by the submerged item [29]. This idea is particularly clear in fluids like water, where items float or have a reduced effective weight because buoyant force opposes gravity [30]. A submerged object's buoyant force is produced by the pressure differential between its top and bottom, with higher pressure at the bottom providing an upward push [31]. This means that new avenues for optimizing energy extraction from dynamic water settings are made possible by the strategic interaction of buoyancy forces with water wheels [32].

Compared to experimental and analytical techniques, CFD methods can depict the flow field in greater detail [33]. Moving Mesh was selected for the temporary method since it uses less processing resources and is frequently employed by researchers [34]–[36]. Using moving mesh as a transient methodology for buoyancy wheels, computational fluid dynamics (CFD) and theoretical analytical methods are used in this study to estimate the power and efficiency produced by buoyancy wheels.

2. METHOD

2.1. Governing equation

The following is equation (1) of continuity for incompressible fluids based on the rule of conservation of mass [37].

\[
\frac{\partial \rho}{\partial t} + \frac{\partial \rho U_i}{\partial x_i} = 0
\] (1)

The Navier-Stokes momentum equation, equation (2), can be computed as follows by applying the momentum conservation law:

\[
\frac{\partial \rho U_i}{\partial t} + \frac{\partial \rho U_i U_j}{\partial x_j} = \frac{\partial \rho}{\partial x_i} + \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right)
\] (2)

where \( \rho \) represents the density of water, \( U_i \) the speed, \( t \) the time, and coordinates \( x_i \) and \( x_j \), in addition to turbulence and molecular viscosity by \( \mu \) and \( \mu_t \) respectively.

2.1.1. VOF method

The distance between the free liquid level and the bed bottom in the multiphase flow module is represented by the VOF model, which investigates the gas-liquid interface in unpressurized flow as a two-phase flow problem. It is defined that the Navier-Stoke equations apply to uniformly incompressible fluids [38]:

\[
\nabla \cdot \vec{u} = 0
\] (3)

\[
\frac{\partial \vec{u}}{\partial t} + (\vec{u} \cdot \nabla)\vec{u} = -\frac{1}{\rho} \nabla p + \mu \nabla^2 \vec{u} + \rho \vec{g}
\] (4)
Where \( \bar{u} \) denotes the speed, \( \mu \) is the dynamic viscosity coefficient, \( \bar{p} \) denotes pressure, \( \rho \) is the density of the fluid, and \( g \) represents the acceleration due to gravity. The location of the gas-liquid interface can be found by solving the volume fraction of each phase as follows:

\[
\frac{\partial \alpha_q}{\partial t} + \frac{\partial \alpha_q \bar{u}_i}{\partial x_i} + \frac{\partial \alpha_q \bar{u}_j}{\partial x_j} + \frac{\partial \alpha_q \bar{u}_w}{\partial x_w} = 0 \tag{5}
\]

Where \( \alpha_q \) is the volume fraction of water from each grid? When \( \alpha_q = 1 \), there is the same amount of water in the grid. There is no liquid in the grid if \( \alpha_q = 0 \). when the air-water-free is in the range of 0 to 1. After determining the volume fraction of water \( \alpha_q \), hence the gas volume fraction \( \alpha_p = 1 - \alpha_q \).

Equation (6) determines the volume fraction, which is used to model the density and dynamic viscosity coefficients [39]:

\[
\mu = \alpha_q \mu_q + (1 - \alpha_q) \mu_p \tag{6}
\]

\[
\rho = \alpha_q \rho_q + (1 - \alpha_q) \rho_p \tag{7}
\]

2.1.2. \( k - \varepsilon \) turbulence model

An enhanced method is employed to compute the turbulent viscosity in the acceptable \( k - \varepsilon \) model. Using the proper transport equation of the fluctuating components' vortex strength, one can construct the dissipation rate equation. It is considered the \( k \) model is admissibly more accurate than the \( k - \varepsilon \) model in forecast the dissipation rate distribution of flat and spherical jets. Furthermore, plausible \( k - \varepsilon \) models allow for better predictions of boundary layer parameters at large pressure gradients and in separated and revolving flows [40]. The model's transport equations are each written as [40].

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} \left( \left( \mu + \frac{\nu_t}{\sigma_k} \right) \frac{\sigma_k}{\sigma_\varepsilon} \right) + G_k + G_b - \rho \varepsilon - Y_M + S_k \tag{8}
\]

\[
\frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_i} \left( \left( \mu + \frac{\nu_t}{\sigma_k} \right) \frac{\sigma_\varepsilon}{\sigma_{\varepsilon}} \right) + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + G_{2\varepsilon} + G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon \tag{9}
\]

2.2. Model matematika

A full-size flow buoyancy wheel consists of an eight-bladed, straight, basic wheel. Diagonal propellers are being created to perform better in future studies. The rational area and flow area of the waterwheel are mostly included in a 3D model that is built using CAD software. The water wheel measures 0.16 meters in width (b), 1 meter in outer diameter (D), 0.3 meters in hub diameter (d), 0.0018 meters in thickness (t) for the wheel blades, and 0.02 meters inside clearance. This optimizes the energy in motion. Waterwheels are made of aluminum type 6061, which has a density of 2700 kg/m\(^3\). The general design specifications are shown in Table 1. Figure 1 shows the test model design for the buoyancy wheel operation.

<table>
<thead>
<tr>
<th>Table 1. Buoyancy wheel design parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Design parameters</strong></td>
</tr>
<tr>
<td>Rated output power, ( P_{\text{rated}} ) (kW)</td>
</tr>
<tr>
<td>The outer radius of the wheel, ( R ) (m)</td>
</tr>
<tr>
<td>Waterwheel width, ( b ) (m)</td>
</tr>
<tr>
<td>Blade dimensions (m)</td>
</tr>
<tr>
<td>(Length x Width x Thickness)</td>
</tr>
<tr>
<td>Blade shape</td>
</tr>
</tbody>
</table>

For the analytical design of the water wheel, the researchers provided technical background. Cornelis defines the power output of a water wheel as Equation (10) [41]:

\[
P_{\text{out}} = T \cdot \omega \tag{10}
\]

Where \( T \) shows the torque and angular speed of the water wheel. Plotkin describes the resulting torque in Equation (11) [42]:
Equation (10) added to Equation (11) will produce equation (12) [43]:

\[ P_{out} = F \cdot \omega \cdot \frac{D}{2} \tag{12} \]

Floating water wheels are designed to operate in both air and water zones, as seen in Figure 2. Figure 2 shows a floating water wheel with a free-body diagram representing the ambient conditions and working forces. The buoyant force of the air in the water zone and the gravity of the water in the air zone are what cause this water wheel to circle. The water wheel is turned by the torque created by the water-filled bucket falling through the air [24]. There is more air trapped in bucket 4 than water when it reaches bucket 5 in position. Because of this, the combined bucket's density is less than that of water, creating a buoyant force (Fb). This process is repeated for all eight buckets, leaving four in the various work zones with water and four with air inside [44]. The combined weight of the water and the air's lifting force generates the torque and power that the water wheel produces as it rotates [45]. The following equations (14)–(21) can be used to compute the force applied to the floating water wheel.

\[ d_{shaft} = \sqrt{\frac{16 \cdot \tau_{max} \cdot FS}{\pi \cdot \sigma_{yield}}} \tag{13} \]

The buoyancy wheel is designed to work in both air and water zones, as seen in Figure 2. Figure 2 shows a buoyancy wheel with a free-body diagram representing the working forces and ambient conditions. The buoyant force of the air in the water zone and the gravity of the water in the air zone are what cause this water wheel to circle. The water wheel is turned by the torque created by the water-filled bucket falling through the air [24]. There is more air trapped in bucket 4 than water when it reaches bucket 5 in position. Because of this, the combined bucket's density is less than that of water, creating a buoyant force (Fb). This process is repeated for all eight buckets, leaving four in the various work zones with water and four with air inside [44]. The combined weight of the water and the air's lifting force generates the torque and power that the water wheel produces as it rotates [45]. The force exerted on the buoyancy wheel can be calculated using equations (14)-(21) below.

\[ F_1 = w \cdot \cos \theta = (m_{air} \cdot \cos 22.5^\circ + m_{blade} \cdot \cos 45^\circ) \cdot g \tag{14} \]
\[ F_2 = w \cdot \cos \theta = (m_{\text{air}} \cdot \cos 22.5^\circ + m_{\text{blade}}) \cdot g \]  
(15)

\[ F_3 = w \cdot \cos \theta = (m_{\text{air}} \cdot \cos 22.5^\circ + m_{\text{blade}} \cdot \cos 45^\circ) \cdot g \]  
(16)

\[ F_4 = w \cdot \cos \theta = (m_{\text{water}} \cdot \cos 45^\circ) \cdot g \]  
(17)

\[ F_5 = \rho_{\text{water}} \cdot g \cdot V_{\text{bucket}} \cdot \cos 22.5^\circ - \rho_{\text{material}} \cdot g \cdot V_{\text{blade}} \cdot \cos 45^\circ \]  
(18)

\[ F_6 = \rho_{\text{water}} \cdot g \cdot V_{\text{bucket}} \cdot \cos 22.5^\circ - \rho_{\text{material}} \cdot g \cdot V_{\text{blade}} \]  
(19)

\[ F_7 = \rho_{\text{water}} \cdot g \cdot V_{\text{bucket}} \cdot \cos 22.5^\circ - \rho_{\text{material}} \cdot g \cdot V_{\text{blade}} \cdot \cos 45^\circ \]  
(20)

\[ F_8 = \rho_{\text{water}} \cdot g \cdot V_{\text{bucket}} \cdot \cos 22.5^\circ \]  
(21)

To make the moment of inertia calculation easier, it is assumed that the water wheel is straight. At that point, the waterwheel bucket can be roughly modeled as a solid cylinder [46]. where \( m \) is the water wheel's mass and \( r \) is its outer radius. Therefore, using equation (22) the buoyancy wheel's inertia can be computed as follows [47]:

\[ I = \frac{1}{2} \cdot m \cdot r^2 = \frac{1}{2} \rho \cdot V \cdot r^2 \]  
(22)

Equations (23) and (24) are used to determine the angular velocity (\( \omega \)) and wheel speed respectively [48],[49]:

\[ \omega = \frac{2\pi \cdot N}{60} \]  
(23)

\[ v = \omega \cdot r \]  
(24)

where \( N \) is the turbine revolutions per minute. Equation (25) is used to calculate the input power (\( P_{in} \)) [50]:

\[ P_{in} = \rho \cdot g \cdot Q \cdot H \]  
(25)

Where \( Q \) is discharge, \( H \) is head, \( \rho \) is water density and \( g \) is acceleration caused by gravity. Equation (26) is the ratio of the power available at the shaft (\( P_{out} \)) to input power (\( P_{in} \)) called hydraulic efficiency (\( \eta \)) [51]:

\[ \eta = \frac{P_{out}}{P_{in}} \]  
(26)

2.3. Boundary conditions and simulation setup

As seen in Figure 3, boundary conditions (BC) define a rotating domain and a static domain. In the rotational domain, the water wheel is set up as a wall-type BC, and the k-ε turbulence model used in this simulation is derived from earlier studies on pico hydro water turbines conducted by different researchers [52]. Since this simulation is multiphase, the rotating domain is where fluids like air and water are defined [53]. Both have a zero-speed initialization. Water and air were once half-domains that were both stationary and rotating. Domain gravity is 9.81 m/s^2 about the suitable coordinate system [54]. The right-hand rule states that throughout domain movement, the domain will spin at a constant speed of 69 revolutions per minute. Numerous boundary conditions (BCs) are provided, such as zero outlet pressure, one meter per second (m/s) input velocity for air 0.121 m/s^3 for water utilizing Qin, and non-slip wall boundary conditions. In a similar vein, it is believed that the domain's surface facing the atmosphere serves as a symmetric border condition. Inner and outer interfaces link the spinning and stationary domains [49].

P-V coupling is handled by the SIMPLE scheme, momentum is handled by the Body Force Weighted for Pressure scheme, and spatial discretization is handled by the 2nd-order upwind scheme [55]. The convergence criteria of the turbulence equations, momentum, and continuity requirements are satisfied by the values 1x10^-4 [56]. An implicit scheme is used to calculate the volume fraction. To achieve a balance between accuracy and efficiency, the simulation time for each time step is set to 20 iterations. A time step size of 0.025s was chosen for transient simulation.
2.4. Grid independency test

A three-dimensional (3D) model of the water wheel and the surrounding flow field domain is created to compare with the analytical results. The characteristics and performance of the buoyancy wheel were analyzed using ANSYS 18.1. The wheel and fluid domains use tetrahedral meshes due to the complexity of the system [57]. The goal of the Grid Independence Test is to ensure that the results do not depend on additional grid refinements [58]. The grid Independence Test is a technique to reduce computing time and costs by avoiding using the best grid. The test-independent grid can be said to have converged if the error is less than 1% [59], [60]. Numerical analysis was performed and five grids with varying degrees of fineness were created in this study. By plotting torque results against the number of elements in a particular mesh type, the Grid Independence Test shows that grid size significantly influences turbine torque performance. From Figure 4 it can be seen that a mesh with 0.57 million elements produces almost the same power results as a mesh with 0.86 million elements and the mesh distribution used is shown in Figure 5. Therefore, to get accurate results with less time and computational costs in this study, a mesh with 0.57 million elements was used for this simulation.

3. RESULTS AND DISCUSSION

3.1. Theoretical analysis results
First estimates of the water resources at the mechanical engineering laboratory were used to build the buoyancy wheel, which has a 2 m head and a 0.121 m$^3$/s water output. Equations (14)–(17) are used to determine the buoyancy force components while equations (18)–(21) are used to calculate the gravity components of the buoyancy wheel (Figure 2). Table 2 and Figure 6 exhibit the style analysis's findings.

<table>
<thead>
<tr>
<th>No. Bucket</th>
<th>Force (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bucket 1</td>
<td>115.98</td>
</tr>
<tr>
<td>Bucket 2</td>
<td>91.33</td>
</tr>
<tr>
<td>Bucket 3</td>
<td>32.39</td>
</tr>
<tr>
<td>Bucket 4</td>
<td>5.02</td>
</tr>
<tr>
<td>Bucket 5</td>
<td>4.26</td>
</tr>
<tr>
<td>Bucket 6</td>
<td>26.46</td>
</tr>
<tr>
<td>Bucket 7</td>
<td>86.26</td>
</tr>
<tr>
<td>Bucket 8</td>
<td>87.37</td>
</tr>
<tr>
<td>Total</td>
<td>449.06</td>
</tr>
</tbody>
</table>

Figure 6. Forces acting on a floating waterwheel.

Equation (11) is used to compute the buoyancy wheel's torque component, which equals 224.53 N-m when the force produced by the buoyancy wheel's radius is multiplied. Equations (22), (23), (13), and (24) are used to compute the moment of inertia (I), angular velocity ($\omega$), shaft diameter (do), and buoyancy wheel speed (v) from the buoyancy wheel design. Table 3 displays the findings of the parameter data.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Moment of Inertia, I (kg.m$^2$)</td>
<td>4.31</td>
</tr>
<tr>
<td>Rotation speed, $\omega$, (rad/s)</td>
<td>7.21</td>
</tr>
<tr>
<td>Shaft diameter, d$_o$ (mm)</td>
<td>30</td>
</tr>
<tr>
<td>Buoyancy wheel speed, v (m/s)</td>
<td>3.61</td>
</tr>
</tbody>
</table>
3.2. CFD analysis results
3.2.1. Effect of buoyancy force on buoyancy wheel performance

The water volume fraction in the eight buckets can function because of the difference in water height between the water and air zones, according to the simulation findings shown in Figure 7. The buoyancy wheel is operated by the force mechanisms of buoyancy and gravity. The findings demonstrated that as the amount of water held in the bucket rose, its buoyancy increased as well. The buoyancy wheel's air and water fluids have different specific gravities of 0.00123 and 0.998, respectively, which causes an increase in buoyancy force. When the air-filled bucket starts to enter the water zone, there is a difference in specific gravity that causes a buoyant force that pushes the bucket upward and causes the wheel to rotate. Every object submerged in a fluid, whether fully or partially, will experience an upward buoyant force equal to the weight of the fluid displaced by the object, according to Archimedes' Law [61].

![Figure 7. The volume fraction of water working on the buoyancy wheel.](image)

3.2.2. Speed contours on the buoyancy wheel

Figure 8 further demonstrates that, throughout the majority of the domain width, the fluid velocity in the upstream region is constant. The downstream velocity profile, which also demonstrates that the fluid velocity is zero at the wall but changes rapidly across the width of the fluid domain due to the rotational motion of the turbine, indicates that the fluid velocity behind the turbine is lower due to the development of the wake region and the extraction of turbine kinetic energy from water molecules. After the water's kinetic energy is extracted by the turbine's buoyancy, the water's kinetic energy is diminished downstream, and the mechanical energy is accessible at the turbine shaft.

![Figure 8. Buoyancy wheel speed contour](image)

Simultaneously, a high-velocity zone appears at the advancing blade tip, indicating the dynamic nature of fluid dynamics during the simulation. Contour plots effectively highlight these variations, with red representing high speeds and blue depicting low speeds. This visual representation allows a clear
understanding of fluid behavior during turbine bucket rotation. Figure 8 shows the findings of the distribution of water flow velocity from each waterwheel bucket. With a maximum value of 30 m/s, the red color shows the area with the highest flow speed, while the light blue color shows the area with the lowest water flow speed, namely 3.3 m/s. The resulting buoyancy wheel speed is 3.6 m/s during this simulation.

3.3. Buoyancy wheel performance validation

It is possible to comprehend buoyancy wheel performance efficiency by employing numerical flow simulation analysis in conjunction with theoretical study. While computational analysis can offer a more comprehensive understanding of the factors influencing buoyancy wheel efficiency, numerical flow simulation analysis can offer a more accurate representation of the water flow surrounding a buoyancy wheel [62]. As demonstrated in Table 4, these two analytic methods can be used concurrently for more accurate findings. Equation (25) yields an input power (Pin) of 2378.99 W. In addition, 414.96 N of force and 207.48 N-m of torque are produced in the numerical flow simulation. The buoyancy wheel's performance efficiency can be computed using equation (26). It was discovered that the power generated by the numerical flow simulation analysis was 1495.95 W, while the power generated by the theoretical analysis utilizing equation (10) was 1619.35 W. In the numerical flow simulation, the buoyancy wheel produced an efficiency of 62.88%, but in the theoretical study, it was 68.07%. These findings show that the buoyancy wheel's performance created a 7.62% variance.

4. SIMPULAN

Indonesia's annual demand for electrical energy is still rising. Research is being done on renewable energy as a source of energy for producing power because non-renewable natural resources are becoming scarce. It is appropriate to use CFD techniques to look into physical flow phenomena in the research of buoyancy wheels to completely comprehend the energy conversion process. Moving meshes are a common way for CFD algorithms to provide the boundary conditions for transient approximations for rotating case objects. Thus, using theoretical analysis techniques and CFD with moving mesh as a transient methodology for the buoyancy wheel, the goal of this research is to ascertain the impact of the ensuing buoyant force on the buoyancy wheel's performance. The force and torque generated by numerical calculations and simulations are 449.06 N and 224.53 Nm, respectively; 414.96 N and 207.48 Nm, according to the research findings. Additionally, the power generated by the numerical simulation and theoretical analysis is 1495.95 W and 1619.35 W, respectively. The buoyancy wheel generates an efficiency of 68.07% and 62.88%, yielding a 7.62% variance.

ACKNOWLEDGEMENT

Thank you to LEMLITBANG UHAMKA for funding this internal research with contract number 53/F.03.07/2022 and the Faculty of Industrial Technology and Information Technology (FTII) UHAMKA for facilitating research facilities and infrastructure.

REFERENCE


[38] T. Bikmukhametov, “CFD Simulations of Multiphase Flows with Particles,” Norwegian
Effect of buoyancy force on buoyancy waterwheel efficiency using numerical flow simulation


