NUMERICAL SIMULATION OF TURBULENT ENVIRONMENTAL FLOWS

Jucá, P.C.S. and Maliska, C.R.
Computational Fluid Dynamics Laboratory - SINMEC
Department of Mechanical Engineering - UFSC
88040-900 - Florianópolis - SC - Brazil

INTRODUCTION

Water is one of the most basic resource used by the human being and since the antiquity has influenced the geographical location of the ancient human agglomerates. To maintain the quality of the water in closed water bodies is a delicate issue due to their limited capacity in assimilating and dispersing the pollutants disposed in it by direct (e.g. waste waters) or indirect actions (e.g. agriculture chemicals carried by rain) of local populations. In studying the effects of any kind of water pollution in closed water bodies the knowledge of its hydrodynamic behavior is of utmost importance. This is one of the most determining factor in the capacity of the water body in assimilating and dispersing any pollutant. For a lake, for example, the hydrodynamic pattern is basically determined by the wind currents acting over its surface. The surface of a lake in a wind day shows surface waves that move toward the wind direction and transport a large amount of energy. However, this waves causes a very little transport of mass due to the oscillatory motion, since particles in waves actually moves in orbits. So, when bulk mass transport must be considered, the wave motion is not of fundamental importance. The shear stress, in the other hand, acting in the surface due the wind motion causes a turbulent movement that is not of oscillatory nature and is quite steady, the so called wind driven currents. Therefore, numerical models able to predict the flow pattern in closed water bodies such channels, lakes and lagoons are very useful tool for environmental studies in such sites. In this paper it is reported the simulation of turbulent wind driven flows in closed water bodies. The main goal is validate the mathematical model using a laminar and a turbulent analytical solution for comparison.

THE MATHEMATICAL MODEL

This class of flow is ruled by the Navier-Stokes equations, the mass conservation equation and a turbulence model to describe the turbulent nature of the motion. Using $\phi$ to represent any of the dependent variables, a general conservation equation can be written as
\[
\begin{align*}
\frac{1}{\sqrt{\gamma}} \left( \begin{array}{c}
\frac{\partial \phi}{\partial t} \\
\frac{\partial \psi}{\partial t} \\
\frac{\partial \chi}{\partial t}
\end{array} \right) + \frac{\partial}{\partial x} \left( \begin{array}{c}
\frac{\partial \phi}{\partial x} \\
\frac{\partial \psi}{\partial x} \\
\frac{\partial \chi}{\partial x}
\end{array} \right) + \frac{\partial}{\partial y} \left( \begin{array}{c}
\frac{\partial \phi}{\partial y} \\
\frac{\partial \psi}{\partial y} \\
\frac{\partial \chi}{\partial y}
\end{array} \right) + \frac{\partial}{\partial z} \left( \begin{array}{c}
\frac{\partial \phi}{\partial z} \\
\frac{\partial \psi}{\partial z} \\
\frac{\partial \chi}{\partial z}
\end{array} \right) &= \mathbf{P} + \mathbf{S}
\end{align*}
\]

where, due to the irregular shapes of water bodies, a general body-fitted curvilinear coordinate system is employed. If an unstructured grid is used, as in this work, the boundary conditions for the governing equations are easily imposed using this type of coordinate systems. In Eq. (1) \( U, V, W \) are the contravariant components of the velocity vector given by

\[
\begin{align*}
U &= (u_1 \xi_x + u_2 \xi_y + u_3 \xi_z) J^3 \\
V &= (u_1 \eta_x + u_2 \eta_y + u_3 \eta_z) J^3 \\
W &= (u_1 \gamma_x + u_2 \gamma_y + u_3 \gamma_z) J^3
\end{align*}
\]

where \( J \) is the Jacobian of the coordinate transformation and \( u_1, u_2, u_3 \). The right hand side contains the pressure source and the diffusion terms. The contravariant velocity components are responsible for the advection across coordinate lines. As stated, \( \phi \) represents the dependent variables for each equation. For \( \phi = 1, u_1, u_2, u_3 \), the mass conservation equation, momentum in x, y and z, are recovered, respectively \( \Gamma \) represents the diffusivity transport coefficient, being zero for the mass conservation equation and equals to the effective viscosity \( (\mu_d) \) for the Navier-Stokes equations. Also in Eq. (2) \( u_i (i=1,2,3) \) represents the mean of fluctuating Cartesian velocities as defined by Rosh [1]. The expressions for \( P^\phi \) and for the source terms \( S^\phi \) can be found in Juca and Maliska [7] and details of coordinate transformation in Maliska [2].

Eq. (1), representing the system of partial differential equations governing the flow, is integrated in time and in a 3D elemental control volume. The velocity-pressure coupling is handled using the SIMPLEC method of Van Doormal and Raithby [3].

Fictitious control volumes are used for the application of the boundary conditions. Due to the 3D nature of the problem, numerical details and expressions of the coefficients are not given here. These can be found in Juca [4] and Maliska [2]. To deal with turbulent flows a turbulence model is implemented and will be discussed in detail later. In the Navier-Stokes Eq. (1), the diffusivity coefficient \( \Gamma \) is the effective viscosity defined by \( \mu_d = \mu + \mu_t \), where \( \mu \) and \( \mu_t \) are the turbulent and laminar viscosity, respectively.

**Boundary Conditions**

Typically, in an environmental flow we have as solid boundaries the shoreline and the bottom of the water body. At the surface the wind action will define the flow pattern in the water. For the solid and surface boundaries no normal flux of momentum is allowed. The turbulent stresses at the solid boundaries, \( \sigma_b \), are determined by

\[
\sigma_b = \frac{k_v v_b}{\ln \left( \frac{\Delta Z_b}{Z_0} \right)}
\]

where \( v_b \) is the component of the velocity parallel to the boundary, \( k_v = 0.41 \) is the von Kármán constant, \( \Delta Z_b \) the distance from the boundary to the nearest grid point and \( Z_0 \) a parameter dependent on the local boundary roughness (Jin [5]). Assuming hydraulically smooth flow Eq. (3) assumes that the velocity near the solid boundaries matches the logarithmic law of the wall. At the surface the wind stress is often calculated by (Huang and Spaulding, [6])

\[
\tau_s = \rho_n C_n v_{wind}^n
\]

where \( C_n \) is the air-water drag coefficient, \( \rho_n \) the air density and \( v_{wind} \) is the wind velocity measured at 10 meters above the water level. The stresses at the boundary imposes the boundary conditions for the velocity as shown in Juca and Maliska [7].

**CLOSED CHANNELS UNDER SURFACE WIND ACTION: LAMINAR MODEL VERIFICATION**

For channel flows as depict in Fig. 1, the simplest case is when a laminar flow is imposed, and an analytical solution is possible. This problem, in spite of its simplicity, is an efficient test for the numerical model since a 3D grid is used for this case. Symmetry conditions are, therefore, also tested. The Navier-Stokes equations for this two-dimensional case is reduced to

\[
\frac{\partial^2 u}{\partial z^2} = 0
\]

under the boundary conditions

\[
\begin{align*}
\frac{\partial u}{\partial z} &= \tau_s \quad \text{at} \quad z = D \\
u(z) &= 0 \quad \text{at} \quad z = 0
\end{align*}
\]

assuming that the wind stress is known. Taking into account the principle of mass conservation (\( \int u(z) \partial z = 0 \)), the analytical solution is

\[
u(z) = \frac{\tau_s}{4u_0 D} z (3z - 2D)
\]

If the surface velocity is known, instead of the stress, the boundary condition (6) changes to \( u(z) = u_s \), for \( z = D \), which leads to an alternative analytical solution to (5), given by

\[
u(z) = \frac{u_s}{D} z (3z - 2D)
\]

The two former solutions are based in the two possible surface boundary condition allowed for wind driven flows, that is, the velocity or the wind stress imposed at the surface. In general cases, based in field data, the wind velocity is known and the surface stress can be estimated by Eq. (4). If the flow is laminar, it is possible, using the former analytical solutions, determine the velocity that will impose the same flow pattern correspondent to a given stress. The relation between the velocity and stress for the same flow pattern is given by

\[
\tau_s = \frac{4 u_s}{D}
\]

or, taking the surface velocity friction definition,

\[
u_s = \left( \frac{\tau_s}{\mu} \right) = \left( \frac{4 u_s}{D \mu} \right)
\]
This analysis is very useful to verify the mathematical model behavior when simulating laminar flows. Although, as stated before, the nature of wind driven flows in closed water bodies is turbulent, this is a good problem for testing the implementation and solution procedures.

The physical domain of 10.0m length, 2.30m wide and 0.50 depth was discretized by a 80x7x20 equally spaced mesh. As the present flow is 2D, symmetry boundary conditions were employed for the “y” direction, so the volume numbers in this direction is not relevant. Refer to Fig. 1 for the other boundary conditions.

![Figure 1 - Two-dimensional flow in a closed channel](image)

A low Reynolds number was chosen for this experiment to guarantee the laminar behavior. It is defined as

\[
Re_s = \frac{\rho u_s D}{\mu} = 18.73
\]

(12)

Fig. 2 compares the vertical velocity profiles for the analytical solution, Eq. (8) and (9), against the numerical model. Both velocity and stress boundary conditions are compared and both show an excellent agreement.

![Figure 2 - Vertical velocity profile for laminar flow in a closed channel](image)

**TURBULENT FLOWS: MODEL VERIFICATION**

Tsantis [8] proposed a parabolic profile for the turbulent viscosity that leads to an analytical solution for the vertical velocity profile in 2D wind driven closed channels[9]. The solution proposed show good agreement with experimental measurements for these vertical velocity profiles. In this work this turbulent viscosity profile is implemented in to the numerical code as a algebraic turbulent model and the result is compared with experimental results in order to verify the accuracy of the numerical model.

The vertical turbulent viscosity profile, given by Tsantis [8] for 2D wind driven closed channels, is

\[
u_t = \frac{H_s}{\rho} \left( \frac{\lambda u_s}{D} \right) \left( z + z_b \right) \left( D - z + z_b \right)
\]

(13)

where \( u_s = (\tau_s / \rho)^{0.5} \) is the surface friction velocity and \( \lambda \) a constant that characterizes the turbulence intensity. \( \lambda \) is given accordingly to the Reynolds number range as

\[10^3 \leq \frac{\rho u_s D}{\mu} \leq 10^6 \quad 0.20 \leq \lambda \leq 0.50\]

The parameters \( z_0 \) and \( z_k \) in Eq. (13) are characteristics lengths determined at the bottom (\( z = 0 \)) and at the surface of the water body (\( z = D \)), respectively. They are a relative measure of the thickness of the viscous sublayers. Tsantis [10] found, through experimental studies, that their values should be taken as \( z_{sh} = z_0 / D = 2,20 \times 10^{-4} \) and \( z_{sh} = z_k / D = 0,60 \times 10^{-4} \).

Based on the former viscosity profile, Wu and Tsantis [9] found an analytical solution for the velocity profile, normalized by the surface shear velocity, given by

\[
\frac{u(z)}{u_s} = A \ln \left( 1 + \left( \frac{z}{z_b} \right) \right) + B \ln \left( 1 - \left( \frac{z}{z_b + D} \right) \right)
\]

(14)

where \( A = f_1 (z, z_b, z_{sh}) \) and \( B = f_1 (z, z_b, z_{sh}) \), as can be seen in [9]. In the experimental works of Baines and Knapp [11] and Wu and Tsantis [9], both stress and velocity are available as experimental results, so both can be specified as boundary conditions. Therefore, these experimental works are well suited to check the mathematical model proposed. Table 1 resumes the experimental data related to these works.

<table>
<thead>
<tr>
<th>Data</th>
<th>Symbol</th>
<th>Unit</th>
<th>Experiment A</th>
<th>Experiment B</th>
</tr>
</thead>
<tbody>
<tr>
<td>Depth</td>
<td>D</td>
<td>m</td>
<td>0.3048</td>
<td>0.3048</td>
</tr>
<tr>
<td>Wind velocity</td>
<td>( u_z )</td>
<td>m/s</td>
<td>3.901</td>
<td>6.906</td>
</tr>
<tr>
<td>Friction velocity at the surface</td>
<td>( u_s )</td>
<td>cm/s</td>
<td>0.6233</td>
<td>0.9416</td>
</tr>
<tr>
<td>Surface velocity</td>
<td>( u_n )</td>
<td>cm/s</td>
<td>10.72</td>
<td>15.25</td>
</tr>
<tr>
<td>Reynolds number</td>
<td>( R_e )</td>
<td>#</td>
<td>32700</td>
<td>46000</td>
</tr>
<tr>
<td>Normalized surface velocity</td>
<td>( u_s / u_t )</td>
<td>#</td>
<td>17.2</td>
<td>16.2</td>
</tr>
<tr>
<td>Equivalent roughness height</td>
<td>( z_{eq} )</td>
<td>mm</td>
<td>0.3521</td>
<td>0.4798</td>
</tr>
<tr>
<td>Height of zero velocity</td>
<td>( z_{_{eq}} D )</td>
<td>#</td>
<td>0.69</td>
<td>0.68</td>
</tr>
</tbody>
</table>

The boundary conditions imposed to the model can be seen in Fig. 1. A vertical profile for \( u_s \), following Eq. (13), using the friction velocity listed in Table 1 was used. The physical domain, as in the experiment, was \( L_x = 2.4 \) m, \( L_y = 0.72 \) and \( L_z = D = 0.3048 \). As stated before two kinds of boundary conditions may be imposed at the surface and bottom boundaries. Table 2 resumes these conditions.
Table 2 - Boundary conditions at surface and bottom

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Case A</th>
<th>Case B</th>
</tr>
</thead>
<tbody>
<tr>
<td>Surface</td>
<td>$u = u_s$</td>
<td>Stress: $u_s = u_s$ (engine)</td>
</tr>
<tr>
<td>Bottom</td>
<td>Velocity $u = 0$</td>
<td>Stress: $u_s = \text{Eq. (3)}$</td>
</tr>
</tbody>
</table>

The bottom friction velocity can be calculated once the equivalent roughness height is available as a experimental data (Jin, [5]) using Eq (3) with $z_h = Z_h$. During the numerical solution procedure this friction velocity must be updated because the velocity in Eq (3) is the one available in the last iteration step. When the solution is achieved, the velocity field will no longer change so the friction velocity. Based on previous laminar simulation an equally spaced mesh of $80 \times 7 \times 20$ volumes ($xyz$ directions) was employed.

The resulting vertical velocity profiles, using both boundary conditions for Experiment A, Table 1, $(Re = 32700)$, is shown in Fig. 3 where is also shown the analytical solution. For $\lambda = 0.35, 2.2 \times 10^4$, $1.4 \times 10^4$ respectively.

The resulting velocity profiles fit very well the analytical solution in the core of the channel. But near the boundaries the profiles are not in good agreement. The B case boundary condition is somewhat better near the surface but near the bottom the result is even worse. Taking $u_s/u_s$ as a parameter to compare the correctness of the solution near the surface the results are not so good as shown in Table 3.

![Figure 3](image)

Figure 3 - Velocity profiles for a closed channel flow (boundary condition A normalized by $u_s = 0.8893$, boundary condition B normalized by $u_s = 0.6233$).

<table>
<thead>
<tr>
<th>Boundary Condition</th>
<th>$u_s/u_s$</th>
<th>Cause</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>12.4</td>
<td>Surface stress not well solved by the numerical model</td>
</tr>
<tr>
<td>B</td>
<td>8.92</td>
<td>Velocity profile near the surface not well solved by the numerical model</td>
</tr>
<tr>
<td>Experimental</td>
<td>17.2</td>
<td>#</td>
</tr>
</tbody>
</table>

What we learn from these results is that if the focus is in the bulk flow, the simulations will provide an improved solution. However, if the focus is more than that, the solutions will be far from the experimental data. The main problem is that the wind driven flows, the boundary conditions, and the fineness of the mesh. The results in Table 3 show that the numerical solution is not a good estimate of the experimental data.

![Figure 4](image)

Figure 4 - Effects of the mesh resolution and distribution over the velocity profiles.

Table 4 - Values of $u_s/u_s$ for different meshes

<table>
<thead>
<tr>
<th>Mesh ($z$ direction)</th>
<th>$u_s/u_s$</th>
</tr>
</thead>
<tbody>
<tr>
<td>21 volumes equally spaced</td>
<td>12.01</td>
</tr>
<tr>
<td>41 volumes equally spaced</td>
<td>13.33</td>
</tr>
<tr>
<td>40 volumes, surface/bottom concentrated</td>
<td>16.18</td>
</tr>
<tr>
<td>Experimental</td>
<td>17.2</td>
</tr>
</tbody>
</table>

CONCLUSIONS

This paper presented part of a systematic procedure for evaluating a full three-dimensional turbulent model developed for predicting environmental flows with bottom effects. The comparisons made show that the model provides a reasonable estimate of experimental data and can be used as a tool for predicting such flows. However, further work is needed to improve the model and its application to more complex cases.
profile for two approaches of applying boundary conditions at the surface of water body and at the bottom, demonstrated that the treatment of the turbulence near the walls is producing physically realistic results. Although the results shown here report only 2D flows, the model developed can solve fully 3D flows in water bodies with complex shapes. Future papers will report the model evaluation in these situations.

![Graphs showing vertical velocities profiles](image)

Figure 5 - Vertical velocities profiles: numerical and experimental [11] results.

REFERENCES