Finite volume modelling of free surface draining vortices

F. Trivellato\textsuperscript{a,∗}, E. Bertolazzi\textsuperscript{a}, B. Firmani\textsuperscript{b}

\textsuperscript{a}Dipartimento di Ingegneria Civile ed Ambientale, Università degli Studi di Trento, via Mesiano 77, I-38050 Trento, Italy
\textsuperscript{b}Dipartimento di Ingegneria Meccanica e Strutturale, Università degli Studi di Trento, via Mesiano 77, I-38050 Trento, Italy

Received 4 November 1998

Abstract

The phenomenon of the free-surface vortex forming over a draining intake is well known, together with its detrimental effects. While analytical solutions have been helpful in clarifying some features of the phenomenon, no extensions have been readily provided in solving instances of practical importance. Therefore, efficient anti-vortex devices have been traditionally conceived by means of physical model studies. However, a numerical simulation of the whole flow field would be nowadays desirable. The proposed numerical solution of the flow field is based on an axial-symmetric finite volume model, which solves the incompressible Navier–Stokes equations on irregular geometries. Boundary conditions include both the Dirichlet and the Neumann type. The mesh is staggered. The numerical scheme is a semi-implicit one, where the terms controlling the diffusion and those controlling the pressure field are discretized implicitly, while the convective terms are approximated via an Euler–Lagrange approach. The discrete version of the continuity equation becomes, by a substitution, a system having the pressure values as the only unknowns. The solution proceeds via an iterative scheme, which solves a symmetric and semi-positive-definite system for the pressure, by a standard pre-conditioned conjugate gradient method. The discrete velocity field at each iteration can then be explicitly obtained. The numerical solution has been verified by the laboratory experimental data obtained by Daggett and Keulegan (1974). This comparison demonstrated that the proposed numerical model is capable of predicting the whole steady flow field. Of special value is the comparison with the radial velocity distribution, which has a typical jet-like profile along the vertical direction. According to the most recent experimental evidence, it seems that the very onset of the vortex can be traced to this special feature of the radial velocity profile. © 1999 Elsevier Science B.V. All rights reserved.

Keywords: Incompressible Navier–Stokes; Finite volume

1. Introduction

Intake draining vortices are free surface vortices forming over a hole discharging from a reservoir (Fig. 1). They occur typically whenever the submergence (or head of water) is less than a critical value.

∗ Corresponding author.

0377-0427/99 – see front matter © 1999 Elsevier Science B.V. All rights reserved.
Pii: S0377-0427(98)00251-9
The knowledge of the critical submergence is of the outmost relevance to the hydraulic engineer, in order to avoid the negative effects due to the vortex formation (such as, for instance, reduction of the discharge coefficient, air entrainment, ingestion of floating debris and, at times, elimination of a serious safety hazard).

Several approaches have been presented in the literature to deal with the problem: that is, to determine whether or not an intake is prone to vortex formation. Basically, these approaches can be labeled as analytical models, physical models and, recently, numerical models.

Many analytical approaches have been presented in the literature in order to attain a theoretical view of the far-field velocity (see, for instance, [7, 8, 13, 14]); still, the flow representation has defied so far any comprehensive analytical analysis. In fact, no mathematical model capable of representing the structure of the flow field in the free surface vortex has obtained so far general consensus, so that a certain degree of confusion arises when reviewing the literature, which covers from refined theories (yet unpractical or without a sounding physical basis), to a model as simple as the well-known Rankine vortex.

Besides of being somewhat laborious and unpractical, the known analytical theories apply only to a limited range of vortices and have weak experimental validation.

Anwar [1], proposed an ingenious solution based on the method of separation of variables, which depended however on a couple of oversimplifying assumptions.

The structure of the flow pattern in the far-field domain cannot be tackled by similarity analysis: the attempts conducted so far have not lead to any useful results and the problem does not seem to admit a similarity variable, either of the form $\eta = r^a z^b$ ($a$ and $b$ any constant) or, more in general, of the form $\eta = f(r)/g(z)$. As for the conformal transformation technique, it would apply only to a 2-D flow domain.
A simple, yet effective method for the representation of the far-field flow, was devised by Cola and Trivellato [5]. The method is based on potential flow theory by adopting a suitable arrangement of image sinks. The comparison with the Daggett and Keulegan [6] experimental data was fairly accurate up to a distance as close to the intake as \( r \approx 1.7 R_0 \), and demonstrated that the bottom boundary layer can be safely neglected in calculations.

Analytical models (a review of which can be found in [5, 21, 22]), have all been derived in steady state and in oversimplified fluid domains; therefore the results obtained turned out to be of uncertain value to model real intakes whose forebay geometry, as a matter of fact, is 3-D.

On the contrary, physical modelling was much more helpful, not only in designing intakes not affected by vortex formation but also in conceiving proper anti-vortex devices. Physical modelling has been, and is still nowadays, the best approach to tackle the problem of vortex prevention; however, physical models are expensive and time consuming. In fact, many hydraulic projects of minor importance, cannot afford the cost and the delay of a physical model study. As a consequence, it appears that a numerical solution would be nowadays most attractive and by far more convenient.

As for numerical methods, however, surprisingly very few have been developed for applications that could be regarded of sufficiently large validity.

Brocard et al. [3] used the finite element method to solve the 2-D depth integrated conservation of mass and momentum equations. The numerical model was used on an actual intake and verified versus experimental measurements. Once the velocities are known at any time, the circulation may be calculated along a closed path around the intake. Even though the calculated overall flow pattern appeared reasonable, the comparison with the actual circulation measured in the physical model was not considered satisfactory by the authors.

A different approach was used by Trivellato and Ferrari [23], who implemented the Casulli and Cheng [4]’s finite difference model, which is based on the shallow water approximation (hydrostatic pressure along the vertical). The knowledge of the flow field induced by the vortex was achieved by solving numerically the Navier–Stokes equations, adopting an Euler–Lagrange approach. The model was checked versus the only useful measurements of radial and tangential velocities which are known from the literature [6]. The above model demonstrated that it was capable of simulating the two most distinctive features of the vortex flow field, i.e., the azimuthal motion and the radial jet. The latter occurrence is typical in free surface vortices in an infinite flow field.

The radial jet is the lowermost region of the flow field where strong radial velocities concentrate near the bottom boundary; this jet provides basically most of the fluid discharge, since the uppermost fluid is mainly involved in the azimuthal pattern. The radial jet has to be viewed as a significant departure from the usual assumptions that were considered in previous theoretical analysis.

High transversal gradients are active in this region, where vorticity production has been visualized; part of this vorticity is convected towards the symmetry axis, where it undergoes a stretching process by the axial component of velocity, giving rise to the vortex. As the water level subsides, the vortex can reach the free surface, becoming eventually visible from outside [20]. The radial jet is therefore essential in explaining the very origin of the vortex. In addition, the above finite difference model showed a range of applicability of the shallow water hypothesis, that is larger than it could be initially suspected.

However, despite these satisfactory results, the finite difference model of Trivellato and Ferrari [23] suffered from two main disadvantages: first of all, the shallow water approximation is clearly an
over-simplification in actual intakes; secondly, the finite difference model is notoriously unsuitable in dealing with complicated fluid domains.

Simulating complicated geometries, which are appropriate for actual 3-D intakes, is the main motivation of the present work. The removal of the two limitations described above by using a finite volume model is the aim of this study.

2. Formulation of the problem

As a first step towards a full 3-D discretization and in view of the comparison which will be performed with the experimental data of Daggett and Keulegan [6], the incompressible Navier–Stokes equations have been considered in cylindrical coordinates and axial symmetry

\[
\rho \left( D_t u - \frac{v^2}{r} \right) = - \frac{\partial p}{\partial r} + \mu \left( \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial v}{\partial r} \right) - \frac{u}{r^2} + \frac{\partial^2 u}{\partial z^2} \right), \tag{1}
\]

\[
\rho \left( D_t v + \frac{uv}{r} \right) = \mu \left( \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial v}{\partial r} \right) - \frac{v}{r^2} + \frac{\partial^2 v}{\partial z^2} \right), \tag{2}
\]

\[
\rho D_t w = - \frac{\partial p}{\partial z} - \rho g + \mu \left( \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial w}{\partial r} \right) + \frac{\partial^2 w}{\partial z^2} \right), \tag{3}
\]

\[
\frac{\partial}{\partial r} (ru) + \frac{\partial}{\partial z} (rv) = 0, \tag{4}
\]

where

\[ D_t f = \frac{\partial f}{\partial t} + u \frac{\partial f}{\partial r} + w \frac{\partial f}{\partial z}, \]

\[ u(t, r, z) \] is the radial velocity; \[ v(t, r, z) \] is the tangential velocity; \[ w(t, r, z) \] is the vertical velocity; \[ p(t, r, z) \] is the pressure; \[ \rho \] is the mass density; \[ \mu \] is the dynamic viscosity and \[ g \] is the gravity acceleration.

The equations are assumed to be valid in the domain \( \Omega = [0, R] \times [0, H] \). Boundary conditions for the dependent variables include both the Dirichlet and Neumann type:

- for \( 0 \leq r \leq R_0 \): \( \hat{u}(t, r, 0)/\hat{\partial} z = \hat{v}(t, r, 0)/\hat{\partial} z = 0 \) and \( w(t, r, 0) = w^{\text{out}} \). Where \( R_0 \) is the radius of the hole and \( w^{\text{out}} \) is the outlet velocity,
- for \( 0 \leq r \leq R_0 \): \( \hat{u}(t, r, r)/\hat{\partial} z = \hat{v}(t, r, r)/\hat{\partial} z = 0 \) and \( w(t, r, 0) = 0 \),
- for \( 0 \leq t \leq R_0 \): \( \hat{u}(t, r, H)/\hat{\partial} z = \hat{v}(t, r, H)/\hat{\partial} z = \hat{w}(t, r, H)/\hat{\partial} z = 0 \),
- for \( 0 \leq z \leq H \) we impose \( u(t, 0, z) = v(t, 0, z) = w(t, 0, z)/\hat{\partial} r = 0 \), and \( u(t, R, z) = v(t, R, z) = v^{\text{in}}, w(t, R, z) = 0 \).

2.1. Semi-Lagrangian approximation of \( D_t \)

In what follows, we use the definition

\[
D_t f(t^0, r^0, z^0) = \lim_{t \to t^0} \frac{f(t^0, r^0, z^0) - f(t(t), z(t))}{t^0 - t}, \tag{5}
\]
where \((r(t), z(t))\) stands for the position at time \(t\) of the particle that has passed through \((r^0, z^0)\) at the initial time \(t^0\). The semi-Lagrangian approximation is based upon a straightforward approximation of (5). Choosing a time step \(\Delta t\), a first-order approximation of the above derivative can be written as

\[
D_t f(r(t), z(t)) \approx \frac{f(t^0, r^0, z^0) - f(t^0 - \Delta t, r(t^0 - \Delta t), z(t^0 - \Delta t))}{\Delta t}.
\]

(6)

A second-order approximation can be constructed as

\[
D_t f \approx \frac{1.5 f - 2\bar{f} + 0.5\bar{\bar{f}}}{\Delta t},
\]

(7)

where

\[
\bar{f}(t^0, r^0, z^0) = f(t^0 - \Delta t, r(t^0 - \Delta t), z(t^0 - \Delta t)),
\]

\[
\bar{\bar{f}}(t^0, r^0, z^0) = f(t^0 - 2\Delta t, r(t^0 - 2\Delta t), z(t^0 - 2\Delta t)).
\]

3. The finite volume discretization

A finite volume scheme has been adopted to approximate the solution of Eqs. (1)–(4). To construct the finite volume partition, the intervals \([0, R]\) and \([0, H]\) are as follows:

\[
0 < r_1 < r_2 < \cdots < r_{N_r - 1} < r_{N_r} < R,
\]

\[
0 < z_1 < z_2 < \cdots < z_{N_z - 1} < z_{N_z} < H.
\]

Also the following holds:

\[
r_{1/2} = 0, \quad r_{N_r + 1/2} = R, \quad r_{i+1/2} = \frac{r_i + r_{i+1}}{2}, \quad i = 1, 2, \ldots, N_r - 1,
\]

\[
z_{1/2} = 0, \quad z_{N_z + 1/2} = R, \quad z_{j+1/2} = \frac{z_j + z_{j+1}}{2}, \quad j = 1, 2, \ldots, N_z - 1,
\]

\[
\Delta r_i = r_{i+1/2} - r_{i-1/2}, \quad \Delta r_{N_r+1/2} = r_{N_r+1} - r_1,
\]

\[
\Delta z_j = z_{j+1/2} - z_{j-1/2}, \quad \Delta z_{N_z+1/2} = z_{N_z+1} - z_1.
\]

Radial and tangential velocities are evaluated at the nodal points \((r_{i+1/2}, z_j)\), while vertical velocities are assumed to be known at the nodal points \((r_i, z_{j+1/2})\). The pressure is assumed to be known at the nodal points \((r_i, z_j)\). The following set defines \(V_j\):

\[
V_j = \{(x, y) \mid x_{i-1/2} \leq x \leq x_{i+1/2}, y_{j-1/2} \leq y \leq y_{j+1/2}\}.
\]

Similar volumes are defined on half-integers such as \(V_{i+1/2,j}\).

To construct a finite volume approximation, Eqs. (1) and (3) are integrated over a control volume \(V_{i+1/2,j}\), (2) over a control volume \(V_{i,j+1/2}\), and finally (4) over a generic control volume \(V_j\). Next the volume integrals are approximated by using the values at the nodal points.
3.1. Semi-discrete scheme for the velocities $u$, $v$ and $w$

After the approximation of the volume integral, the following semi-discrete system is obtained:

$$
D_t u_{i+1/2} = h_{i+1/2}(v_{i+1/2})^2 - k_{i+1/2}u_{i+1/2} - \frac{p_{i+1} - p_{ij}}{\rho \Delta r_{i+1/2}} + \frac{\mu}{\rho} \mathcal{D}\{u_{i+1/2}\},
$$

$$
D_t v_{i+1/2} = -h_{i+1/2}u_{i+1/2}v_{i+1/2} - k_{i+1/2}v_{i+1/2} + \frac{\mu}{\rho} \mathcal{D}\{v_{i+1/2}\},
$$

$$
D_t w_{ij+1/2} = -g - \frac{p_{ij+1} - p_{ij}}{\rho \Delta z_{j+1/2}} + \frac{\mu}{\rho} \mathcal{D}\{w_{ij+1/2}\},
$$

(8)

where

$$
k_{i+1/2} = \frac{\mu}{\rho r_{i+1}}, \quad h_{i+1/2} = \frac{1}{r_{i+1/2}},
$$

$$
\mathcal{D}\{U_{ij}\} = \frac{1}{r_{i+1} \Delta r_i} \left( r_{i+1/2} \frac{U_{i+1} - U_{ij}}{\Delta r_{i+1/2}} - r_{i-1/2} \frac{U_{ij} - U_{i-1,j}}{\Delta r_{i-1/2}} \right),
$$

$$
+ \frac{1}{\Delta z_j} \left( \frac{U_{ij+1} - U_{ij}}{\Delta z_{j+1/2}} - \frac{U_{ij} - U_{ij-1}}{\Delta z_{j-1/2}} \right).
$$

4. The discrete system

The temporal discretization of system (8) has then been performed. In particular, the $D_t$ derivative has been approximated using a semi-Lagrangian approach with approximation formula (6) or (7), if a second-order approximation is wanted. The diffusion terms are considered implicitly. Finally, inertial terms, coupling $u$ and $v$ are discretized semi-implicitly.

The Euler–Lagrange scheme uses bilinear interpolation with a second-order Runge–Kutta scheme (Collatz scheme) for numerical integration. The resulting method is as follows:

$$
(\gamma + \Delta t k_{i+1/2}) u_{i+1/2}^{n+1} = F_{i+1/2}^{n+1} + \Delta t h_{i+1/2} v_{i+1/2}^{n+1} - \Delta t \frac{p_{i+1}^{n+1} - p_{ij}^{n+1}}{\rho \Delta r_{i+1/2}},
$$

(9)

$$
(\gamma + \Delta t k_{i+1/2}) v_{i+1/2}^{n+1} = G_{i+1/2}^{n+1} - \Delta t h_{i+1/2} v_{i+1/2}^{n+1} - \Delta t \frac{p_{ij}^{n+1}}{\rho \Delta r_{i+1/2}},
$$

(10)

$$
\gamma w_{ij+1}^{n+1} = H_{ij+1}^{n+1} - \Delta t \frac{p_{ij+1}^{n+1}}{\rho \Delta z_{j+1/2}},
$$

(11)

$$
\Delta z_j (r_{i+1/2} w_{ij+1}^{n+1} - r_{i-1/2} w_{ij-1/2}^{n+1}) + r_n \Delta r (w_{ij+1/2}^{n+1} - w_{ij-1/2}^{n+1}) = 0,
$$

(12)

where the temporal index $n + (L)$ stands for an approximation of the $n + 1$th time value. In Eqs. (9)–(11), $\gamma = 1$ or 1.5, respectively, if the Lagrangian step formula (6) or (7) is used.
Moreover,

\[ F^{n+1}_{i+1/2,j} = \mathcal{L}\{u^{n+1}_{i+1/2,j}\} + \frac{\Delta t \mu}{\rho} \mathcal{D}\{u^n_{i+1/2,j}\}, \]
\[ G^{n+1}_{i+1/2,j} = \mathcal{L}\{v^{n+1}_{i+1/2,j}\} + \frac{\Delta t \mu}{\rho} \mathcal{D}\{v^n_{i+1/2,j}\}, \]
\[ H^{n+1}_{ij+1/2} = \mathcal{L}\{w^n_{ij+1/2}\} + \frac{\Delta t \mu}{\rho} \mathcal{D}\{w^n_{ij+1/2}\} - g \Delta t, \]

where the semi-Lagrangian discrete operator is defined as: \( \mathcal{L}\{u_{i,j}\} = u^n_{i-z,j-\beta}. \)

### 4.1. Solution of the discrete system

To solve (9)-(12), an iterative procedure has been implemented. Let \( f^{n+1/2} \) be the approximation of \( f^{n+1} \) by the \( \gamma \)th iterate. Eq. (10) suggests an approximation of \( v^n_{i+1/2,j} \) as follows:

\[ v^{n+1/2}_{i+1/2,j} = \frac{G^{n+1}_{i+1/2,j} - \Delta th_i v^{n+1}_{i+1/2,j} u^{n+1}_{i+1/2,j}}{\gamma + \Delta k_{i+1/2}}, \tag{13} \]

The substitution of Eq. (13) into (9), suggests an approximation of \( u^{n+1}_{i+1/2,j} \) as follows:

\[ u^{n+1}_{i+1/2,j} = M^{n+1}_{i+1/2,j} - \Delta t \left( \frac{p^{n+1}_{i+1/2,j} - p^{n+1}_{i,j}}{\rho \Delta r_{i+1/2} S^{n+1}_{i+1/2,j}} \right), \tag{14} \]

where

\[ S^{n+1}_{i+1/2,j} = \gamma + \Delta k_{i+1/2} + \frac{(\Delta th_i v^{n+1}_{i+1/2,j} u^{n+1}_{i+1/2,j})^2}{\gamma + \Delta k_{i+1/2}}, \]
\[ M^{n+1}_{i+1/2,j} = \left( \frac{F^{n+1}_{i+1/2,j} + \Delta th_i v^{n+1}_{i+1/2,j} G^{n+1}_{i+1/2,j}}{\gamma + \Delta k_{i+1/2}} \right) / S^{n+1}_{i+1/2,j}. \]

The substitution of Eqs. (13) and (14) into (12) suggests the following linear system for the pressure updating:

\[
\begin{align*}
(a_{i+1/2,j} + a_{i-1/2,j} + a_{i,j+1/2} + a_{i,j-1/2}) p^{n+1}_{i,j} & = \Delta t \left( \frac{r_{i-1/2,j} \Delta z_j}{\rho} - \frac{\Delta t}{\gamma} \frac{\Delta z_j}{r_{i+1/2,j} \Delta z_i} \right) + c_{ij}, \\
a_{i+1/2,j} & = \Delta t \left( \frac{r_{i+1/2,j} \Delta z_j}{\rho} + \frac{\Delta t}{\gamma} \frac{\Delta z_j}{r_{i-1/2,j} \Delta z_i} \right), \tag{15} \\
\end{align*}
\]

Eq. (15) constitutes a symmetric and semi-positive-definite linear system. The only solution in the kernel of the matrix is the constant solution. This is expected since the pressure is determined up to a constant.

A preconditioned conjugate gradient method has been implemented to solve (15) [10, 16].
5. Solution algorithm

For any time step, the solution algorithm proceeds as follows:

- Set \( p^{n+1} = p^n \), \( u^{n+1} = u^n \), \( v^{n+1} = v^n \), \( w^{n+1} = w^n \).
- for \( \ell = 0, 1, \ldots, L - 1 \) the following four steps are performed:
  1. Evaluate \( p^{n+1(\ell+1)} \) by solving system (15).
  2. Evaluate \( u^{n+1(\ell+1)} \) by (14).
  3. Evaluate \( v^{n+1(\ell+1)} \) by (13).
  4. Evaluate \( w^{n+1(\ell+1)} \) by (11).

Also, it is to be noted that the solution procedure involves only repeated inversion of a symmetric and semi-positive-definite matrix, since the other terms are evaluated straightforwardly.

6. Results

The only useful measurements of radial and tangential component of velocities are those from Daggett and Keulegan [6] in steady flow conditions and axial-symmetry. The circulation was imposed at the border.

The experimental set-up consisted of a cylindrical reservoir (testing radius \( R = 0.45 \) m), having a circular hole in the bottom floor (radius \( R_0 = 0.05 \) m). The head of water over the bottom floor was \( H = 0.305 \) m, and the discharge \( Q = 0.019 \) m\(^3\)/s. The coefficient of discharge was very close to unity and the circulation was imposed at the border by means of orientable vanes (angle of deviation from radial direction: 45°). Therefore, the velocity vector, both at the border and throughout the depth, was uniformly set at an angle of 45° with respect to the radial direction. Measurements of radial and tangential components of velocity were performed only in steady-state condition by a miniature flow meter (diameter 1 cm).

Along the numerical border cells, the velocity vector, which is known from the experiments, is decomposed into its radial and tangential components, and the discharge is distributed uniformly over the numerical border cells. Since the experimental discharge coefficient were very close to unity, no \textit{vena contracta} section was considered in the outlet hole. The vertical velocity was assumed constant along any cell belonging to the bottom circular hole. The free surface evolution was not calculated in the numerical simulation and it was simply set equal to the stationary headwater, known from the experiments.

In Figs. 2 and 3, the comparison between numerical and experimental velocities is presented. A satisfactory agreement is achieved basically at any radial station, apart from the bottom boundary layer, where a small discrepancy is seen to occur. The flow field is everywhere accelerated towards the discharging hole, particularly along the bottom floor. The bottom boundary layer is definitely confined, which means that it neither grows nor does it undergo any separation, as it has also been demonstrated experimentally [20]. Thus the vorticity produced at the boundary layer remains confined there.

Being rather thin, the bottom boundary layer has little relevance on the whole flow field and, in fact, analytical solutions based on velocity potential theory (that is, nonviscous fluid), have been derived and successfully applied to Daggett and Keulegan’s data [5]. A detailed representation of the
boundary layer would imply a massive refinement of the mesh near the bottom; based on the above considerations, it was felt that such a mesh refinement would introduce a great deal of complication that is, as a matter of fact, unnecessary. The numerical node nearest to the bottom was placed at 0.004 m from the floor, so that the simulation of the boundary layer was precluded.

6.1. The radial jet

A sort of jet is clearly discernible in the lowermost part of the radial velocity profile, as it was already described in the introductory remarks. This radial jet, swiftly flows underneath the relatively stagnant superficial region, which is mostly involved in the rotational motion.
The radial jet bears a strong resemblance to the Ekman layer that, however, occurs if all the experimental apparatus is forced to rotate (see, for instance, [15]). The Ekman layer was recognized as controlling the external vortex flow. Also, vortex circulation was found to be determined both by the flowrate and by the Ekman layer depth [15].

Referring to the free surface vortex, Levi [11] was the first researcher to discover the jet, as it was termed by the author, i.e., the fast flowing fluid along the bottom layer. This jet has much relevance in explaining the mechanism of vortex generation, that is based on momentum transfer between the jet and the overlying region of still fluid.

Since vortex generation in a liquid is usually associated with the formation of a gradually deepening dimple that undergoes a progressive stretching, it is considered obvious that the vortex begins aloft and subsequently stretches downwards. Levi refutted this common belief and argued instead that the vortex should grow from below since the generating momentum comes from the underlying jet. Actually, the very first scientist who ventured in presenting this brilliant conjecture was Venturi (1797), cited by Levi [12], who stated: ...the rotation always begins and appears more rapid in those parts of the fluid closer to the bottom .......... the funnel begins to form below .......

Levi's hypothesis was confirmed by a simple experiment [20] consisting of injecting dye simultaneously on the free surface and on the vertical axis of symmetry at a depth \( z \approx 0.6 \). The swirling motion was found to develop very quickly in the lower part, while at the same time the dye on the free surface remained perfectly still. Therefore, the vortex motion starts from below and cannot be seen from outside until the stretching of vortex line manages to reach the free surface, where the dimple is eventually generated.

The phenomenon described above is well simulated by the numerical model: in fact, after having attained the radial and vertical flow field, the tangential velocity was imposed at the border. This tangential velocity propagates inside the domain and concentrates over the bottom hole, where it undergoes a huge amplification. This early vortex, as it was speculated by Venturi (1797), Levi [11], and as seen experimentally by Trivellato [20], is stretched by the vertical component of velocity and propagates towards the free surface.

7. Conclusions

While the numerical model needs to be corroborated by more experimental data, according to the presented results (to be considered as yet as preliminary), it seems capable of reproducing the two most distinctive features of vortex motion, namely the azimuthal motion and the radial jet swiftly flowing along the reservoir bottom.

Not only are Eqs. (1)–(4) valid for a laminar vortex, but they can also be accurate for turbulent vortices as well, since they are sufficient to replace the kinematic viscosity \( \mu/\rho \) by the eddy viscosity \( \varepsilon \). However, the value of \( \varepsilon \) is still a matter of speculation and no general consensus has been achieved so far.

A linear relationship, \( \varepsilon = k_1 \Gamma_\infty \), was first postulated by Squire [19]. Referring to the decay of trailing vortices behind an aircraft, Scorer [18] proposed \( k_1 \approx 4 \cdot 10^{-4} \). Anwar [2] observed that the eddy viscosity for turbulent vortex flow should not be constant. Odgaard [17] estimated \( k_1 \approx 6 \cdot 10^{-5} \), while Hite [19] deduced \( k_1 = 0.041 \).
As the last observation, the knowledge of the vortex far-field circulation for a free surface vortex in an infinite domain is only known from experiments; in fact, an analytical method for the a priori prediction of the circulation is currently not available.

Acknowledgements

This research received financial support by the Ministry of University and of Scientific and Technologic Research (funds M.U.R.S.T. 40%).

References