Computer modelling of technogenic thermal pollution zones in large water bodies

Ya N Parshakova\textsuperscript{1}, T P Lyubimova\textsuperscript{1, 2}
\textsuperscript{1} Institute of Continuous Media Mechanics UB RAS, Acade. Koroleva 1, Perm 614013, Russia
\textsuperscript{2} Perm State University, Bukireva 15, Perm 614990, Russia
E-mail: parshakova@icmm.ru

Abstract. In the present work, the thermal pollution zones created due to discharge of heated water from thermal power plants are investigated using the example of the Permskaya Thermal Power Plant (Permskaya TPP or Permskaya GRES), which is one of the largest thermal power plants in Europe. The study is performed for different technological and hydrometeorological conditions. Since the vertical temperature distribution in such wastewater reservoirs is highly inhomogeneous, the computations are performed in the framework of 3D model.

1. Introduction

The density stratification effects caused by inhomogeneities of temperature and mineralization fields can play an important role in the formation of both hydrological and hydrochemical regimes of surface water bodies. However, they can not be described in the framework of hydrodynamical models based on the shallow water approximation. Such effects can be accurately described only using the hydrodynamic models in three-dimensional formulation.

The significance of the problem under consideration is confirmed by the fact that it came into the view of researchers as early as in 30-ies of the XX century which gave an impetus to the development of the first applied models of “plane” hydrodynamics for its solution [1,2]. There are a number of factors why finding solutions to these problems is a challenging task. Among these are the fractality of morphometry of natural water bodies, considerable difference in the scales of natural and technological parameters, essential variability of hydrometeorological factors.

Earlier, the restricted efficiency of the computational means was one of the biggest limiting factors. That is why, starting from the pioneering works of N.M Bernadskii [1,2] and until recently, the most widely used approach to these problems was 2D modelling based on the shallow water approximation (see, for example [3-7]). In [3], the authors outline the methods of computational hydrodynamics which are based on 1- and 2-D approaches. The compact finite difference methods are described in [4]. In [5], a model of sea ice motion taking into account the atmospheric and water flows due to temperature inhomogeneities is constructed based on the data of Arctic and Antarctic regions. A 2D-numerical simulation of the spread of thermal pollution in the coastal area of the Red Sea is performed in [6]. An effect of thermal pollution produced by Iran thermal power plant is numerically investigated in [7] with the use of 2D approach. The solution is constructed for three different scales - 25 km x 25 km, 12 km x 12 km, 7 km x 7 km. Consideration is given to different scenarios of meteorological and season conditions. However, numerous data of field observations strongly suggest the need for revising the formulation which is based on the two-dimensional representation of the examined fields.
and homogeneity of the depth distribution of water temperature. Therefore, to obtain more adequate results, it is necessary to change to 3D models.

In works [8-10], a three dimensional numerical simulation of turbulent mixing of water masses of different temperatures was carried out based on the solution of the Navier-Stokes equations and LES model of turbulence. Computations were done with the use of a three – stage scheme with parallelization. A dependence of the computing time on the number of grid nodes and processors was obtained. The computational aspects of the LES model of turbulence were discussed in [11-12].

The authors of the present paper developed hydrodynamical models of surface water bodies, which are based on 3D numerical simulations [13-15]. In the present paper, with reference to the Permskaya thermal power plant (TPP) (Permskaya GRES), we investigated the temperature fields generated due to discharge of heated waters depending on technological and hydrometeorological parameters. It is constructed 3D model for the domain of 10 km which includes the intake and discharge channels of the Permskaya TPP.

2. Computational technique.

The 3D hydrodynamical model was built for the reservoir part of linear dimensions of 1000 m adjacent to the Perm TPP and including the points of water intake and water discharge. Software package ANSYS Fluent was used for the 3D simulations at the computer cluster URAN of the IMM UB RAS. The problem was solved in the framework of a non-stationary non-isothermal approach on the basis of the $k – \varepsilon$ model describing turbulent pulsations. We implement the Reynolds-averaged Navier-Stokes equations:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = \frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right) \right] +$$

$$+ \frac{\partial}{\partial x_j} \left[ \mu_i \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \left( \rho k + \mu \frac{\partial u_i}{\partial x_i} \right) \delta_{ij} \right] + \rho g_i \tag{2}$$

Here, $\rho$ is the density, $x_i$ are coordinates (we use Cartesian coordinate system), $u_i$ are the velocity components, $\mu$ is the kinematic viscosity, $\mu_t$ is the turbulent viscosity.

The turbulence kinetic energy $k$ and rate of its dissipation $\varepsilon$ are obtained from the following transport equations:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left[ \mu + \frac{\mu_t}{\sigma_k} \right] \frac{\partial k}{\partial x_j} + G_k + G_s - \rho \varepsilon \tag{3}$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left[ \mu + \frac{\mu_t}{\sigma_{\varepsilon}} \right] \frac{\partial \varepsilon}{\partial x_j} + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{2\varepsilon} G_s) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \tag{4}$$

In equations (3)-(4), $G_k$ represents the generation of turbulence kinetic energy due to the mean velocity gradients $G_k = \mu S^2$ where $S$ is the modulus of the mean strain rate tensor, defined as $S = \sqrt{2S_{ij} S_{ij}}$, $S_{ij} = 0.5 \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)$, $G_s$ is the turbulence kinetic energy due to buoyancy, which is calculated as

$$G_s = g_i \left( \frac{\beta \mu_t}{\Pr_t} \right) \frac{\partial T}{\partial x_i} \tag{5}$$

where $\mu_t$ is the turbulent viscosity determined as: $\mu_t = \rho C_{\mu} k^2 / \varepsilon$, where $C_{\mu}$ is a constant.
Simulation of turbulent heat transfer is performed using the Reynolds model similarly to that of
turbulent momentum transfer. Hence, the equation of energy is expressed as

$$\frac{\partial}{\partial t} (\rho E) + \frac{\partial}{\partial x_i} \left[ u_i (\rho E + p) \right] = \frac{\partial}{\partial x_j} \left( k_{\text{eff}} \frac{\partial T}{\partial x_j} + u_i (\tau_{ij})_{\text{eff}} \right)$$  \hspace{1cm} (6)

where \( E = c_h + \frac{p}{\rho} \) denotes total energy, \( h = C_p T \) denotes system enthalpy, \( k_{\text{eff}} \) denotes effective
thermal conductivity, and \( (\tau_{ij})_{\text{eff}} \) is the stress tensor deviator defined as

$$ (\tau_{ij})_{\text{eff}} = \mu_{\text{eff}} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \mu_{\text{eff}} \frac{\partial u_k}{\partial x_k} \delta_{ij} $$  \hspace{1cm} (7)

The model constants \( Pr_t, C_{1t}, C_{2t}, C_{\mu}, \sigma_k \) and \( \sigma_e \) were taken to have the following values [16]:
\( Pr_t = 0.85 \), \( C_{1t} = 1.44 \), \( C_{2t} = 1.92 \), \( C_{\mu} = 0.09 \), \( \sigma_k = 1.0 \), \( \sigma_e = 1.3 \).

The spatial discretization scheme of second-order accuracy was applied. Simulation of temporal evolution was carried out using an explicit second-order scheme.

Boundary conditions imposed on the edges of the computation domain were as follows. At the bottom and at the banks the no-slip conditions and fixed temperature were imposed:
\( u_1 = u_2 = u_3 = 0, \quad T = T_0 \),
At the inlet to the computational domain the main flow velocity was assumed to have one nonzero component which was taken to be constant over the inlet cross-section, and the temperature was assumed to be equal to the background temperature of the reservoir water: \( u_i = V_1, \quad T = T_0 \),
At water intake and water discharge points, the water velocity and temperature were taken to be constant: \( u_i = V_{\text{intake}}, \quad T = T_0 \) at the input channel entrance and \( u_i = V_{\text{discharge}}, \quad T = T_{\text{discharge}} \) at its outlet.

The upper boundary of the fluid was assumed to be free and non-deformable, and the effect of wind is taken into account by specifying the tangential stresses according to Ekman formula [17], the linear thermal transfer law was applied accounting for the surface heating from the surrounding air, the heat transfer coefficient was chosen from the analysis of the obtained in-situ measurement data.

A computational grid was generated with Gambit 2.4 package of ANSYS Fluent. The number of nodes through the depth of the computational domain was taken to be 21. The non-uniform mesh was constructed using the bottom morphometric data obtained from in-situ measurements in 2014. In a horizontal direction, the computational mesh consisted of tetragonal elements distributed uniformly along the entire length, with the characteristic linear size of 20m. The mesh included 400 hundred thousands of nodes.

To adapt the morphological data available in a coordinate-depth format to the capabilities of the mesh generator Gambit, the reservoir bottom morphology was represented as a set of simple geometrical objects of some specified resolution, which were then introduced into the file.

A code has been written to produce a batch file for the Gambit grid generator of the ANSYS Fluent package from the data array describing the reservoir bottom morphology. Thus, the complex geometry of the computational domain is realized. The proposed code is of general character and is applicable to the construction of similar geometries and in other tasks.

Some preliminary studies have been conducted to get a numerical solution to the problem of interest using ANSYS Fluent. First, we obtained a stationary solution to the examined problem ignoring wind effects. This solution takes several hundreds of iterations to rapidly converge. Then, the problem was solved using a non-stationary approach with the time step of 2 seconds in the presence of wind effects.
3. Results of calculations

3D numerical modeling was conducted for different scenario conditions of the impact of the Perm TPP on the Kamskii reservoir. The calculation results for two of these critical scenarios which are of most interest from the ecological and technological points of view are given below. According to the current regulatory documents determining the permissible loads on fishery facilities and the Kama Reservoir, it is not allowed to increase the water temperature to more than 28°C in summer.

![Figure 1](image)

**Figure 1.** Map with the control points location. Each point is denoted by two numbers, of which the first corresponds to the number of a control line, and the second to the point number along the line.

3.1 First scenario

The goal of this study is to assess the zone of thermal pollution in the "normal" mode of operation of Permskaya GRES and for the most probable meteorological and hydrological conditions. Technological parameters: 3 operating power units (2 stream-power units and 1 combined-cycle unit under construction), flow rate of discharged water - 42.5 m³/sec, and discharge water temperature – 32.4 °C.

Under these conditions, the formation of the flow structure in the surface layer is governed by the wind action and the discharge of heated waters from the Permskaya GRES. In general, the flow acquires a unidirectional structure rather quickly. When approaching the mixing zone, the surface flow velocities are ~0.1-0.15 m/s. The flows formed by the discharge of heated waters have a velocity of about 0.2-0.4 m/sec (Fig. 2a). The wind practically does not shift the stream to the bank of the river, intensive propagation of the heated waters all over the reservoir takes place, due to which there is intensive mixing of waters up to the depth of 3-4 meters is observed (Figure 3). Figures 3a-d show the depth-distribution of the water temperature in the control points 1, 2, 3 and 11 for the first scenario 1.

As one can see from Figures 11a, b, c, in the control sections 1, 2, 3, the flow of warm water is directed downstream, there is a significant heterogeneity of the depth-distribution of temperature. At the entrance to the working channel (Fig. 11 g, control point 11), temperature inhomogeneities in depth and width of the reservoir are not observed. However, along the left bank, were the outflow channel is located, there is a significant increase in temperature downstream. The area at which the temperature increases with respect to background values by 3 degrees or more is about 1.2 km².

3.2 Second scenario

The goal of calculations for this water discharge variant is to determine the zone of thermal impact in the presence of southern winds at most probable meteorological and technological parameters. Technological parameters are the same as in variant 1. Hydrological and meteorological parameters are the same as in variant 1 except for the wind direction. South-east wind of velocity of 3 m/s and 8 m/s.
Figure 2. Numerical data for the first scenario: (a) velocity vector field in the surface layer of the Kama reservoir, (b) temperature field (°C) in the surface layer of the Kama reservoir (the lines denote the boundaries of temperature rise by 3 and by 5 °C relative to the background temperature)

Under these conditions, a rather complex flow structure is formed, both near the surface and at the depth. This is due to the fact that the runoff, wind and the discharge flows (formed due to the discharges of the Permskaya GRES) have nearly the same velocities, but they are multidirectional. In addition, there are the density effects due to the different temperatures of the streams. Therefore, there is no any distinguished flow direction at depths of up to 5-6 meters.

Figure 3. Depth-distributions of water temperature in the control points for the first scenario: a) the first control point series, b) the second, c) the third, d) the eleventh

In the case of southeastern winds, the most unfavorable situation with the arrival of warm water in the supply channel of Permskaya GRES is observed (Figure 4). In this case, the increase in wind velocity reduces the time-period needed for warm water to reach the supply channel and increases the intensity of the arrival of warm water and the increase in the flow rate exerts the opposite effect on these processes. Decrease in the flow rate increases the probability of warm water arrival to the water intake facilities. As follows from the results of calculations, the reverse flows from the outlet channel to the supply channel are observed only in the upper layer of thickness, as a rule, no more than 2-3 m. In general, when the second scenario is implemented, the maximum thermal pollution effect is observed: the increase in temperature with respect to the background values by 3 and 5 °C occurs, respectively, in the water area of 2.8 and 1.5 km². In the case of significant wind in the direction opposite to the direction of the flow in the river, a three-dimensional vortex is formed within a few hours, the horizontal dimension of which is equal to the distance between the interfaces of the supply and return channels with the reservoir, and the vertical dimension is equal to the depth of the river. The presence of this vortex leads to the motion of warm water against the flow in the river. In this case, less than in a day, warm water reaches the site of the intake of the cooling channel, which is extremely undesirable from a technological point of view. There is also a significant temperature inhomogeneity in depth, and the temperature gradient is greatest near the bottom of the river.
Figure 4. Temperature field (°C) in the surface layer of the Kama reservoir (the lines indicate the boundaries of temperature increase by 3 and by 5 °C relative to the background temperature), (a) SE wind 3 m/s, (b) SE wind 8 m/s

4. Conclusion
The study of hydrodynamic characteristics and the calculation of the propagation of thermal pollution zone in the river basin has been carried out using the example of Permskaya GRES. Numerical modelling based on the three-dimensional model yielded hydrodynamic characteristics of directions and velocities of flow. Calculation of the propagation of thermal pollution zone due to the discharge of heated water from Permskaya GRES were performed for different wind directions. It was found that the arrival of warm water into the supply channel during the ice-free period can be observed when the following conditions are realized: there is fairly strong south-eastern wind (W> 3 m / sec) during the long enough time (t> 12-15 hours); the flow rate of the water discharge in the Kama HPP section is low, Q <1000 m3 / sec; the temperature of the discharged water from the Permskaya GRES water is at least 30 °C; the discharge rate is greater than 40 m3/s.

Acknowledgments
The work was supported by Russian Scientific Foundation (grant No. 17-77-20093).

References
[1] Bernadksii N M 1933 Mater. on hydrology, hydrography and water resource. of the USSR, 20
[2] Bernadksii N, Proskuryakov B 1933 Theory and Practice of Calc.s of the Cooling Pond. 1
[3] Fletcher C A 1988 Special Techniques for Differential Flow Categories 2 493
[4] Tolstykh A I 1990 Compact Difference Scheme and Their Applications to Fluid D. Problems
[5] Parkinson C L 1979 J. Geophys. Res. 84 311
[6] Chieh S H 1987 J. Hydraulic Engineer. 113 (8) 1032
[7] Abbaspour M, Javid A H, Moghimi P, Kayhan K 2005 J. of Envir. Science and Techn. 2 (1) 13
[8] Issakhov A 2011 J. of Physics 318 (4) 42051
[9] Issakhov A 2013 Power, Control and Optimization 239 165
[10] Issakhov A 2014 Scientific World J. doi 10.1155/2014/678095
[11] Tennekes H, Lumley J L 1972 The MIT Press
[12] Lesieur M, Metais O, Comte P 2005 Large Eddy Simulation of Turbulence
[13] Lepikhin A P, Lyubimova T P, Parshakova Y N, Tiunov A A 2012 J. of Min. Science 48 (2) 390
[14] Lyubimova T, Lepikhin A, Konovalov V, Parshakova Ya, Tiunov A 2014 J. Hydrology 508 328
[15] Lyubimova T, Lepikhin A, Parshakova Ya, Tiunov A 2016 J. of Hydrology. 534 579
[16] Launder B E, Spalding D B 1972 Lectures in Mathematical Models of Turbulence
[17] Welander P 1968 Tellus XX I 1