2.5-D NUMERICAL MODELING ON FLOW AROUND A CYLINDER

B. Yulistiyanto

ABSTRACT

The equations for two-dimensional, horizontal flow are obtained by vertical depth-integration of the continuity and momentum equations. Consequently, the diffusion-dispersion terms are included, containing turbulent and "dispersion" stresses. A special emphasis was placed in elaborating the diffusion-dispersion terms. Turbulent stresses are expressed with the eddy viscosity concept; dispersion stresses are evaluated using velocity distributions along and across the curved streamlines. The differential equations are solved using MacCormack scheme. The numerical simulation was done for two runs; the expected alteration of the flow field around the cylinder is evident, notably the wake behind the cylinder.

For comparison, two similar runs were performed in a laboratory channel. The velocity vectors upstream from the cylinder and along its sides are in reasonably good agreement. However, very close to the cylinder, the simulation sometimes underpredicts the velocities. Downstream from the cylinder, agreement is satisfactory both inside and outside the wake. The flow depths at the center line both upstream and downstream from the cylinder are also in good agreement, however along the cylinder circumference, the simulated flow depths are lower than the observed ones.

INTRODUCTION

Flow around a cylinder positioned normal to the fixed-bed channel bottom represents a classical problem of fluvial hydraulics. It serves as a pre-study to flow over a moveable-bed bottom, which in turn is of importance to determine the local scour around the cylinder.

The objective of the present study is to gain a better understanding of the flow fields around the cylinder. This can be obtained either by mathematical modeling the flow fields and/or by performing extensive measurements around the cylinder.

Since the flow is highly three-dimensional, a three dimensional model would be the best; this is difficult task. We have opted for developing a 2.5-dimensional model by depth-averaging the velocity components and taking into account the depth-averaged turbulent stresses and the dispersion stresses. In the latter stresses, longitudinal and transversal velocity profiles are considered.

The governing Saint-Venant 2-D equations are derived from the Navier-Stokes equations. Of particular interest here are terms of the depth-averaged effective stresses. Similar forms of these equations have been presented—without given the derivation—before by Vreugdenhil, 1973, Fivmeister ,1977, and Rodi, 1984, Khan and Chaudhry, 1992. This set of equations has to our knowledge—never been applied to the geometrical situation of our problem.

In order to solve this system of equation, we have opted for the explicit finite-difference MacCormack scheme using forward and backward finite-differences. These types of boundary conditions are imposed: mirror conditions along the inflow and downstream stage. The latter conditions are based on the characteristic form of the flow equations. The MacCormack scheme has been successfully used for different kind of flow with free surface (Fennema, R. J. and Chaudhry, M. H., 1986, Ballamuri, S. M. and Chaudhry, M. H., 1992).

An extensive measurement program was also launched (Yulistiyanto, 1997), but only the depth-averaged values are here used for our present study. No similar measurement—our knowledge—are available in the literature.

GOVERNING EQUATIONS

The shallow-water two-dimensional equations are obtained by time averaging the Navier-Stokes equations and integrating these over the flow depth with some assumptions: (1) vertical velocity and acceleration are negligible; (2) pressure distribution is hydrostatic; (3) depth-averaged values are sufficient to describe properties which vary over flow depth; (4) the bottom slopes are small in both directions. The continuity and momentum equations (Khan and Chaudhry, 1992; Vreugdenhil, 1994; Yulistiyanto, 1997) are the following:

\[ \frac{\partial h}{\partial t} + \frac{\partial (hu)}{\partial x} + \frac{\partial (hv)}{\partial y} = 0 \]  

(1)

\[ \frac{\partial (hu)}{\partial t} + \frac{\partial (hu^2 + \frac{1}{2}gh^2)}{\partial x} + \frac{\partial (hv)}{\partial y} = gh(S_0 - S_y) + \frac{\partial}{\partial x} \left( \frac{h}{\rho} \right) T_x + \frac{\partial}{\partial y} \left( \frac{h}{\rho} \right) T_y \]  

(2)

\[ \frac{\partial (hv)}{\partial t} + \frac{\partial (hu^2 + \frac{1}{2}gh^2)}{\partial x} + \frac{\partial (hv)}{\partial y} = gh(S_0 - S_y) + \frac{\partial}{\partial x} \left( \frac{h}{\rho} \right) T_x + \frac{\partial}{\partial y} \left( \frac{h}{\rho} \right) T_y \]  

(3)
where \( h \) is the flow depth and \( g \) is the acceleration of gravity; \( U \) and \( V \) are the depth-averaged velocity components, defined as:

\[
U = \frac{1}{h} \int_{h}^{0} \bar{u}(z) \, dz \quad \text{and} \quad V = \frac{1}{h} \int_{h}^{0} \bar{v}(z) \, dz
\]

where \( \bar{u}(z) \) and \( \bar{v}(z) \) are the local time-averaged velocity components. The depth-averaged effective stresses in vertical planes are defined as:

\[
T_u = \frac{1}{h^2} \rho \left[ 2 \frac{\partial \bar{u}}{\partial x} \mathbf{u}' \mathbf{u}' - U \mathbf{u}' \mathbf{u}' \right] \, dx
\]

\[
T_v = T_u = \frac{1}{h^2} \rho \left[ 2 \frac{\partial \bar{v}}{\partial y} \mathbf{v}' \mathbf{v}' - U \mathbf{v}' \mathbf{v}' \right] \, dy
\]

\[
T_w = \frac{1}{h^2} \rho \left[ 2 \frac{\partial \bar{v}}{\partial z} \mathbf{v}' \mathbf{v}' - V \mathbf{v}' \mathbf{v}' \right] \, dz
\]

where \( \rho \) and \( \nu \) are respectively the density and the kinematic viscosity of the fluid; \( \mathbf{u}' \) and \( \mathbf{v}' \) are the turbulent velocity fluctuations; the local time-averaged velocity components \( \bar{u}(z) \) and \( \bar{v}(z) \), are distributed over the flow depth; the deviations from the depth-averaged value, eqs. (4) are expressed as:

\[
U' = (\bar{u} - U) \quad \text{and} \quad V' = (\bar{v} - V)
\]

and correspond to dispersion stresses.

**DIFFUSION-DISPERSION TERMS**

Diffusion-Dispersion terms consist of three types of effective stresses, namely molecular viscosity, turbulent and "dispersion" stresses. At sufficiently high Reynolds number, the dispersion stresses can be neglected compared to the turbulent stresses. Using the Boussinesq's eddy-viscosity concept, the turbulent stresses are assumed to be proportional to the mean-velocity gradients. They may be expressed as (Yulistiyanto, et al., 1998):

\[
T_{u} = \frac{1}{h} \rho \nu \frac{\partial U}{\partial x} \, dx \quad T_{v} = \frac{1}{h} \rho \nu \frac{\partial V}{\partial y} \, dy \quad T_{w} = \frac{1}{h} \rho \nu \frac{\partial W}{\partial z} \, dz
\]

The "dispersion" stresses appear from the depth-integration of the advective momenta balance and (Fokken 1977) are
The dispersion coefficients $K_x$, $K_y$, $K_z$, and $K_{xy}$ are related to the streamline-oriented coefficients $K_{x0}$, $K_{y0}$, $K_{z0}$, and $K_{xy0}$ by the following transformation (Yulistianto, 1997):

\begin{align}
K_x &= K_{x0} \cos \theta - K_{y0} \sin \theta \cos \theta - K_{z0} \sin \theta \sin \theta + K_{xy0} \sin \theta^2 \\
K_y &= K_{y0} \sin \theta \cos \theta - K_{y0} \sin \theta \cos \theta - K_{z0} \sin \theta \sin \theta + K_{xy0} \sin \theta^2 \\
K_z &= K_{z0} \sin^2 \theta + K_{y0} \sin \theta \cos \theta - K_{z0} \sin \theta \sin \theta + K_{xy0} \sin \theta^2 \\
K_{xy} &= K_{xy0} \sin^2 \theta + K_{y0} \sin \theta \cos \theta + K_{z0} \sin \theta \sin \theta + K_{xy0} \sin \theta^2
\end{align}

where $\theta$ is the angle between the streamline direction and the $x$-coordinate.

For use of eqs. (13), assumptions about the velocity profiles are needed. The velocity along the streamline is assumed to be given by a logarithmic distribution, such as (Rozovskii, 1957, p.33):

$$
\bar{u}(\eta) = U_\infty + \frac{U'}{\kappa} (1 + \ln \eta)
$$

where $\eta = z/h$ is the relative depth, $\kappa$ is the von Karman constant and $U_\infty$ is the friction velocity. Using this mean velocity distribution and assuming that both components vanish at the bottom, Rozovskii (1957, p.41 and p.54) derived a formula for the transverse velocity profile for a smooth bed:

$$
\bar{v}_e(\eta) = \frac{h}{\kappa^2} \frac{U_\infty}{r} \left[ F_1(\eta) - \frac{U'}{\kappa U_\infty} F_2(\eta) \right]
$$

and for a rough bed:

$$
\bar{v}_e(\eta) = \frac{h}{\kappa^2} \frac{U_\infty}{r} \left[ F_1(\eta) - \frac{\eta}{\kappa U_\infty} F_2(\eta) + 0.8(1 + \ln \eta) \right]
$$

where $F_1(\eta)$ and $F_2(\eta)$ are functions of the relative depth $\eta$:

$$
F_1(\eta) = \frac{2 \ln(\eta)}{\eta - 1} \Delta \eta \quad \text{and} \quad F_2(\eta) = \frac{\ln^2(\eta)}{\eta - 1} \Delta \eta
$$

The radius of the mean streamline $r$, can be obtained from the known values of $U$ and $V$, since the curvature is given by:

$$
\frac{1}{r} \left( \frac{d^2y}{d\eta^2} \right)^2 \frac{d(y/\eta)}{dy} = \frac{d(an\theta)}{dx}
$$

It is now possible to write the lateral momentum equation, corresponding to the two last terms of eqs. (2) and (3):

\begin{align}
\frac{\delta}{\delta \rho} \left[ \frac{h}{\rho} \frac{\rho}{\rho} \right] &+ \frac{\delta}{\delta \rho} \left[ \frac{h}{\rho} \frac{\rho}{\rho} \right] = \frac{\delta}{\delta \rho} \left[ (\bar{V} - K_{xy0} + K_{xy0}) h \frac{\rho}{\rho} \right] \\
&+ \frac{\delta}{\delta \rho} \left[ (\bar{V} - K_{xy0} + K_{xy0}) h \frac{\rho}{\rho} \right] + \frac{\delta}{\delta \rho} \left[ (\bar{V} - K_{xy0} + K_{xy0}) h \frac{\rho}{\rho} \right]
\end{align}

**FINITE-DIFFERENCE APPROXIMATION**

The system of differential equations (1)-(3) is too complex to be solved by analytical predictor-corrector scheme (Fletcher 1988, p.404).

The scheme is second-order accurate in space and time and is also able to capture shocks (Ballamudi and Choudhry, 1992). The schematic definition of this discretization scheme and the finite-difference approximations for the first- and second-order partial derivatives are given elsewhere (Yulistianto, 1997).

Forward finite-differences, using the known variables, are used for approximating the spatial partial differential terms in the predictor step while backward finite-differences, using the predicted variables, are used in the corrector step. Forward and backward finite-differences are now exchanged for the next predictor-corrector cycle to avoid the directional bias of the scheme.

For the steady flow, these processes are repeated until the convergence to a steady state condition is reached.

With some simplifications, it was demonstrated (Yulistianto, 1997) that the MacCormack scheme for the 2-D convection-diffusion equation is stable, if the following condition is satisfied:
\[ \Delta t < C_n \left( \frac{1}{\Delta x} \left[ U \sqrt{\frac{\rho}{\mu}} + \sqrt{\frac{\sigma}{\mu}} \right] + \frac{1}{\Delta y} \left[ V \sqrt{\frac{\rho}{\mu}} + \sqrt{\frac{\sigma}{\mu}} \right] \right) \]

where \( C_n \) is the desired Courant number, which should be: \( C_n \leq 1.0 \).

**BOUNDARY CONDITIONS**

The initial condition are defined according to uniform flow: the corresponding \( U, V, \) and \( h \) values are imposed at all grid points in the initial time step. In this present study, three types of boundaries are encountered: the wall boundaries, the upstream and the downstream (open) boundary. The finite-difference scheme is valid for the interior grid points. At the boundaries, some terms of these equations become indefinite. Therefore, to determine conditions at the boundaries, the mirror conditions (Roache, 1972) are used. The free-slip condition is used along the walls. The direction of the flow is determined in such a way that the normal velocity of each point at the wall is zero. The upstream or inflow boundary is an open boundary above the flow enters the computational domain. Two-directional discharges, \( Q_x, Q_y \) and \( Q_\theta, Q_r \), are given and the other parameter, \( h \), is found by solving the difference equation based on the characteristic form of the flow equations. In the present simulation, the transverse velocity is prescribed equal to zero and the discharge is constant with the time; the flow depth is calculated by the characteristic boundary procedure (Streeter, 1971, pp.582-690, Graf and Altinakar, 1996). Since a fixed grid is being considered, the method of characteristics needs an interpolation procedure. The downstream condition generally imposes the water depth \( h = h(f) = h_{down} \) and is handled in a similar way.

**NUMERICAL RESULTS**

The numerical simulation was done for two test runs; the hydraulic parameters are given in Table 1 and Fig. 1, where the symbols are explained. On the same figure is also shown the limited domains, used for the grid visualization and for the velocity visualization, according to Fig. 2.

---

**Table 1 Uniform flow variables and channel parameters.**

<table>
<thead>
<tr>
<th>Test</th>
<th>( n ) [m]</th>
<th>( B_D ) [ft]</th>
<th>( n ) [m]</th>
<th>( Q ) [cfs]</th>
<th>( U ) [fps]</th>
<th>( V ) [fps]</th>
<th>( h ) [ft]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.22</td>
<td>1.48</td>
<td>2( \sqrt{0.40} )</td>
<td>0.346</td>
<td>0.012</td>
<td>0.67</td>
<td>0.000</td>
</tr>
<tr>
<td>2</td>
<td>0.22</td>
<td>0.95</td>
<td>2( \sqrt{0.40} )</td>
<td>0.149</td>
<td>0.012</td>
<td>0.43</td>
<td>0.00</td>
</tr>
</tbody>
</table>

The computation was carried out with a non-uniform grid (see Fig. 2). The size of the grid is smaller close to the cylinder, in order to get more information on flow near the cylinder. The cell numbers of 80 in the \( x \)-direction and 60 in the \( y \)-direction were used in the computation. Distance intervals, \( \Delta x \) and \( \Delta y \), have a minimum and a maximum value of 6.6 mm and 0.66 mm for test 1 and test 2, respectively. The time interval, \( \Delta t \), is limited by the stability of the scheme (21), and is restricted by the Courant number, which was set for all the test equal to \( C_n = 0.667 \) (another Courant number, \( C_n = 0.85 \), was also used in the numerical simulation; the scheme remained stable).
The results of the numerical simulations of the flow field are given in Fig. 3 for test 1, and in Fig. 4 for test 2. The boundary conditions at the channel inlet and outlet are situated far enough from the cylinder, thus they do not influence the condition of the flow around the cylinder. The inflow discharge, $Q_i$, is maintained constant with time. The water depth at the channel outlet, $h_{down}$, is also maintained constant. Along the wall boundaries, consisting of two vertical side walls and the perimeter of the cylinder, the normal velocity is zero with a perfect slip condition.

The governing equations (1-3) describe unsteady flow, however by applying steady conditions at both upstream and downstream boundaries, a convergence to steady state is obtained. The simulation reaches this steady state after 5 sec simulated time from the beginning of the computation. Simulations were stopped at $t = 4.2$ sec for test 1 and at $t = 4.4$ sec for test 2.

The relative importance of each term in the momentum equations (2-3) was also investigated (Yuliantyanto, 1997, Appendix E). It provided interesting detailed information; the inertia terms are of dominating importance; the slope terms are insignificant; the diffusion-dispersion terms take important values.

The numerical simulation allowed also to obtain information about the change in the flow depth, $h(x, y)$, due to the presence of the cylinder (Figs. 5). The typical bow wave, having the highest upwelling in front of the cylinder — i.e., $h_1 - h_0$ at $y/2$ — is observed. Its level decreased considerably going around the cylinder, becoming the lowest depression at the rear of the cylinder.

Conclusively, it can be said, that the depth-averaged simulation of the flow around a cylinder gives results which are to be considered reasonable in the light of the existing literature.
COMPARISON WITH EXPERIMENTAL RESULTS

In order to check the numerical simulations, two runs (test 1 and test 2) were performed (Yuliastianto, 1997). The velocity measurements are done with an Acoustic Doppler Velocity Profiler (ADVP). The vertical distribution of the velocity in all three directions was measured, however only velocities in the longitudinal, u, and transversal, v, directions are used in the comparison with the numerical simulation. These measurements are done along 5 lines radiating from the cylinder, under the angles of 0°, 45°, 90°, 135°, and 180° for test 1, and under the angles of 0°, 45°, 90°, 135°, and 180° for test 2; for each line, these measurements were performed at 8 to 12 positions. At each of them, the flow depth was also measured. From the vertical mean-velocity distributions, the depth-averaged velocities, \( U \) and \( U' \), in the flow field around the cylinder were then calculated. These data are used to verify the applicability of the numerical model.

The velocity vectors resulting from the numerical simulation for test 1 and test 2 at steady state, together with the depth-averaged velocities from the experimental works, are shown in Figs. 6 and 7. The velocity vectors upstream from the cylinder and along the sides are in reasonably good agreement. Nevertheless very close to the cylinder, the simulation sometimes underpredicts the velocities. Downstream from the cylinder, where the flow structure is complex (see Figs. 3 and 4), the agreement is satisfactory both inside and outside the cylinder wake.
CONCLUSION

The Saint-Venant equations (1-2) for two-dimensional depth-averaged flow, including exchanges through vertical planes, were used to simulate flow around a cylinder positioned in an open channel. A special emphasis was placed in elaborating the diffusion-dispersion terms.

Numerical solutions were obtained using an explicit finite-difference scheme according to MacCormack, with appropriate boundary conditions. Two different cases, (see Table 1), were simulated, from which resulting flow fields are given in Figures 3 and 4. The expected flow alteration due to the presence of the cylinder is evident, notably the wake behind the cylinder. The simulated topography of the flow depths, (see Fig.5), showed clearly the existence of a bow-wave and the subsequent depression in the vicinity of the cylinder.

Laboratory experiments were performed in parallel for two cases. The comparison given in Fig. 6 and Fig. 7, between experimental data and simulated results is considered as being reasonably satisfactory, both for the flow fields and for the elevations. This shows that 2-D finite-difference modeling, including lateral momentum exchanges and dispersion terms without any parameter to be calibrated, is able to approach a relatively complex situation.

ACKNOWLEDGEMENTS

The author acknowledges the contributions from Prof. Dr. Yves Zech of UCL-Belgium, and from Prof. Dr. W. Graf of EPFL Lausanne-Switzerland, for their constructive advice.

REFERENCES


