Volume Of Fluid (VOF) Method Simulation Of High Rate Algae Pond (HRAP)

Mohammadreza Vafaee, Morteza Almassi, Mohammad Hydarian, Hamid Mashhadi Meyghan

1PhD student, Department of Agricultural Mechanization, Science and Research Branch, Islamic Azad University, Tehran, Iran. No 20, Unit 4, Hakim High way, Taavon bolivar, Farsad St, Edalat St, 4Th west Alley, Tehran, Iran.
2Scientific Member, Department of Agricultural Mechanization, Islamic Azad University, Tehran, Iran.
3Department of Biotechnology, Iranian Research Organization for Science and Technology (IROST), Tehran, Iran. Iranian Research Organization for Science and Technology (IROST), Sh. Ehsani Rad St., Enqelab St., Parsa Sq., Ahmadabad Mostoufi Rd., Azadegan Highway, P. O. Box 3353-5111, Tehran 3353136846, I. R. Iran.
4Scientific Member, Arak Branch, Islamic Azad University, Arak, Iran. P. O. Box 38135-567, Agricultural and Natural Research Faculty, Islamic Azad University, Arak, Iran.

ABSTRACT

The Algae fuel price is highly affected by operating costs which is the function of energy efficiency of raceway ponds. A two phase, three-dimensional Computational Fluid Dynamic model is used to investigate the various pond configurations effect on power consumption of paddlewheel and stagnation regions within the flow. Water free surface is modeled by volume of fluid (VOF) method and the paddlewheel movement is taken into account by implementing sliding mesh approach. Minimum stagnation region and best velocity uniformity is seen in simple curved inner wall channel at bends and maximum stagnation region is observed in U type channel. Adding another paddlewheel in U type channel drastically decrease the stagnation regions while it has negative effect on simple curved bend channel which has the minimum stagnation region. Using the flow rectifiers on channel bend has an adverse effect on the flow field and the stagnation region is increased in to inner side of baffles, also vortex shedding occurs at the downstream of baffles. The paddlewheel movement influences the flow field. Placement of paddlewheel in the center of channel causes reduction in flow velocity after the paddle as it approaches the channel bends. Maximum velocity of the fluid is close to free surface and occurs after some distance from the paddlewheel due to water depth variation induced by paddlewheel movement.

INTRODUCTION

The world is highly dependent on petroleum-based fuels as the primary transportation fuel. In recent years, the heightened awareness of greenhouse gas emissions led to an increase in research and development of alternative fuel sources. Most sources of energy on earth can be said to originate from energy transmitted from the sun. Oil and petroleum fields around the world were originally large amounts of organic matter, such as algae, that under specific conditions were transformed into fossil deposits through slow chemical reactions. The most productive source of ‘fresh’ oil is green algae, microscopic water living organisms, which capture sunlight and store it as oil [1].

Cultivation of microalgae has received increasing attention over the last few years, as it is comprised of triglycerides and other fatty acids, which can be converted to a variety of fuel. There are several approaches for this process, including the conversion of lipids to FAME biodiesel via transesterification [2], or the conversion of lipids to n-alkanes through either thermo cracking or catalytic hydro treating [3]. FAME Biodiesel generally consists of long chain alkyl esters while conventional diesel consists of a mixture of alkanes, naphthenic, and aromatics. Most of the existing technologies to produce fuel from vegetable oils and lipids involve the production of fatty acid methyl esters (FAME).

Byproducts such as biomass could be used for production of pharmaceuticals, chemicals, and Ethanol. These products are more valuable than livestock feed and could potentially bring in even greater value. Carbon credits from potential cap-and-trade programs could be considered as an additional source of byproduct revenue.
Due to their flexibility, costs and easy to scale-up, almost all commercial algal biomass production is currently produced with open ponds [4, 5]. Generally shallow algal ponds are used to ensure adequate exposure to sunlight [6, 7]. The velocity between 0.1 to 0.3 m/s is commonly appropriate to reach a good mixing and avoid the stagnation regions within the flow [8], which causes sedimentation of cultivation phase in raceway ponds. The low velocity mixing regions are known as “dead zones”. These dead zones reduce the productivity of algae in raceway ponds due to developing the anaerobic conditions [9, 10].

A typical raceway pond consists of a closed loop channel along with one or more paddle wheels are used to keep the algae circulation. Pond geometry, the hairpin bends shape [11], flow rectifiers and the paddlewheel location influence the hydrodynamic characteristics of flow field in the raceway and pond and affecting the power requirement to circulate water through the system [9, 12].

Although some numerical investigations studied the effects of channel geometry on flow behavior [9, 11, 13-15] in raceway ponds, the effect of paddlewheel movement and flow depth variation is not taken into account yet. We studied three dimensional raceway real scale pond with the paddle wheel rotation was included in the calculation.

**Numerical modeling:**

Solutions domain (Fig. 1) includes a pond with a total volume of 3000 m$^3$ with a length to width ratio equal to 10 and a height of 50 cm. The depth of water in the pond is 25 cm and is initially at rest. The paddlewheel which rotates with the constant angular velocity of 12 rpm circulates the fluid in the pond. A Stationary and a rotating mesh domain are used to allow for the relative motion between the paddlewheel and a pond. The Arbitrary Mesh Interface (AMI) is used as a boundary condition to couple the flow between stationary and moving zones. We use OpenFOAM software version 2.1.1 which is an open-source software package with the interDyMFoam solver, which is capable of solving unsteady multiphase flow with considering rotational movement of the mesh domain around a paddlewheel, to solve the equations of motion. The constant pressure boundary condition is applied to the top of channel which is open to air. The channel walls and spokes of paddle are defined as no slip walls by specifying zero velocity in all directions. The Arbitrary Interface (AMI) is used as a boundary condition to couple the flow between stationary and moving zones.

Three equations of conservation of momentum in Cartesian coordinate system have to be solved. These equations are:

$$\frac{\partial \mathbf{u}}{\partial t} + \frac{\partial (\rho \mathbf{u}_j)}{\partial x_j} = 0$$  \hspace{1cm} (1)

$$\frac{\partial \mathbf{u}_i}{\partial t} + \frac{\partial (\rho \mathbf{u}_i \mathbf{u}_j)}{\partial x_j} = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} - \frac{\partial \mu}{\partial x_j} \mu \left( \frac{\partial \mathbf{u}_i}{\partial x_j} + \frac{\partial \mathbf{u}_j}{\partial x_i} - \frac{2}{3} \frac{\partial \mathbf{u}}{\partial x_i} \right) - \frac{\partial \mathbf{v}}{\partial x_i} + g + \frac{1}{\rho} f_{SR}$$  \hspace{1cm} (1)

Which $\mathbf{u}$ is the velocity vector, $P$ denotes the pressure, $\mu$ is the fluid viscosity, $\rho$ is the density, $g$ is acceleration due to gravity, and $f_{SR}$ is surface force due to surface tension. $\mathbf{v}$ is the Reynolds stress components which can be determined by solving turbulence equations and the means of eddy dissipation concept. Properties such as density and viscosity are weighted with the volume fraction of cells in each phase, by the following equations:

$$\rho = \sum \rho_i \alpha_i$$ \hspace{1cm} (2)

$$\mu = \sum \mu_i \alpha_i$$ \hspace{1cm} (3)

VOF method is used to capture the air-liquid free surface. In this method, volume fraction of each cell ($\alpha_i$) obtained by the following equation:

$$\left[ \frac{\partial (\rho \alpha_i)}{\partial t} + \mathbf{v} \nabla (\rho \alpha_i) \right] = 0$$ \hspace{1cm} (4)

The air is taken as the primary phase. Volume fraction of air is calculated by the fact that summation of volume fractions is equal to 1.
Simulation is performed with the k-epsilon type, renormalization group (RNG) turbulence model [18, 19] coupled with a Wall function for wall treatment, in order to model the hydrodynamics of turbulence. The equation for the turbulent kinetic energy $k$ is as follows:

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho ku_i)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \rho \varepsilon$$

(6)

A model equation for Turbulent dissipation rate ($\varepsilon$) is derived by multiplying the $k$ equation by $(\varepsilon/k)$ and introducing model constants. The following model equation for $\varepsilon$ is solved to get turbulence dissipation rate

$$\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_i)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} P_k - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k}$$

(7)

where

$$C_{2\varepsilon} = C_{\varepsilon} + \frac{C\mu_t n^3 (1 - \eta/\eta_0)}{1 + \beta \eta^3}$$

(8)

The $\beta=0.012$, $\eta_0=4.38$ and the quantity $\eta$ is obtained as follows.

$$\eta = \frac{Sk}{\varepsilon}$$

(9)

And $S$ is the viscous dissipation term and is given by:

$$S = \left( 2\varepsilon k \right)^{1/2}$$

(10)

The Prandtl number $\sigma$ connects the diffusivity of $\varepsilon$ to the eddy viscosity. It has value of 0.7194 in RNG model, and it is equal to $\sigma$. The values for the model constants $C_{1\varepsilon}$ and $C_{2\varepsilon}$ are equal to 1.44 and 1.68 respectively. The turbulent viscosity ($\mu_t$) is calculated from

$$\mu_t = \rho C_{\mu} \frac{k^2}{\varepsilon}$$

(11)

which $C_{\mu}=0.845$. $P_k$ is a turbulence production due to viscous forces, which is modeled using:

$$P_k = \mu_t S^2$$

(12)

The solution algorithm is based on Pressure Implicit with Splitting of Operators (PISO) scheme. The governing equations are advanced in time using a second-order semi-implicit scheme and all transport equations is discretized by the QUICK scheme. In the studies reported in this paper, the unsteady calculation took approximately 200 h on an eight processor computer. The time step size is equal to 0.0001 sec, which guarantees the courant number below 0.2 and the total time of simulation is taken as 600 sec as we observe repeating pattern in the flow field and achieve the quasi steady solution.

The mesh consists of two stationary domain (pond and the inner cylinder), and one rotating domain (hallow cylinder which contains the paddle wheel). The numerical grid contains Hexahedral cells which we use an unstructured cooper mesh near the paddlewheel due to complexity of geometry and the rest of the domain, is meshed by the structured map scheme. A mesh independency study is then carried out in order to ensure that the solution is independent of the mesh size. it was found that the 1.2 million cells number is sufficient to achieve a mesh independent solution. Since the majority of the energy loss occurs at the hairpin bends, which momentum changes direction, the effect of various channel shapes, the curvature shapes and the impact of adding baffles on the flow field are studied.

RESULTS AND DISCUSSIONS

The dead zone is defined as the region which its velocity is below than 0.1 m/s. The power input to the paddlewheel is calculated from the torque $T$, on the impeller shaft which is the function of pressure difference $(P_1 - P_2)$ and shear stress $(\tau_j)$ applied on the blades.

$$T = \sum_i (P_1 - P_2) \cdot r_i \cdot \partial A_i + \sum_i \tau_{ij} \cdot \partial A_i$$

(13)
summation around the control cells \( i \) is corresponded to each blade and \( r \) denotes the distance from the axis of rotation. The power needed to rotate the paddlewheel can be easily calculated according to:

\[
Power = 2\pi \frac{\text{rpm} \cdot T}{60}
\]  

(14)

Due to blade interaction with the fluid free surface, the input power has a repetitive pattern as we reach a quasi-steady solution. The maximum power needed to rotate the paddlewheel is about twice of mean power.

As can be seen in Error! Reference source not found. and table 1. The channel shape of type “a” has the minimum dead zone volume. By adding another similar paddlewheel (type “b”) the dead zone volume increases due to separation of boundary layer from the corner of inner arc. The U channel shape (type “c”) had the maximum dead zone. U-shaped channel can be used when the land dimensions is not appropriate for the construction of the common ponds. Here, Increases of dead zone volume occur as the flow momentum needs two changes its direction two times of straight channel and the paddlewheel get closer to channel hairpin bends (as in “f” channel). The bend wall acts as a barrier to the flow acceleration, consequently, shortly after the paddlewheel the hydrostatic energy cannot be easily converted to kinetic energy. So it causes to decrease in the flow velocity and increase of dead zone volume. Adding another paddlewheel to U type channel (type “d”) drastically decreases the dead zone but it needs a more mean power to circulate a flow in pond. As we use simple straight channel without the curved inner walls at bends (Type “e”) the dead zone is increases in comparison to inner side curved channel of type “a” and we will have high flow separation at the channel bends.

To prevent the flow separation two curved walls are added to inner side of pond bends with various paddlewheel locations (type “f”, “g” and “h”). As can be seen in Error! Reference source not found., locating the paddlewheel on the center of channel (type “f”) causes the reduction in flow velocity after the paddle as it approaches the channel bends. Thus a large volume of dead zone appears in front of paddlewheel. placing the paddle wheel just after the bend (type “g”) increases the dead zone in the outer side of the bend, due to suction of adjacent fluid of paddle wheel and shifting the maximum velocity in to the inner curvature.

We also examine adding the baffles to the channel bends. As it can be seen in Fig. 2 it has an adverse effect on the flow field and the dead zone is increased in the inner side of baffles. Also the vortex shedding occurs at the downstream of baffles and affect the fluid flow (Fig. 4). Tough the baffles reduce the flow separation from the inner surface of the bend and decrease the intensity of vortices formed by the flow separation. Also it can be mentioned that, while the flow cross section area changes smoothly, it would not have an effect on the increasing of dead zones volumes. (Type “i”).

**Table 1:** Power requirement and the dead zone volume for various channel shapes

<table>
<thead>
<tr>
<th>Channel shape</th>
<th>Mean Power of first paddle wheel (W)</th>
<th>Mean Power of second paddle wheel (W)</th>
<th>Dead zone volume (m³)</th>
</tr>
</thead>
<tbody>
<tr>
<td>a</td>
<td>1211.7</td>
<td>-</td>
<td>0.92</td>
</tr>
<tr>
<td>b</td>
<td>577.8</td>
<td>585</td>
<td>12.01</td>
</tr>
<tr>
<td>c</td>
<td>1421.2</td>
<td>1128.2</td>
<td>8.51</td>
</tr>
<tr>
<td>d</td>
<td>1116.3</td>
<td>1038.7</td>
<td>24.35</td>
</tr>
<tr>
<td>e</td>
<td>1193.5</td>
<td>-</td>
<td>79.92</td>
</tr>
<tr>
<td>f</td>
<td>1029.6</td>
<td>-</td>
<td>28.53</td>
</tr>
<tr>
<td>g</td>
<td>885.8</td>
<td>-</td>
<td>69.92</td>
</tr>
<tr>
<td>h</td>
<td>1412.1</td>
<td>-</td>
<td>23.42</td>
</tr>
</tbody>
</table>

The velocity and pressure for type “a” pond on three lines after the paddlewheel which have a distance of 12.5 cm from the bottom of the pond were plotted in Fig. 5. At the downstream of paddle, as we distances from the paddlewheel, the flow velocity firstly decreases because of increasing the height of liquid and the static pressure (Fig. 6), but due to conversion of fluid static pressure to kinetic energy the flow velocity increases. Sudden increase in velocity near the paddlewheel is because of paddle spokes movement which affects the fluid flow. Near the entrance of the bend the flow accelerates in to the outer curvature due to the centrifugal force and thus the flow velocity increases adjacent to the outer wall and decreases near the inner wall. On the other side of the channel which does not have paddlewheel, the velocity is almost uniform and just slightly changes near the channel bends due to centrifugal force (Fig. 7).

Fig. 8 shows the velocity profile at different channel depths. As it can be seen, the maximum velocity occurs near sides of the channel. The fluid velocity rapidly increases along the channel depth, so that the maximum velocity of the fluid will be close to free surface.

**Conclusion:**

Different ponds with various geometry and paddle configuration are numerically studied. As mentioned the paddlewheel rotation and flow depth variation influence the fluid flow in the pond. The “a” type channel has the minimum dead zone and the more uniform velocity observed in this kind of channel. Maximum dead zones
occur in the U shape channel (type “c”) which adding another paddlewheel greatly reduces the dead zones. Adding a paddlewheel to “a” type channel (type “b”) has inverse effect and increasing the dead zones volume. As paddle approaches the side of channel the dead zones volume increases the best location for a paddle wheel is found to be about a one third of channel length to reduce the stagnation region within the flow. Using flow rectifiers at the bend has an adverse effect on the flow field and the dead zone is increased in the inner side of baffles. Also the vortex shedding occurs at the downstream of baffles and reduces the flow uniformity.

Fig. 1: Solution domain
Fig. 2: Dead zone volumes colored by fluid velocity for various channel shapes
Fig. 3: Velocity vectors for different channel types at the middle depth
Fig. 4: Contours of velocity for various channel shapes, at the surface which locates 12.5 cm from the bottom.

Fig. 5: Velocity after the paddle wheel at 12.5 cm from the bottom for the “a” type pond positive values to negative values of the position.
Fig. 6: Pressure after the paddle wheel at 12.5 cm from the bottom for the “a” type pond

Fig. 7: Velocity variation on the sides of “a” type pond which does not have paddlewheel (the flow direction is from the

Fig. 8: Velocity profile 15 meter after the paddle at the various depths for the “a” type pond

Fig. 9: Pressure contours for the ponds of type “h” and “i” at surface which locates 12.5 cm from the bottom
REFERENCES


