Simulation of natural convection in a reservoir

P. Jelínek\textsuperscript{a}, V. Havlík\textsuperscript{b}, R. Černý\textsuperscript{a}, P. Přikryl\textsuperscript{c}

\textsuperscript{a}Czech Technical University, Faculty of Civil Engineering, Department of Physics, Thákurova 7, 166 29 Prague 6, Czech Republic
EMail: jelinek@fsv.cvut.cz
EMail: cernyr@fsv.cvut.cz

\textsuperscript{b}Czech Technical University, Faculty of Civil Engineering, Department of Hydraulics and Hydrology, Thákurova 7, 166 29 Prague 6, Czech Republic
EMail: havel@hpux.fsv.cvut.cz

\textsuperscript{c}Mathematical Institute of the Academy of Sciences of the Czech Republic, Žitná 25, 115 67 Prague 1, Czech Republic
EMail: prikryl@math.cas.cz

Abstract

The governing equations for modelling transport processes in reservoirs caused by the influence of temperature gradient are presented. Numerical solution based on the finite element method illustrates the possibility of solving transport processes in simple and more complex domains. The influences of shape and size of the computational domain, various boundary conditions and the Rayleigh number will be analysed. Computational results for the natural convection in the Pastviny reservoir (Czech Republic) are shown. The input data consists of the reservoir dimensions, measured material parameters, and temperature profiles. The computed results are presented using streamline contours and velocity vector plots, and temperature contours as well.
1 Introduction

Improvement of the water quality in reservoirs requires a decrease in the input of pollutants from the catchment and the implementation of a reservoir management plan in order to meet water quantity and quality requirements. Hydrodynamics of reservoirs with thermal stratification is a key issue which has a direct impact on transport processes and eutrophisation.

One-dimensional models for the simulation of lake and reservoir hydrodynamics and water quality have been known since 1970 (see Chen and Orlob [1], Imberger et al. [2], for instance). While one-dimensional models can simulate only the vertical or horizontal behaviour of water bodies, two-dimensional and three-dimensional flow models are required to simulate the important mechanisms. A great effort is being paid to the use of Computational Fluid Dynamics (CFD), Ta and Brignal [3], Cox et al. [4].

Natural convection generated in reservoirs by the temperature gradient must take the buoyancy effects into account correctly. Near the heating or cooling walls, the flow can be strongly affected by buoyancy as well as by the viscous forces. Since the flow is turbulent, the classical $k - \varepsilon$ models must be modified, see e.g. Rodi [5] and Patel et al. [6].

2 Governing equations

The governing equations for the turbulent flow which is strongly affected by buoyancy effects consist of the equations of continuity, momentum and energy. The fluid is supposed to be incompressible and the Boussinesq approximation is used for the gravity forces. The equations, in Cartesian co-ordinates, can be written as follows:

Continuity:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0. \quad (1)$$

Momentum:

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = - \frac{1}{\rho_{ref}} \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \nu \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - u'_i u'_j \right]$$

$$+ f_i \frac{\rho - \rho_{ref}}{\rho_{ref}}. \quad (2)$$
Thermal energy:

\[
\frac{\partial T}{\partial t} + \bar{u}_i \frac{\partial T}{\partial x_i} = \frac{\partial}{\partial x_i} \left( D_T \frac{\partial T}{\partial x_i} - \bar{u}_i \bar{T}' \right) + S_T. \tag{3}
\]

Here, \( \bar{u}_i \) are the mean velocity components, \( \bar{p} \) is the mean pressure, \( f_i \) is the density of volume forces, \( \rho \) denotes the density, \( \rho_{\text{ref}} \) is a reference density, \( \nu \) is the kinematic viscosity, \( T \) is the mean temperature, \( D_T \) is the thermal diffusivity, and \( S_T \) is a source term of thermal energy. A set of five equations for the turbulent flow of incompressible fluid (1)–(3) has 14 unknowns altogether (\( \bar{u}_i' \bar{u}_j', \bar{u}_i, \bar{p}, \bar{T} \) and \( \bar{u}_i \bar{T}' \)). Hence, this set of differential equations can be closed with the aid of new completing relations only to determine the Reynolds stress components.

One possibility of solving this set of equations consists in using two-equation models. Among a variety of two-equation models, a promising one is the \( k - \omega \) model defined by Patel et al. [6], where \( \omega \) is defined by

\[
\omega = \frac{1}{c_\mu} \frac{\varepsilon}{k}. \tag{4}
\]

In Eqn. (4), \( c_\mu \) is a model constant (\( \approx 0.09 \)) (see [6] for details), \( \varepsilon \) is the dissipation, and \( k \) is the turbulent kinetic energy.

For the turbulent flows which are affected by viscous forces, a modification of these standard \( k - \omega \) models for low Reynolds number flows has to be considered. The model can be described as follows (Wilcox and Rubesin [7]):

\[
\frac{\partial k}{\partial t} + \bar{u}_i \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ (\nu + \frac{\nu_t}{\sigma_k}) \frac{\partial k}{\partial x_i} \right] + G - \omega k,
\]

\[
\frac{\partial \omega}{\partial t} + \bar{u}_i \frac{\partial \omega}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ (\nu + \frac{\nu_t}{\sigma_\omega}) \frac{\partial \omega}{\partial x_i} \right] + c_1 \frac{\omega}{k} G - c_2 \omega^2,
\]

where

\[
\nu_t = c_\mu \frac{k}{\omega}, \quad c_1 = 0.555, \quad c_2 = 0.8333, \quad c_\mu = 0.09,
\]

\[
\sigma_k = \sigma_\omega = 2,
\]

and the other parameters are the same as with the classical \( k - \epsilon \) models (see [1] for details).
3 Simulation results and discussion

The storage reservoir Pastviny is used for producing peak electricity, for recreation, flood protection, and for conserving minimum and/or maximum discharges downstream. There is only one main inflowing stream. While minimum discharges are within a range of 0.55 m$^3$.s$^{-1}$, maximum permissible discharge in the river downstream is 30 m$^3$.s$^{-1}$. The daily variation of water level in the reservoir is 0.1 m but during the peak hours it can increase up to 0.45 m. We try to simulate the turbulent flows in the reservoir which are driven by buoyancy effects due to the temperature gradient in vertical direction and by the bottom slope. For the numerical simulation of the steady state natural convection, the CFD code FIDAP [8], [9], which contains the model of Wilcox, was used.

The length of the main reservoir is 5780 m and near the dam the maximum depth is 33.4 m and width 165 m. The bottom slope is approximately 0.0057, the aspect ratio $a = 5780/33.4 = 173$. In order to perform the simulation of natural convection the whole domain has been approximated by a trapezoidal longitudinal cross-section. In numerical experiments, we considered the aspect ratios 50, 100 and 200. Figure 1 shows the schematic of the computational domain.

From a set of measured average daily temperature data, a typical linear variation of the temperature with depth from October 1996 has been used for simulations. The temperature of the surface layer was 14.8°C and that of the bottom layer 8.0°C. The physical parameters of water were as follows:

- kinematic viscosity $\mu = 8.55 \cdot 10^{-4}$ [kg.m$^{-1}$.s$^{-1}$];
- specific heat $c_p = 4179.0$ [J.kg$^{-1}$.K$^{-1}$];
- thermal conductivity $\lambda = 0.613$ [W.m$^{-1}$.K$^{-1}$];
- volume expansion coefficient $\beta = 2.761 \cdot 10^{-4}$ [K$^{-1}$];
- density $\rho = 997.0$ [kg.m$^{-3}$];
- thermal diffusivity $D_T = 0.15$ [m$^2$.s$^{-1}$].

The following boundary conditions (see Fig. 1) have been considered for the numerical simulation:

1. WALL, INLET, and BOTTOM

    (a) the velocity components $u$, $v$ (direction $x$, $y$) are equal to zero
(b) the temperature is defined as \( T = T_B + \left( \frac{T_E - T_B}{H} \right) y \), where
\[ T_E = 14.0^\circ C, \quad T_B = 8.0^\circ C \]

2. ELEVATION

(a) the velocity components \( u, v \) (direction \( x, y \)) are equal to zero

(b) the temperature is defined as \( T = T_E = 14.0^\circ C \)

A set of numerical calculations was performed in order to optimize the numerical parameters of the model, namely the type of the iteration procedure, the magnitude of the pressure penalty \( \epsilon_p \), the initial values of \( k \) and \( \epsilon \), and the optimal value of the coefficient \( \alpha_c \), which affects the rate of convergence. The following optimum parameters were obtained: the successive approximation method of solution, \( \epsilon_p = 10^{-10} \), the initial values \( k = 3 \times 10^{-5} \) and \( \epsilon = 10^{-5} \), and \( \alpha_c = 0.5 \). Particularly the pressure penalty had a remarkable influence on the accuracy of results because no reasonable convergence was achieved for any penalties outside the interval \([10^{-11}, 10^{-9}]\). Also
the initial values of $k$ and $\varepsilon$ had a significant influence on the stability of solution. A typical number of iteration steps for achieving the convergence of all variables was of the order $10^5$.

Figure 2: Velocity vector field in the region close to the dam for $a = 50$.

Figure 3: Temperature contours in the region close to the dam for $a = 50$.

The flow pattern was very similar for all aspect ratios in our computations. Several isolated recirculating flow regions arose in the reservoir in the vicinity of the dam, while on the opposite side practically no flow was observed. As an example of numerical calculations, Figures 2, 3, and 4 illustrate the velocity vector field, streamlines and temperature in the vicinity of the left boundary (dam). The veloci-
ties were typically $10^{-4} \text{ m.s}^{-1}$, close to the dam they increased up to $10^{-2} \text{ m.s}^{-1}$. The Rayleigh number was $5.2 \times 10^{15}$.

Figure 4: Streamline contours in the region close to the dam for $a = 50$.

4 Conclusion

An attempt to simulate numerically the natural convection in the Pastviny reservoir using the CFD code FIDAP showed that the solution strongly depends on the reasonable selection of the computational grid and on a proper choice of the numerical parameters. The realistic steady state flow field has been achieved for the aspect ratio $a = 50$, while for $a = 100$ and 200 no stable converged solution was obtained.

However, the use of CFD provides a tool, which can have a direct application in the formulation of management strategies to protect water body environments in the modelled reservoir.

Acknowledgement

This paper is based upon work supported by the Grant Agency of the Czech Republic, under grant # 103/96/1710.
References


