J. Hydrol. Hydromech., 63, 2015, 4, 334–341
DOI: 10.1515/johh-2015-0033

3-D CFD simulations of hydrodynamics in the Sulejow dam reservoir

Aleksandra Ziemińska-Stolarska*, Andrzej Polańczyk, Ireneusz Zbiciński

Technical University of Lodz, Faculty of Process and Environmental Engineering, Wólczańska 213 Str., 90-924 Łódź, Poland.
* Corresponding author. E-mails: aleksandra.zieminska-stolarska@p.lodz.pl, andrzej.polanczyk@gmail.com, ireneusz.zbicinski@p.lodz.pl

Abstract: This paper reports the processes by which a single-phase 3-D CFD model of hydrodynamics in a 17-km-long dam reservoir was developed, verified and tested. A simplified VOF model of flow was elaborated to determine the effect of wind on hydrodynamics in the lake. A hexahedral mesh with over 17 million elements and a k-ω SST turbulence model were defined for single-phase simulations in steady-state conditions. The model was verified on the basis of the extensive flow measurements (StreamPro ADCP, USA). Excellent agreement (average error of less than 10%) between computed and measured velocity profiles was found. The simulation results proved a strong effect of wind on hydrodynamics in the lake, especially on the development of the water circulation pattern in the lacustrine zone.

Keywords: CFD; 3-D hydrodynamic model; Effect of wind; Dam reservoir.

INTRODUCTION

In recent years, numerical models of flow in water bodies have been widely used; this is because the complexity of hydrodynamics requires an adaptation of numerical modelling to provide an accurate description of flow velocity distribution. Understanding of flow through open channels, whether in natural water bodies such as rivers, lakes and estuaries, or man-made structures including dam reservoirs, spillways and weirs, is of crucial importance for addressing numerous hydraulic engineering problems related for example to hydrodynamics, water retention time, and nutrient and sediment transport.

Although all the hydrodynamic processes going on in an aquatic system are difficult to describe, some authors have analysed mechanisms of flow in large-scale (Dubnyak and Timchenko, 2000; Zinke et al., 2010) and small-scale mixing processes (Andersson et al., 2013; Kriš and Hadi, 2010; Wüest and Lorke, 2003), or both (Fischer-Antze, 2005). 3-D numerical models of flow calculations in natural water bodies have been successfully applied for rivers with simple geometries (Dargahi, 2004; Sinha et al., 1998). In a study conducted by Sinha and colleagues (Sinha et al., 1998) a 3-D model of flow through a 4-km stretch of the Columbia River, downstream of the Wanapum Dam, was developed and validated. The authors succeeded in modelling the flow with rapidly varying bed topography and the presence of islands. Work on flow through curved or straight open channels of simple cross-sectional shape (rectangular or trapezoidal) has been reported by Demuren (Demuren, 1993) and Meselhe and colleagues (Meselhe et al., 1995). Both these studies presented a 3-D numerical model for the calculation of turbulent flow in meandering channels and provided a hydrodynamic basis for the study of the mechanisms which form river meanders. The case studies exhibited some of the features encountered in real rivers, including longitudinal curvature and varying bed topography. Meselhe and colleagues proposed a simplified approach for calculating the water surface elevation as a part of the overall solution procedure.

Due to limited computational resources for accurate simulations of large reservoirs, 3-D CFD modelling has only recently come to be used in hydrodynamics simulations in dam reservoirs (Kennedy et al., 2006) including analysis of density of flow (Ünes and Varcin, 2015).

Early studies in numerical modelling of lakes were carried out by Simons (Simons, 1974), who applied a 3-D hydrodynamic model to Lake Ontario. Simons' model took most physical processes into consideration, including water levels, currents, temperature, and the transport of dissolved or suspended materials in large water bodies. The model has been verified extensively with current and water-level data, mostly in episodic responses (tropical storm). The constant eddy-viscosity model was applied to compute turbulence. Falconer (Falconer et al., 1999 used a similar approach to calculate wind-induced currents in Esthwaite Water, UK. Good agreement was found with field measurements of water velocities after calibration of a constant eddy-viscosity model. Field measurements of velocity and turbulent quantities have been made much more accessible with the development of acoustic measurement methods. Nowadays ADCP instruments are often used to validate CFD models (Dargahi, 2004; Sándor, 2009; Zinke et al., 2010).

The aim of this work was to develop a CFD model of hydrodynamics in large water bodies, taking into account the effect of wind on the water flow pattern.

CASE STUDY

In contrast to flowing waters, dam reservoirs were not emphasized in the early years of water-quality modelling, as this issue had never been a major focus of urban development. Starting in the 1970s, however, it was recognized that natural and man-made lakes are equally important as, if not more important than estuaries and rivers from a recreational standpoint. The Sulejow reservoir was selected as the study area as representative of many lowland dam reservoirs, and also because of the availability of a large data base on the hydrological and morphological parameters of the lake. Sulejow reservoir is situated in the Voivodeship of Lodz, Poland (51^{o}22` – 51$^o$28`N, 19^{o}51 – 20^{o}01`S). The basic parameters of the lake are described in Table 1 (Ziemińska-Stolarska et al., 2013).

The mathematical model

In order to determine all aspects of the hydrodynamics in the dam reservoir, a two-phase 3-D model needed to be developed. Owing to the large size and complexity of the computational grid, as well as still limited computer capacity, such an approach would be difficult to apply for a large water body. In the literature, 2-phase models of reservoir were developed only for simple, down-scale model (1:50) (Andersson et al., 2013).

Table 1. Parameters of the Sulejow reservoir.

Name	Value
Length	17.1 km
Maximum width	2.1 km
Average width	1.5 km
Average depth	3.3 m
Maximum depth	15 m
Shoreline length	58 km
Surface area	22 km^2
Usable capacity	61 x 10^6 m^3
Maximum capacity	75 x 10^6 km^2
Drainage basin area	4900 km^2

The aim of the present work is to build a single-phase 3-D model, taking into account the effect of wind on the flow hydrodynamics. Such an approach excludes only modelling of the dynamics of waves, which in the case of lowland, dam reservoirs does not affect hydrodynamics in the lake.

The implementation of the 3-D CFD model for the Sulejow reservoir required significant effort in terms of the determination of basin geometry, definition of the physical domain, generation of the computational mesh, imposition of boundary and initial conditions, and estimation of the solution convergence.

Under most natural and practical open-channel flow conditions, the flow is turbulent. Even at the low velocities occurring in water reservoirs, laminar flow does not appear due to differences in depth and associated flow instabilities. In practice, for water flow in large reservoirs, the Reynolds number assumes high values because of the low viscosity and large transverse dimensions of the channel.

There is no clearly delineated guidance in the literature as to which model of turbulence should be used in CFD calculations in large water bodies. The different characteristics of each modelled object (differences in width and depth, water velocity, bottom roughness, the presence of swirls, etc.) render difficult the selection of the most suitable turbulence model for an individual case. As a result of the substantially lower computational effort required, the k-ε model is one of the most commonly used turbulence models for the solution of practical engineering problems. There is however a large amount of evidence that, although the k-ε model reproduces qualitatively many of the important flow parameters, it is not entirely satisfactory in some complex flow situations, particularly those involving swirling and rotating flows in curved geometries (Wilcox, 1998).

In this work, following suggestions of (Nichols, 2008), a k-ω shear-stress transport (SST) model was used to simulate the turbulence character of the flow, which usually gives better results than the other turbulence models for this type of the objects.

2-D, two-phase model under wind conditions

In the single-phase CFD model, the phases (water-air) are separated by a plate which allows calculations of the flow of one phase (water) only. As the wind affects the velocity of flow, in order to take into account the effect of wind on hydrodynamics in a single-phase CFD model the plate must move with a speed and direction equal to the speed and direction of the upper water layer.

To determine the velocity at which the plate will move on the water surface and interact with the underlying layers of water, and to apply this value in a single-phase CFD model, a two-phase, 2D model of a straight channel with a length of 80 m, a 3-m layer of water and a 3-m layer of air was elaborated. The hexahedral, structured mesh was generated for the channel geometry, which contained 159,242 elements (159,116 hexahedra, 126 wedges) and 321,872 nodes, respectively.

Wind accelerates near-surface fluid particles by imparting momentum to the fluid through the surface stresses. The assumption was that in the computational domain, the water phase consists of two outlets allowing for fluid reversing. In the conducted analysis, movement of the water was induced only by the wind action. Analysis of flow in the 2-D model was intended to determine which boundary conditions best describe the prevailing situation on the water surface under the effect of wind.

In all the calculations, ANSYS FLUENT 14.0 software was applied.

Boundary conditions

For the 2-D, two-phase CFD model of hydrodynamics in the channel the following boundary conditions were imposed.

Inlet

The definition of the inlet requires the values of the velocity vectors and turbulence properties. For the air inlet, the simulations were first conducted at a speed of 3 m/s, which is equal to the average wind velocity in the area of the reservoir. The velocity profile obtained at the outlet of the computational domain was loaded as an input file to receive the velocity profile at the inlet. This approach enabled a fully developed velocity profile to be obtained for a small domain.

Implementation of the volume-of-fluid approach requires additional boundary conditions to be specified; the turbulence intensity at the inlet and the turbulence viscosity ratio.

Kennedy and colleagues (Kennedy et al., 2006) carried out a sensitivity analysis on the turbulence intensity in the simple channel geometry and found that a value of 4% was able to match the conditions well. The turbulence viscosity ratio (the ratio of turbulent to laminar viscosity) was set to 10%.

Outlet

Pressure outlet boundary conditions require specification of a static pressure at the outflow. Convergence difficulties were minimized by implementing the specified values for the backflow quantities (backflow turbulence intensity and viscosity ratio).

Wall region

A no-slip boundary condition was applied on the wall, i.e., at the bottom and top of the meshed model.

Results for the two-phase, 2-D flow CFD model

The two-phase 2-D CFD model enabled a velocity profile to be determined in the channel under the same input data and average wind conditions as in the analysed dam reservoir. As the result, the velocity of the plate (equal to the velocity of the upper water layer), which corresponds to the real, average velocity in the area of the Sulejow reservoir, was obtained. Figure 1 depicts the velocity profile of water at the outlet of the computational domain. We found that the average value of the upper water layer's velocity was equal to 0.147 m/s (Figure 1). This value was assumed as the velocity of the plate separating the phases in the simulations of flow hydrodynamics in the single-phase 3-D model.

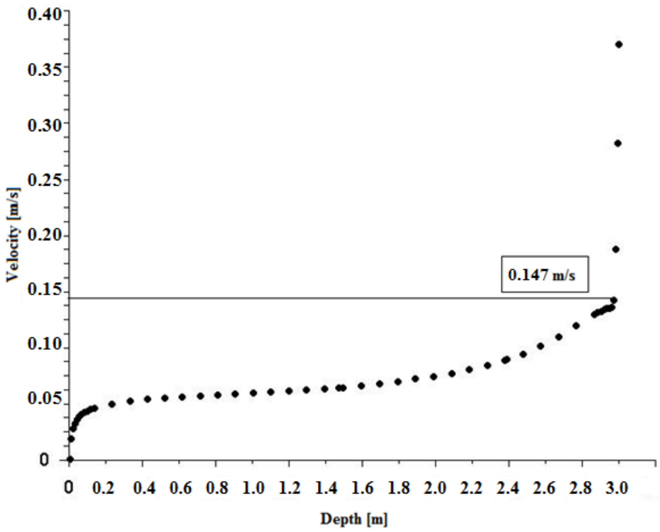

Fig. 1. The velocity profile of water at the outlet of the computational domain.

Single-phase 3-D CDF model of flow in dam reservoir
Development of 3-D geometry of the lake

The first step in the numerical procedure to elaborate the single-phase 3-D CFD model of flow in dam reservoir was to develop the 3-D geometry of the 17-km-long Sulejow reservoir (Gambit 2.2.30 (ANSYS, USA, 2004)). In this study, the "bottom-up" approach was implemented in order to obtain a geometry from the set of separated surfaces. Each measured cross-section was defined by the data on the distance between two banks (connection length between probing points in the section was 5 m) and elevation (normal impoundment level in the Sulejow reservoir is 166.6 m a.s.l.).

Final verification of the accuracy of the reservoir shape was carried out on the basis of satellite photographs (Source: geoportal.gov.pl), which confirmed an excellent match of the approximated geometry (Figure 2).

Generated in this way, 3-D geometry of the artificial lake was meshed to perform the numerical calculations.

Computational mesh

To apply a finite-volume method in the CFD model of the reservoir, the three-dimensional space must be subdivided into a large number of cells. However, the high ratio of the breadth to the depth dimensions and the irregular shape of the reservoir make this process difficult. Appropriate mesh resolution is linked to the hydrodynamic conditions, flow features and discretization schemes.

In finite-volume methods for numerical simulations, the hexahedral mesh ("hex mesh") is preferred over the tetrahedral mesh ("tet" mesh), owing to the reduced error and smaller number of elements (Shepherd and Johnson, 2008). Typically, tet mesh is preferred for filling irregular spaces, since existing algorithms can semi-automatically subdivide most of the spaces (Kennedy et al., 2006). Structured meshes are better suited to shallow reservoirs, while unstructured meshes match better to deeper reservoirs (or those with smaller aspect ratio). To fill the shallow space representing the reservoir area with minimally skewed, hexahedral cells, the typical cell dimension must be small compared to the depth of the water. The meshing process thus becomes computationally expensive due to the requirement of a large number of elements. The domain surface was discretized using structural (hexahedral) mesh, with 16,787,820 active cells and 17,717,364 nodes, respectively. The use of a coarser grid ($<10^5$ and 10^6 elements) caused rapid solution divergence. However, increasing the mesh number to $18x10^6$ had no further influence on the results.

Figures 3 a) and b) show the hexahedral, structured mesh generated for the Sulejow reservoir geometry. Exclusion from the model of some near-shore regions of the basin, with the shallow depth (< 0.3 m) and inconsiderable slope was necessary to avoid generation of cells with highly acute angles.

The quality of the numerical grid was determined by the shape and size of the computing field, the total number of elements used in the generated numerical grid, and the position of the first node relative to the plane of the wall. To assess the quality of the numerical grid elements, three parameters were used (Gambit User's Guide 2.2.30, 2004):

- y^+: a non-dimensional variable based on the distance from the wall through the boundary layer to the first node away from the wall;
- skewness: determining the quality of the individual grid elements;
- aspect ratio: defining the degree of deformation of the mesh elements.

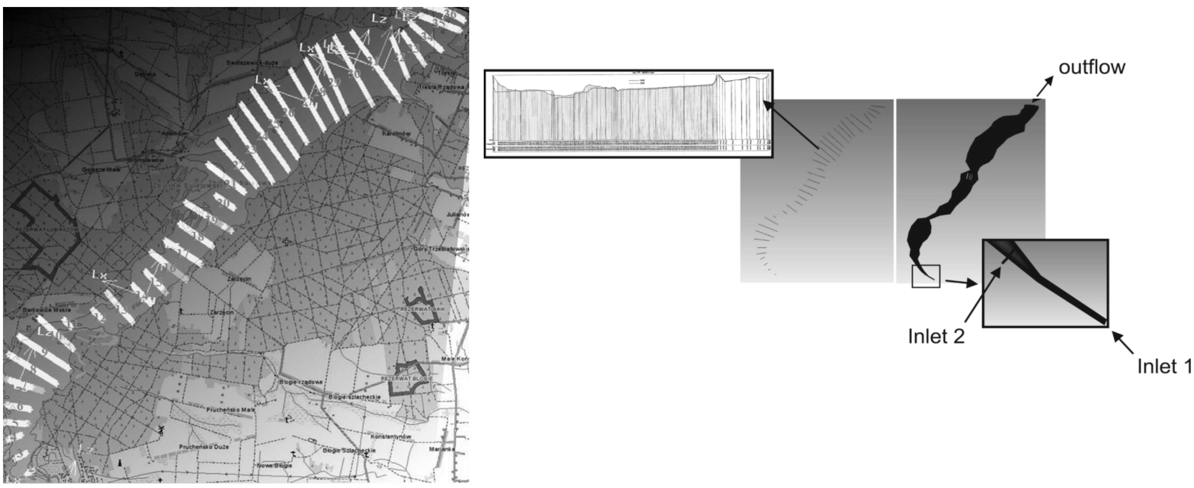

Fig. 2. Sulejow reservoir geometry, 36 cross-sections marked.

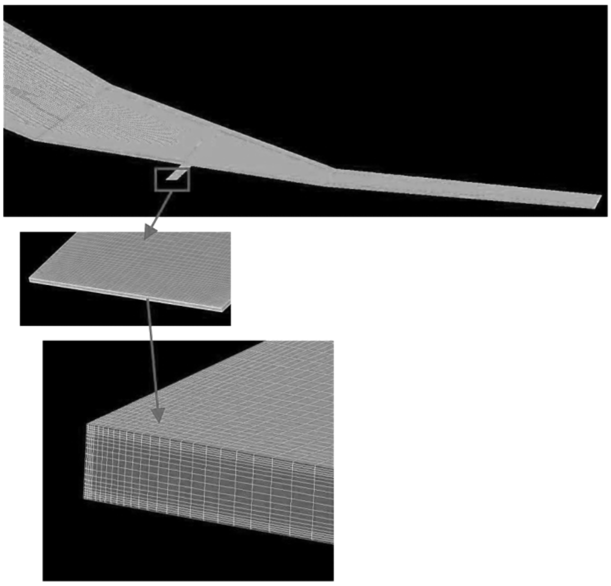

Fig. 3a. Fragment of the structural mesh with the boundary layer.

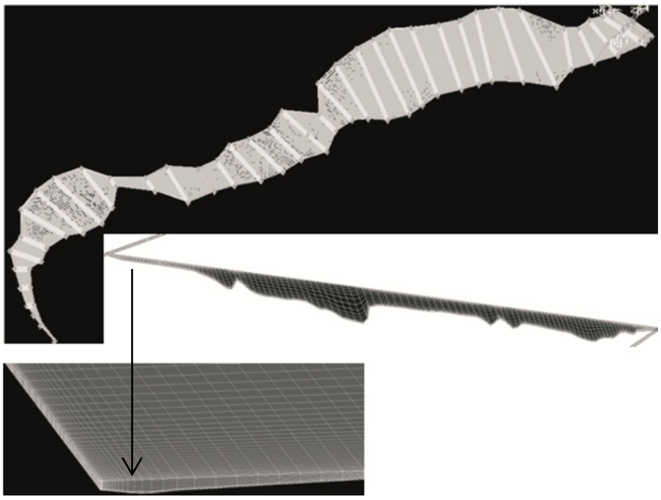

Fig. 3b. Example of the computational domain with the structural mesh for the whole domain and one cross-section; under magnification, hexahedral elements are included in the boundary layer.

The application of the SST turbulence model requires a fine mesh near the wall, with a value of y+ of less than 2 nodes inside the boundary layer. By specifying the height of the first layer (0.01 m) and the growth factor (0.1), following the exponential law of increasing the height, the layers were automatically sized by the Gambit program (Gambit User`s Guide 2.2.30, 2004).

Boundary conditions

Prior to running the CFD calculations, the input file, which consisted of the information on the properties of the inlets and outlet, and the simulation settings needed to be specified.

Indispensable elements of the reservoir geometry include the Pilica and Luciaza Rivers, as well as dam-outflow.

As inlet boundary conditions in the analysed domain, inflows from the two tributaries were specified. Simulated inflow boundaries were defined with mass-flow discharges, normal to the boundary. The values of the rivers' discharge in selected months are presented in Table 3.

Table 2. Values of y $^+$, "skewness" and "aspect ratio" parameters for the analysed numerical grid.

Parameter	Values for numerical grids in present work	The limit values in the numerical simulations
y^+	~1.8	≤ 2
Skewness	~0.9	0 ÷ 1
Aspect ratio	~1.7	≥ 1

Table 3. The selected values of flow rate in the Pilica and Luciaza Rivers in 2007.

Month	Pilica River Mean flow discharge [m^3/s]	Luciaza River Mean flow discharge [m^3/s]
March	22.33	4.92
July	16.45	1.91
October	16.37	1.69
December	25.65	2.55

The basin outlet was specified as an outflow boundary. Due to the fact that the study was focused on flow behaviour within the basin, where conditions are steady, the flow rate at the outflow boundary was set as equal to the sum of the inflow rates.

On the border of fluid and solid, wall conditions were applied. At the walls, the no-slip conditions were assumed, implying that at a solid boundary the fluid would have zero velocity relative to the boundary.

The bottom and sides were assigned k_s = 0.02 m sand-grain roughness height (for smooth beds, k_s = 0), with a roughness constant of 0.5 (Wu et al., 2000). No-slip boundary conditions were applied between the moving plate and water surface.

Solution method

Solving the Navier-Stokes equations on a highly resolved mesh is a significant computational challenge requiring a range of decisions on solution technique and numerical control parameters. The numerical solution to the momentum equations of fluid for numerical prediction of flow was obtained using CFD software. To calculate the derivatives of the flow variables, a second-order upwind/central difference scheme was applied (Patankar, 1980; Versteeg and Malalasekera, 1995).

The choice was motivated by evidence reported in the literature showing the greater accuracy of the scheme in comparison with central differencing or first-order upwind schemes (Dargahi, 2004). A conventional procedure to evaluate solution convergence uses residuals. Convergence is reached when the normalized changes in variables between successive iterations are equal to or less than a certain limit. In this work, a maximum residual level of 10^{-4} was set to maintain the accuracy of the solution at a level sufficient for most engineering applications. To obtain a converged solution, 30 hours of computations and approximately 1400 interactions were necessary.

For pressure velocity coupling, following suggestions made by Versteed and Malalasekera (Versteed and Malalasekera, 1995), who carried out a comprehensive review of the solution procedure, the SIMPLEC (SIMPLE-Consistent) algorithm was used (see ANSYS documentation).

Flow-field measurements

The aim of the flow velocity measurements was to verify the developed CFD model of hydrodynamics in the dam reservoir. A Teledyne RD Instruments' StreamPro ADCP (Acoustic Doppler Current Profiler) was used. The places selected for the verification of the hydrodynamic model were arranged along the longitudinal axis of the Sulejow reservoir. The selection of the areas was dictated by the different expected characters of the flow at the indicated locations (Figure 4):

1) The location closest to the backwaters of the reservoir, located in the riverine zone, characterized by small depths (< 3 m), and the highest flow rates.
2) Located in the upper, narrow part of the reservoir, with higher flow velocities, resulting in a half-river character. The depth at this point was about 4 m.
3) Situated in the central part of the reservoir, characterized by a low flow velocity, due to greater width (1500 m) and basin depth of approx. 6 m.
4) Located in the lower part of the artificial lake, closest to the dam of the reservoir, with depth of about 7-8 m. The location is characterized, in addition to greater depths, by the largest cross-sectional width, approximately 2000 m.

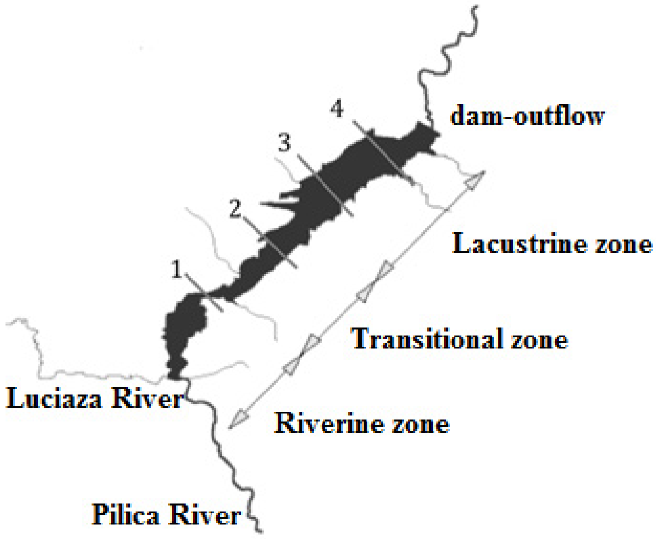

Fig. 4. Division of the Sulejow reservoir into riverine, transitional and lacustrine zones, with the locations used for model verification marked.

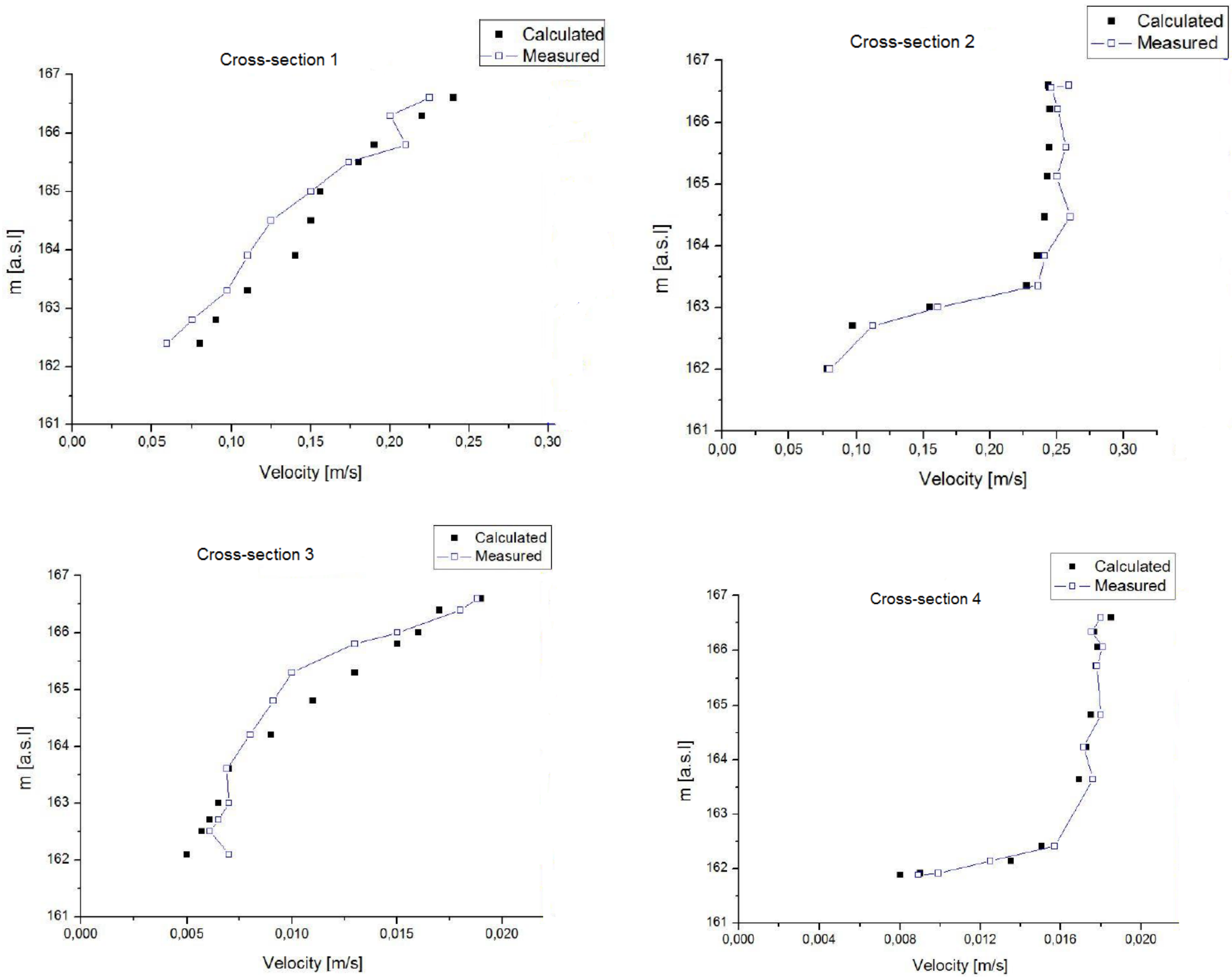

Fig. 5. Velocity profiles from time-averaged fixed boat ADCP measurement and CFD calculations.

The velocity profiles from the CFD results were compared with time-averaged ADCP profiles. Figure 5 shows the examples of the velocity profiles from the horizontal locations of four cross-sections. Comparison of measured and calculated velocity profiles showed good agreement between the flow pattern predicted by the CFD model and that determined experimentally. Analysis of the velocity profiles in different cross-sections shows differences in shape and velocity magnitude which results from bathymetric and hydrographic conditions in the lake. Maximum speed of water (0.25 m/s) was found in the riverine zone because of the lake contraction. The lowest water velocity (around 0.019 m/s) was detected in the lacustrine zone, due to the geometry of this part of the lake (the widest part of the reservoir) and backflow of water in recirculating zones. Average error between the computed and measured velocity for each profile was not higher than 10%, which proves the accuracy of the CFD model of hydrodynamics in the Sulejow reservoir.

Single-phase 3-D CFD model of hydrodynamics in the Sulejow reservoir

Extensive single-phase CFD calculations of hydrodynamics in the Sulejow reservoir were carried out in steady-state, isothermal conditions for the year 2007, for which the largest database of water parameters was available.

Examples of the simulations for selected months are shown in Figs. 6 a, b and c. The Figure presents the surface streamlines distribution for the 17-km-long artificial lake in March, July and December 2007, respectively (Table 3). The distinctions in the courses of the pathlines, result from the different hydrodynamic conditions in summer and winter of the reference year (see Table 2). Average velocity in the reservoir was 0.2 m/s. Higher velocities observed at the Pilica inlet (up to 1 m/s) were caused by flow curvature and narrower cross-section (30 m). Rapid decrease in speed in the backwater, accelerating only in the vicinity of the constrictions near the islands, is visible. In the lacustrine part of the reservoir, flow velocity was slow (< 0.1 m/s) and the pathlines formed a closed circulation. Such circulating zones could be as large as 0.5 km across, with increased water retention time in these regions.

Figure 7 presents the comparison of velocities (m/s) in the Sulejow reservoir in October 2007 (Table 3) under conditions of wind and no wind. Analysis of the figure shows significant differences in the distribution of flow pattern and velocity fields when the wind is taken into consideration.

The results show that when steady flow pattern developed in the basin, large regions of recirculation form below the outlet of the reservoir. At this point, significant slowdown in the flow occurred (< 0.01 m/s), developing, e.g., favourable conditions for the growth of algae species. Analysis of Figure 7 proves that the wind drift affects the hydrodynamics of the water body, developing large recirculation zones.

Figure 8 presents the thalweg arising from the connections of the deepest points in the cross-sections, characterized simultaneously by the highest flow rates. The calculations reveal that the maximum velocity occurs in the centre of the section, while the minimum velocity occurs in the near-wall region, reflecting the real nature of flow in the channel. A series of simulations indicate that the main path of flow approximately follows the old channel of the Pilica River.

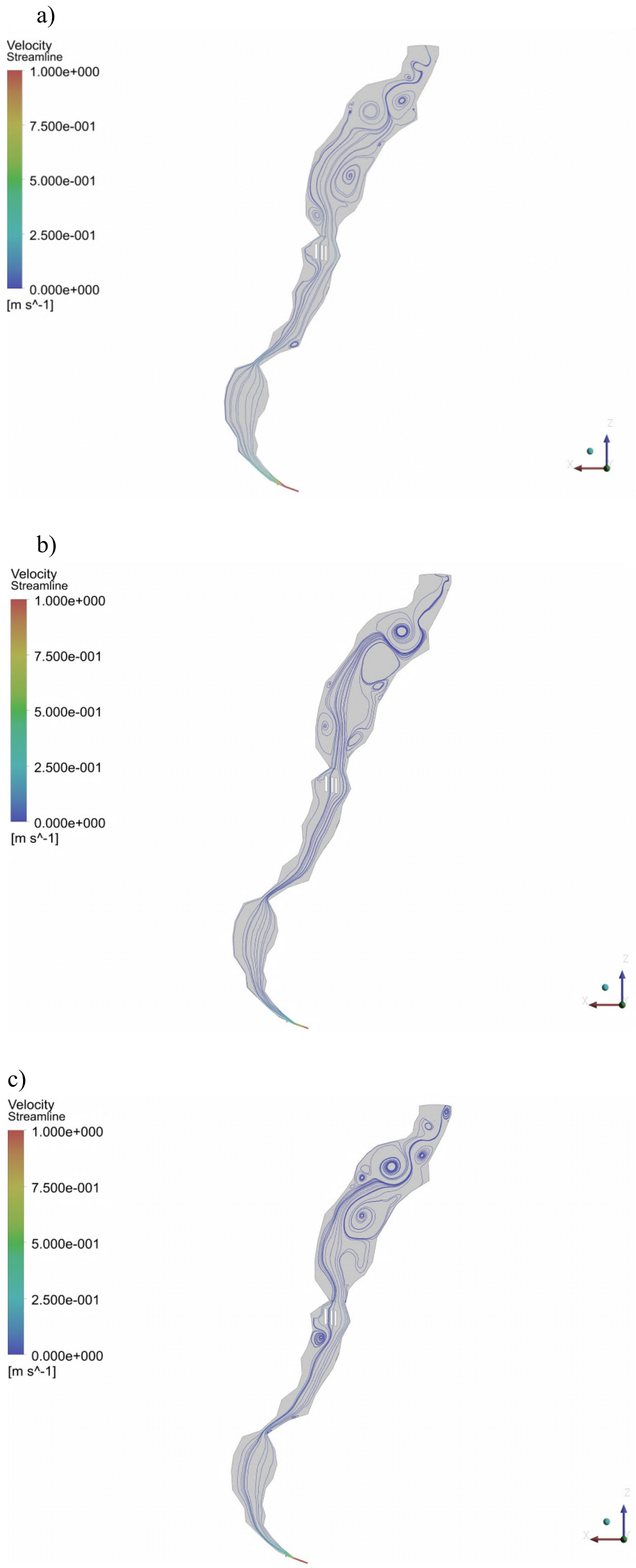

Fig. 6. Surface streamlines in the Sulejow reservoir in a) March b) July c) December (Zieminska-Stolarska, 2014).

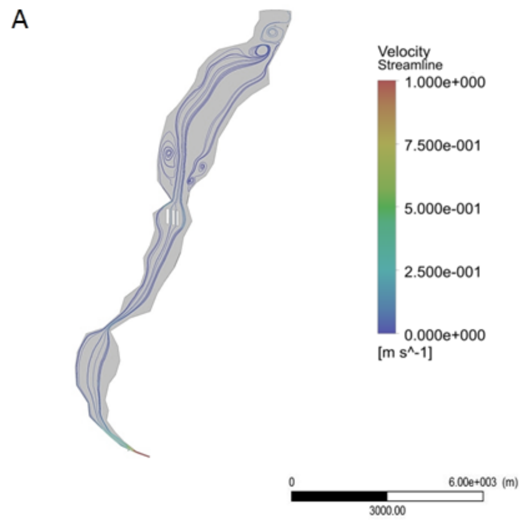

Fig. 7. Comparison of velocity field (m/s) in the Sulejow reservoir in October 2007 under wind and under no wind.

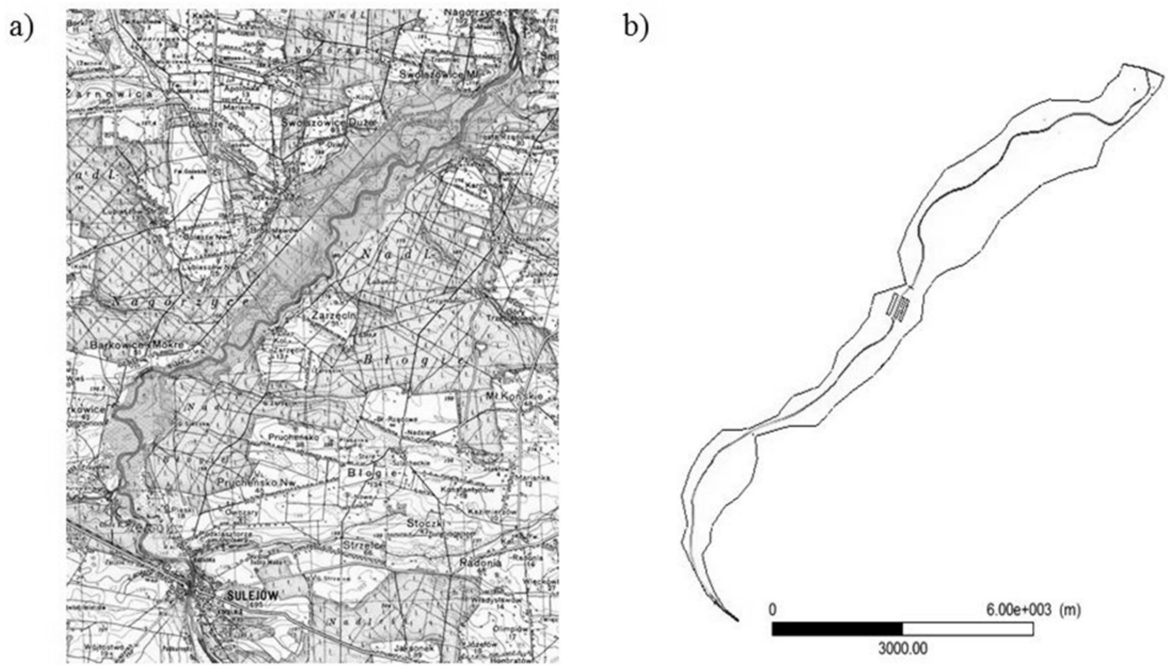

Fig. 8. Comparison of main current in a) real conditions, b) CFD model.

CONCLUSIONS

The objective of the present work was to develop and validate a 3-D CFD numerical model for simulating flow through 17-km-long dam reservoir of a complex bathymetry. As a result of work, the following main conclusions can be presented:

1. A single-phase 3-D CFD model of flow hydrodynamics in large water bodies was developed and verified, using the example of the Sulejow reservoir with accurate depiction of basin bathymetry. The results indicate that the flow field in the Sulejow reservoir is transient in nature, with visible swirl flows in the lower part of the lake. Recirculating zones with a size of up to half a km across may increase water retention time in this region.
2. The results of simulations confirm the pronounced effect of wind on the development of water circulation zones in the reservoir, which might affect the accumulation of nutrients in the epilimnion layer and result, e.g., in algae bloom.
3. The results of the simulations are complementary to the direct measurements of the surface-water quality. Their practical use could be to assist in the development of forecasts of water quality and in making decisions concerning complex changes in water management of the reservoir.

The resulting model is accurate, and the methodology developed in the framework of this study could be applied to all types of storage reservoir configurations, characteristics, and hydrodynamics conditions.

REFERENCES

Andersson, A.G., Andreasson, P., Lundström, T.S., 2013. CFD-modeling and validation of free surface flow during spilling of reservoir in down-scale model. Engineering Applications of Computational Fluid Mechanics, 7, 1, 159–167.

Dargahi, B., 2004. Three-dimensional flow modeling and sediment transport in the River Klaralven. Earth Surface Processes and Landforms, 29, 7, 821-852.

Demuren, A.O., 1993. A numerical model for flow in meandering channels with natural bed topography. Water Resources Research, 29, 4, 1269–1277.

Dubnyak, S., Timchenko, V., 2000. Ecological role of hydrodynamic processes in the Dnieper reservoirs. Ecological Engineering, 16, 1, 181–188.

Falconer, I.R., 1999. An overview of problems caused by toxic blue-green algae (Cyanobacteria) in drinking and recreational water. Environmental Toxicology, 14, 1, 5–12.

Fischer-Antze, T., 2005. Assessing river bed changes by morphological and numerical analysis. Ph.D. Thesis. Vienna University of Technology, Vienna, 179 p.

Gambit User`s Guide 2.2.30, 2004.

Kennedy, M.G., Ahlfeld, D.P., Schmidt, D.P., Tobiason, J.E., 2006. Three-dimensional modeling for estimation of hydraulic retention time in a reservoir. Journal of Environmental Engineering, 132, 9, 976–984.

Kriš, J., Hadi, G.A., 2010. Improvement performance of Al-Wathba settling tank by a computational fluid dynamics model. J. Hydrol. Hydromech., 58, 3, 201–210.

Meselhe, E.A., Sotiropoulos, F., Patel, V.C., 1995. Three-dimensional numerical model for open-channels. In: Proceedings of the ASCE Waterpower '95, San Francisco, CA, USA, 3, 2315–2324.

Nichols, R.H., 2008.Turbulence Models and Their Application to Complex Flows. University of Alabama, Birmingham.

Patankar, S.V., 1980. Numerical Heat Transfer and Fluid Flow. Hemisphere Publishing Corporation, Washington, pp. 126–134.

Sándor, B., 2009. PhD thesis book: Three-dimensional analysis of river hydrodynamics and morphology, Budapest.

Shepherd, J.F., Johnson, C.R., 2008. Hexahedral mesh generation constraints. Engineering with Computers, 24, 3, 195–213.

Simons, T.J., 1974. Verification of numerical models of Lake Ontario, Part I, Circulation in spring and early summer, Journal of Physical Oceanography, 4, 4, 507–523.

Sinha, S.K., Sotiropoulos, F., Odgaard, A.J., 1998. Three-dimensional numerical model for flow through natural rivers, Journal of Hydraulic Engineering, 124, 1, 13–24.

Ünes, F., Varcin, H., 2015. Investigation of seasonal thermal flow in a real dam reservoir using 3-D numerical modeling. Journal of Hydrology and Hydromechanics, 63, 1, 38–46.

Versteeg, H.K., Malalasekera, W., 1995. An Introduction to Computational Fluid Dynamics. The Finite Volume Method. Longman Group Ltd., Loughborough, United Kingdom.

Wilcox, D.C., 1998. Turbulence modeling for CFD. DCW Industries Inc., La Canada, CA.

Wu, W.M., Rodi, W.G., Thomas, W., 2000. 3D numerical modeling of flow and sediment transport in open channels. Journal of Hydraulic Engineering, 126, 1, 4–15.

Wüest, A., Lorke, A., 2003. Small-scale hydrodynamics in lakes. Annual Review of Fluid Mechanics, 35, 373–412.

Ziemińska-Stolarska, A., Zbiciński, I., Imbierowicz, M., Skrzypski, J., 2013. Waterpraxis as a tool supporting protection of water in the Sulejow. Desalination and Water Treatment, 51, 1–13.

Zieminska-Stolarska, A., 2014. Analysis of the impact of changes in the flux of biogenic substances on water eutrophication in the Sulejów Reservoir. PhD Thesis. University of Technology, Lodz.

Zinke, P., Reidar Bøe Olsen, N., Bogen J., Rüther, N., 2010. 3D modelling of the flow distribution in the delta of Lake Øyeren, Norway. Hydrology Research, 41, 2, 92–103.

Received 10 February 2015
Accepted 8 June 2015

Note: Colour version of Figures can be found in the web version of this article.