ICHE 2014, Hamburg - Lehfeldt & Kopmann (eds) - © 2014 Bundesanstalt für Wasserbau ISBN 978-3-939230-32-8

Multi-dimensional Numerical Simulation of Wind-induced Flow and Transport Processes in an Urban Water System

A. Jourieh & R. Hinkelmann
Technische Universität Berlin, Chair of Water Resources Management and Modeling of Hydrosystems, Germany

ABSTRACT: The Unterhavel water system in Berlin has been chosen to numerically investigate wind-induced two- and three-dimensional flow and transport processes. Mean wind and a storm have been considered. The results show that the wind has a considerable influence on the flow field as well as on the tracer distribution. Wind generates complex horizontal and vertical circulations with different flow directions at the surface and the bottom in parts of the water systems.

Keywords: Wind-induced flow, Numerical simulation, 2D and 3D flow and transport, TELEMAC modeling system

1 INTRODUCTION

Urban water systems are generally stressed by various contaminations coming from, for example, treated waste water, industry, households, runoff from streets. They consist of natural (e.g. rivers, lakes, groundwater) and technical compartments (e.g. wastewater treatment plants, sewer systems) which are often in strong interactions.

The major advantage of using multi-dimensional numerical models is their capability of giving details about the flow and transport processes in two or three directions. These models also help to understand the physical, chemical and biological processes in rivers, estuaries, lakes, and coastal waters. A three-dimensional model should be chosen for example in the case of density and wind-induced flow (e.g. in this study). If only the horizontal variability of the flow is important (e.g. flood modelling), a two-dimensional approach is generally sufficient.

In this contribution, the Unterhavel river in Berlin, Germany has been chosen. This system consists of a shallow lake (Wannsee) and some channels (see Fig. 2), and it has, of course, limnic character being characterized by slow flow velocities and temporarily inflow of treated waste water. For the numerical simulation of flow and transport processes, the modeling system TELEMAC has been applied to solve the 2D and 3D flow and transport equations using unstructured grids. Taking into account various conditions (e.g. low, mean and high discharge, tracer injection), a 2D and 3D model was set-up and hydrodynamic as well as transport studies were computed, in order to investigate the impact of the mean and strong wind on the flow and transport processes (Jourieh 2014).

2 MODELING SYSTEM

The TELEMAC modelling system was used to set up a 2D and 3D model. It has been developed by the National Hydraulics and Environment Laboratory (Laboratoire National d´Hydraulique et Environnement - LNHE) of the Research and Development Directorate of the French Electricity Board (EDF-DRD) and is now managed and further developed by a consortium of core organizations: Artelia (formerly Sogreah, France), Bundesanstalt für Wasserbau (Germany), Centre d'Etudes Techniques Maritimes et Fluviales (France), Elec tricité de France R&D (France) and HR Wallingford (United Kingdom). TELEMAC is mainly based on the FEM and it offers various stabilisation techniques and fast solvers. The FEM using triangular elements in 2D enables a good approximation of complex boundaries, islands etc. The grid can be refined in areas of special interest such as around narrow channels.

2.1 *2D simulation of flow and transport*

For the 2D simulation, TELEMAC-2D has been chosen to solve the shallow water equations together with the corresponding transport equation for a conservative tracer (Hervouet 2007, Hinkelmann 2005). TELEMAC-2D solves the following equations:

2D shallow water equations:

$$\frac{\partial h}{\partial t} + \vec{u}\, grad\, h + h\, div\, \vec{u} = S_h \tag{1}$$

$$\frac{\partial u}{\partial t} + \vec{u}\, grad\, u = \frac{1}{h}\, div\,(h\, v_t\, grad\, u) - g\frac{\partial z}{\partial x} + S_x \tag{2}$$

$$\frac{\partial h}{\partial t} + \vec{u}\, grad\, v = \frac{1}{h}\, div\,(h\, v_t\, grad\, v) - g\frac{\partial z}{\partial y} + S_y \tag{3}$$

2D tracer conservation equation:

$$\frac{\partial T}{\partial t} + \vec{u}\, grad\, T = \frac{1}{h}\, div\,(h\, v_T\, grad\, T) + S_T \tag{4}$$

where h = water depth (m), $\vec{u}$ = velocity vector (m/s), u, v = velocity components (m/s), T = tracer concentration (g/l), S_h = source or sink of fluid (m/s), υ_t = turbulent viscosity (m²/s), g = gravitational acceleration (m/s²), z = free surface elevation (m), S_x, S_y = source or sink terms of momentum (e.g. bottom friction, wind, see 2.3), υ_T = turbulent diffusivity (m²/s), S_T = source or sink of tracer (g/(ls)), x, y = horizontal space coordinates (m), t = time (s).

The same discretisation methods, solvers and numerical parameters have been chosen for the 2D simulation of hydrodynamics and transport. The Methods of Characteristics (with fractional step) and the conservative scheme with SUPG method have been chosen to solve the advection step for the velocity components and the water depth respectively. The BiCGSTAB (Biconjugate Gradient Stabilized Method) was chosen as the solver.

The main results at each node of the computational mesh are the depth of the water, the depth-averaged velocity components and tracer concentrations.

2.2 *3D simulation of flow and transport*

TELEMAC-3D solves the three-dimensional shallow water equations together with a corresponding transport equation for a conservative tracer.

3D hydrodynamic equations:

$$div\, \vec{u} = q \tag{5}$$

$$\frac{\partial u}{\partial t} + \vec{u}\, grad\, u = div\, \underline{\underline{\upsilon_t}}\, grad\, u + \frac{1}{\rho_0}\left(f_x - \frac{\partial P}{\partial x}\right) \tag{6}$$

$$\frac{\partial v}{\partial t} + \vec{u}\, grad\, v = div\, \underline{\underline{\upsilon_t}}\, grad\, v + \frac{1}{\rho_0}\left(f_y - \frac{\partial P}{\partial y}\right) \tag{7}$$

$$\frac{1}{\rho_0}\frac{\partial P}{\partial z} + g = 0 \Leftrightarrow P = P_{atm} + \rho_0\, g\,(z_s - z) + \rho_0\, g \int_z^{Z_s} \frac{\Delta\rho}{\rho_0} dz \tag{8}$$

3D tracer transport equation:

$$\frac{\partial T}{\partial t} + \vec{u}\, grad\, T = div\,(\underline{\underline{\upsilon_T}}\, grad\, T) + Q \tag{9}$$

$$\underline{\underline{\upsilon}} = \begin{vmatrix} \upsilon_h & 0 & 0 \\ 0 & \upsilon_h & 0 \\ 0 & 0 & \upsilon_v \end{vmatrix} \quad ; \quad \underline{\underline{\upsilon}}_T = \begin{vmatrix} \upsilon_{Th} & 0 & 0 \\ 0 & \upsilon_{Th} & 0 \\ 0 & 0 & \upsilon_{Tv} \end{vmatrix}$$

where $\vec{u}$ = velocity vector (m/s), u, v, w = velocity components (m/s), x, y, z = space coordinates (m), t = time (s), z_s = vertical component of the water surface (m), ρ_0 = reference density (kg/m³), $\Delta\rho$ = density difference (kg/m³), P = pressure (Pa), P_{atm} = atmospheric pressure (Pa), f_x, f_y = momentum source terms (kg/(s²m²)), Q = source or sink of tracer (kg/m³s), v_t = turbulent viscosity tensor (m²/s), $\underline{\upsilon_t}$ = horizontal turbulent viscosity (m²/s), υ_v = vertical turbulent viscosity (m²/s), v_T= turbulent diffusivity tensor (m²/s), $\upsilon_{T\,h}$ = horizontal turbulent diffusivity (m²/s), $\upsilon_{T\,v}$ = vertical turbulent diffusivity (m²/s).

TELEMAC-3D basic algorithm is spilt into three computational steps called fractional step method (Hervouet 2007, Hinkelmann 2005). In the first step, the advection is computed with the Method of Characteristics. In the second step, the diffusion and some right-hand side terms are determined with the FEM, the transport simulation is then finished. The third step computes the free surface, continuity and pressure terms using TELEMAC-2D. The BiCGSTAB (Biconjugate Gradient Stabilized Method) was used as the solver. The main results at each point in the mesh are the velocity in all three directions, the water level and the tracer concentration.

2.3 *Accounting for wind*

In this section it is explained how the wind is accounted for in 2D and 3D. Wind causes flow in the horizontal as well as the vertical direction such as circulations and thus also may have a considerable impact on the transport processes.

2D

In the two-dimensional model, the wind resistance is as part of the momentum source terms S_x and S_y. The resistance of the wind has the following form, neglecting the slope of the free surface:

$$S_{x,wind} = -\frac{1}{h}\rho_{air} c_D v_{ax} \sqrt{v_{ax}^2 + v_{ay}^2} \; ; S_{y,wind} = -\frac{1}{h}\rho_{air} c_D v_{ay} \sqrt{v_{ax}^2 + v_{ay}^2} \tag{10}$$

where v_{ax}, v_{ay} = components of the wind velocity, $\rho_{air}\,[kg/m^3]$ = air density, $c_D\,[-]$ = wind-resistance coefficient.

In the TELEMAC modeling system the following approach is available:

$$\begin{aligned} &|\vec{v}_a| < 5\, m/s \Rightarrow c_D = 0{,}565 \cdot 10^{-3} \\ &5 < |\vec{v}_a| < 19{,}22\, m/s \Rightarrow c_D = (-0{,}12 + 0{,}137\,|\vec{v}_a|) \cdot 10^{-3} \\ &|\vec{v}_a| < 19{,}22\, m/s \Rightarrow c_D = 2{,}513 \cdot 10^{-3} \end{aligned} \tag{11}$$

where $\vec{v}_a\,[m/s]$ = vector of wind velocity

The coefficient c_D hides complex phenomena. In fact, the influence of the wind depends on many effects such as the smoothness of the free surface, the distance over which it acts, the location of the measurement station, etc. Generally, the wind is measured 10 m above the water depth. The wind can be constant in space and time. Often a time series of wind velocity and direction with for example hourly values is used. If data of more than one measurement stations are available, a spatial interpolation is necessary.

3D

In the three-dimensional model, the wind resistance is taken into account through a boundary condition similar as in 2D. The profile illustrated in Figure 1 shows the effect of a wind-induced circulation. The wind aligns with the flow at the surface imposing return flow areas along the bottom (Lui and Perez 1971, Shanahan et al. 1981). A similar velocity profile was expected in the Wannsee.

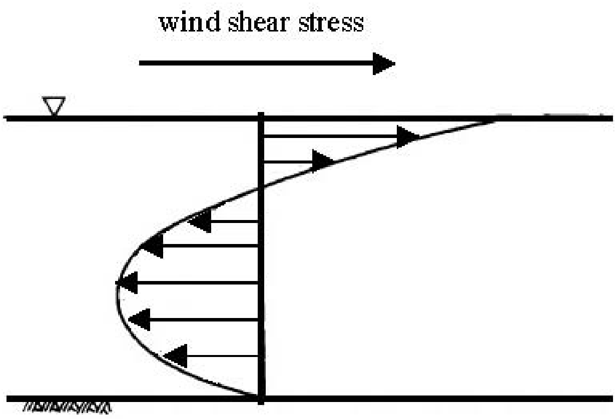

Figure 1. Theoretical wind-induced velocity profile (after Lui and Perez 1971, Shanahan et al. 1981)

3 STUDY AREA AND COMPUTATIONAL DOMAIN

The study area is the Unterhavel river in Berlin, Germany (Fig. 2). The Unterhavel river flows from north to south along Berlin's western boundary. This system consists of shallow lakes (e.g. Wannsee, Glieniker See and Jungfernsee) and small islands (e.g. Pfaueninsel). Further, the river Spree joins the Unterhavel in Spandau (north), while the Teltow channel (which is characterized as channel-like river) joins it in the south east. The Unterhavel river is slow-flowing.

The computational domain is located between Pichelssee in the north, where the Spree joins the river Havel, the junction point with the Teltow channel in the south east, the Jungfernsee and Glieniker See in the south west (Fig. 2). The whole domain has a mean depth of 5.5 m, a maximum depth of 9.5 m, an area of 30 km² and a volume of 155 million m³ (SenGUV 2006).

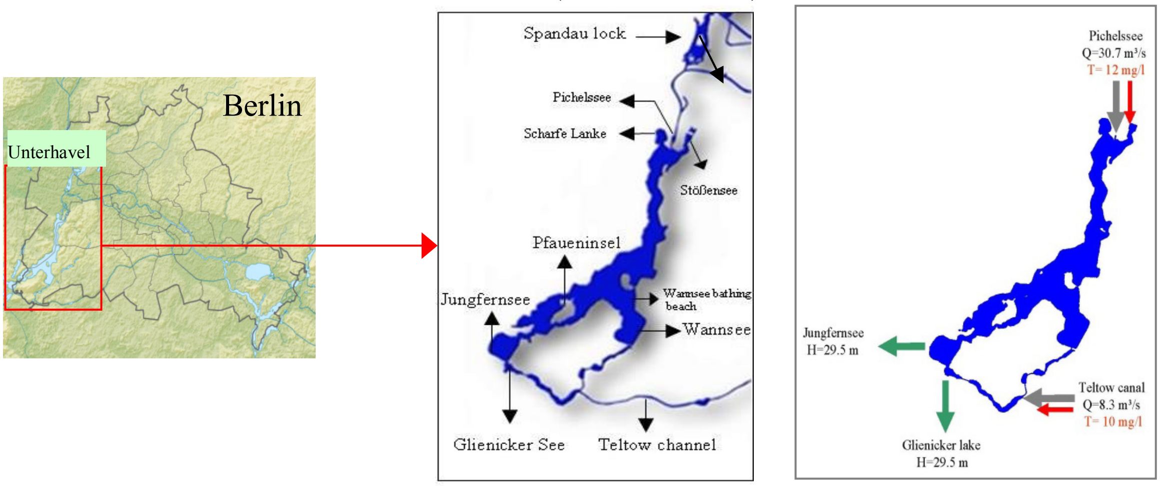

Figure 2. Study area (left); computational domain (middle); boundary conditions (right)

4 RESULTS

4.1 *Model setup, initial and boundary conditions*

For the preprocessing, an unstructured grid has been generated using the grid generator JANET (Smile consult 2009). The grid was generated in such a way that the edge size is in the range of 50 m in areas, where the Unterhavel is wide, while the narrow passages are highly resoluted, i.e. the edge size is gradually decreased and the minimum edge length is 5 m. The total number of 2D elements in the grid is 99110 and the total number of 2D nodes is 71300.

To mesh the 3D domain with the TELEMAC modeling system, we just need to take the 2D grid for the horizontal domain and then duplicate it along the vertical in a number of curved surfaces known as "planes". The σ-discretization has been chosen for the vertical direction. This method is generally recommended to better approximate the processes close to the vertical boundaries, for example to refine close to the bottom and the water surface as done here. The 3D mesh consisted of 17 planes, 1212100 nodes and 1540870 prisms.

The 2D and 3D simulations started with a prescribed initial water level of 29.5 m (SenGUV 2008) equal in all points of the computational mesh and initial flow velocities were zero. The following boundary conditions have been selected for the hydrodynamic simulations (Fig 2, right):

i) Neumann-boundary conditions: At two inflow boundaries, discharges were imposed. They were set as constant values representing mean discharge conditions (SenGUV 2008). A value of 30.7 m^3/s was imposed as inflow boundary condition at Pichelssee and 8.3 m^3/s as inflow boundary condition at Teltow channel.

ii) Dirichlet-boundary conditions: Water levels were prescribed at outflow boundaries at Jungfernsee and Glienicker See. The prescribed constant water level is equal to 29.5 m. As the water level variations are small, choosing a constant value is justified.

The horizontal turbulent viscosity was set to 0.0001 m^2/s and the friction coefficient of Manning-Stickler was set to 30 $m^{1/3}/s$. For the 2D and 3D simulations of hydrodynamics and transport a time step of 3 seconds was chosen corresponding to a Courant number of about 6.

The computations for the hydrodynamics were carried out with constant boundary conditions until steady state was reached and then the simulation of transport has been carried out. In these simulations, the spreading of a passive tracer was investigated. The COD (chemical oxygen demand) which is a representative component of treated waste water, has been considered as passive and conservative tracer.

As initial conditions, the tracer concentration was assumed to be 0 mg/l and the tracer's turbulent diffusivity was set to 0.0001*m²/s* (as the turbulent viscosity). The mean COD concentration record of seven years (2000-2007) was determined and set to 12 mg/l at the upper open boundary (Pichelsee) and to 10 mg/l at the lower open boundary (Teltow channel) (SenGUV, 2008). It was expected that the duration of the inflow of the tracers is 10 hours.

Strong storms in the Unterhavel generally occurred during the winter months coming from southwest and west. The climatological winds from 10 years collected data of hourly values at the station Wannsee have been considered. Two scenarios are computed in the following:

i) South wind with a velocity of 5.7 m/s representing mean conditions.

ii) West wind with a velocity of 15.5 m/s, representing the maximum conditions.

The transport simulations have been carried out for 120 days, and then west and south wind were imposed for 5 hours.

4.2 *2D results*

The 2D flow field and tracer concentration distribution are presented in the Wannsee as well as its bathing beach for the cases without wind and with south wind (mean conditions, Fig. 3). As the flows are in opposite direction for the case with and without wind, this has a considerable influence on the tracer concentration.

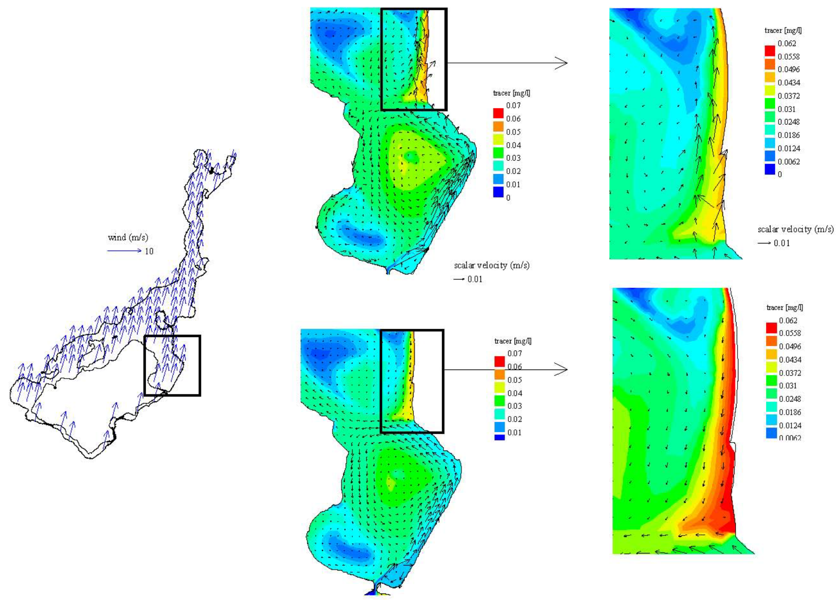

Figure 3. Wind field (left), 2D flow field and tracer concentration at Wannsee with south wind (top) and without wind (bottom), after 5 hours

4.3 *3D results*

Figure 4 (A) shows the velocity field in a horizontal section T-T (shown in the same figure, left bottom), while Figure 4 (B) shows this velocity field in a vertical section P-P (shown in the same figure, left bottom). Two horizontal circulations appear in the horizontal section 0.5 m under the free surface (A), while the wind-induced vertical circulation can be observed in the upper part of the vertical section (B).

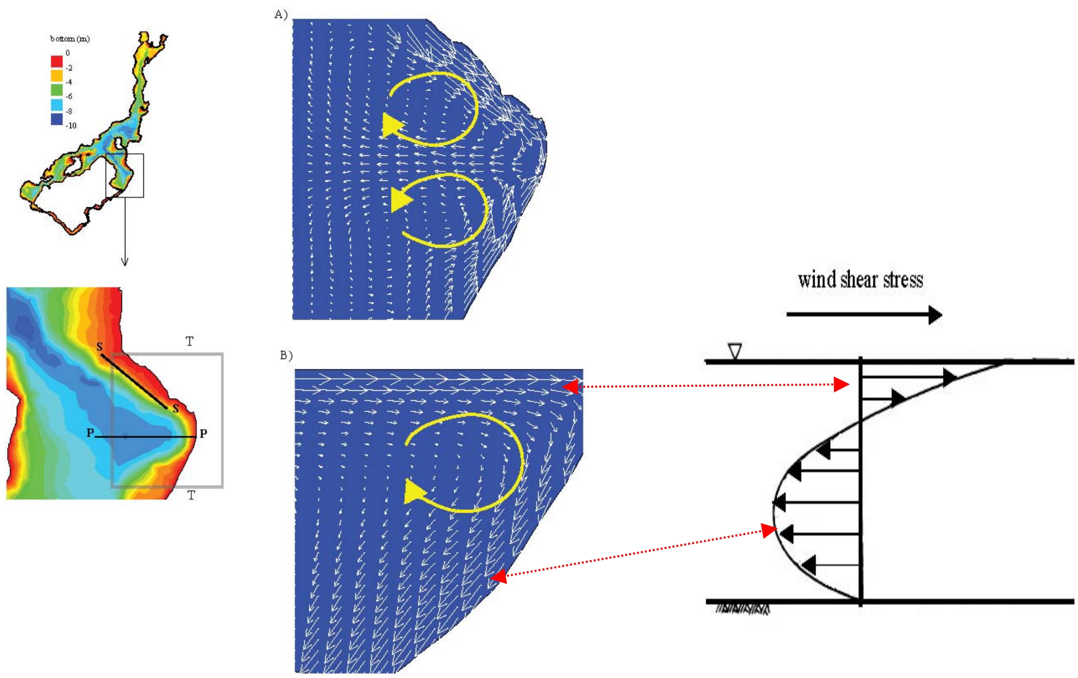

Figure 4. Velocity field induced by west wind after 5 hours, A: horizontal circulation at section T-T, B: vertical section at section P-P

In Figure 5 we see the flow field and tracer distribution in two vertical sections. There are considerable differences in the velocities and the tracer concentrations when the cases with and without wind are compared. The influence of the wind is significant in the upper part of the water depth, where the flow velocity on the surface is in the direction of the wind and the velocity close to the bottom is in the opposite direction.

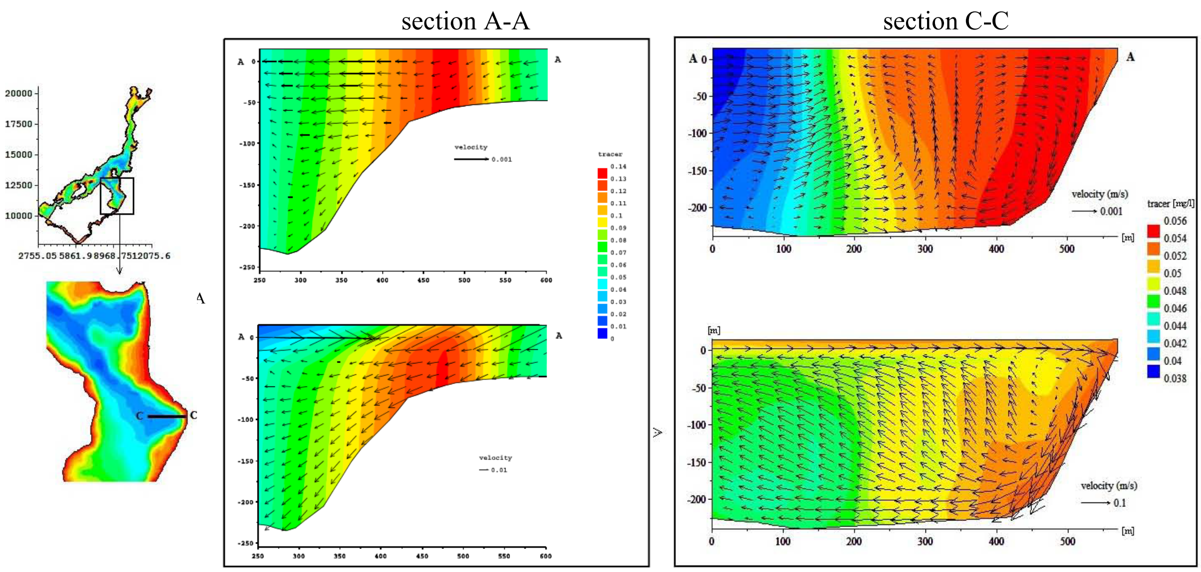

Figure 5. Velocity field and tracer distribution in vertical sections A-A and C-C after 5 hours with west wind (top: without wind, bottom: with west wind)

Figures 6 and 7 show the velocity field in three horizontal sections (at the free surface, at mid depth and at the bottom). Overall, distinct three-dimensional velocity fields occur. With these figures we can demonstrate, that the velocities at the free surface are strongly wind dependent and have the direction of the wind. The west wind forces the surface water to move eastwards (Fig. 6), while the south wind induces it to move northwards (Fig. 7). Moreover, the velocities decrease with increasing water depth, and in the deeper layers horizontal circulations occur. Further, it can be observed that the velocity at the free surface is smaller for the south wind compared to the west wind. The reason is that the west wind is about 3 times higher than the south wind.

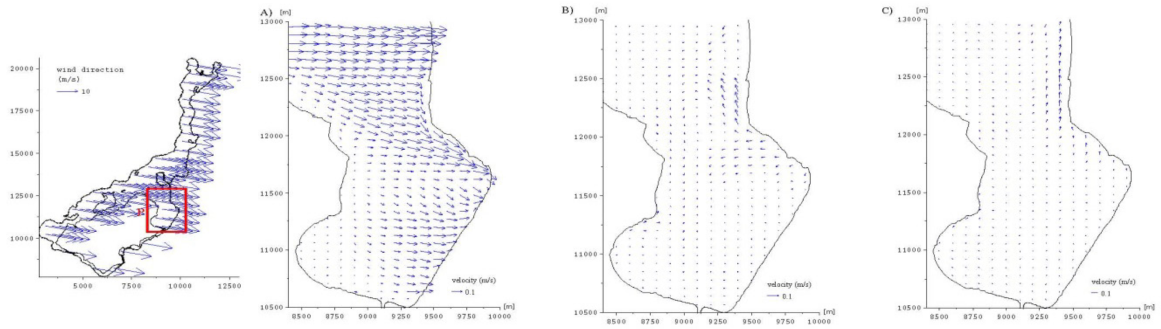

Figure 6. Flow field for west wind at three horizontal levels at Wannsee: A) free surface, B) mid-depth, C) bottom, after 5 hours

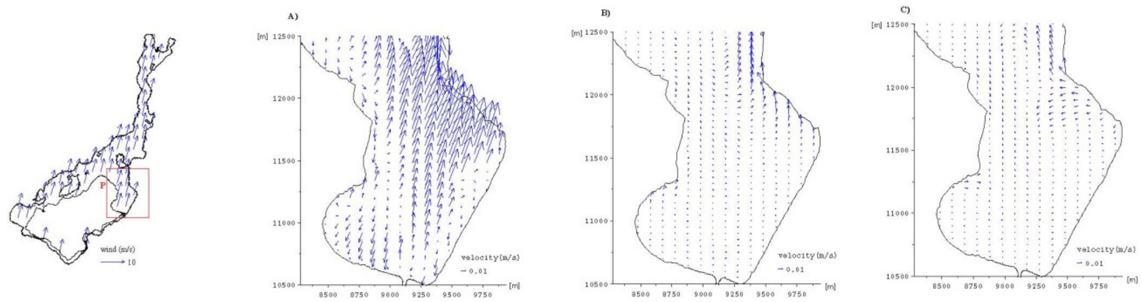

Figure 7. Flow field for south wind at three horizontal levels at Wannsee: A) free surface, B) mid-depth, C) bottom, after 5 hours

5 CONCLUSIONS

In this paper, two- and three-dimensional numerical simulations of wind-induced flow and transport processes in the Unterhavel river in Berlin, Germany are presented. Mean wind and a storm have been considered. The 2D and 3D results show that the wind has a considerable influence on the flow fields as well as the tracer distributions. Complex horizontal and vertical circulations occur. In same cases the velocity follows the direction of the wind at the surface, while the velocity is in the opposite direction at the bottom.

REFERENCES

Hervouet, J. M. (2007): Hydrodynamics of Free Surface Flows - Modelling with the Finite Element Method. John Wiley and Sons Ltd

Hinkelmann, R. (2005): Efficient Numerical Methods and Information-Processing Techniques for Modeling Hydro- and Environmental Systems, Vol. 21, Springer-Verlag, Berlin, Heidelberg, Germany

Jourieh, A. (2014): Multi-dimensional Numerical Simulation of Hydrodynamics and Transport Processes in Surface Water Systems in Berlin. Dissertation, School VI - Planning Building Environment, Technische Universität Berlin

Lui, H., Perez, H. (1971): Wind-induced Circulation in shallow water. Journal of the Hydraulics Division, Vol. 97, No. 7, pp. 923-935

Shanahan, P., Harleman, D., Somlyody, L. (1981): Modeling Wind driven Circulation in Lake Balaton. Collaborative paper/International Institute for Applied Systems Analysis, Working Paper WP-81-7

SenGUV (2008): Senatsverwaltung für Stadtentwicklung und Umweltschutz: Umweltatlas. Besondere Mitteilung zum Gewässerkundlichen Jahresbericht für Berlin und Umland: Berlin

Smile consult GmbH (2009): www.smileconsult.de