Heat Transfer Predictions of Thermal Discharges in Water Bodies

C.R. MALISKA*, A.F.C. SILVA* e S. POLINA

Depto. Engenharia Mecânica — UFSC
Cx. Postal 476 - 88049 - Florianópolis - SC

J.A.O. PEREZ*

Departamento Ingeniería Mecánica
Universidad de Magallanes - Punta Arenas, Chile

ABSTRACT

The prediction of thermal discharges in water bodies is of great importance in the designing fase of a power plant as well as for predicting the thermal impact in the environment. In this paper it is presented a numerical model which solves the depth-averaged Navier-Stokes and energy equations for arbitrary shoreline of rivers and lakes. The method is applied for real thermal discharges with numerical and experimental results available for comparisons. It is demonstrated that the model developed can predict well the behavior of thermal discharges in shallow waters.

INTRODUCTION

The waste water from the cooling systems of electric power generation plants is normally discharged into lakes and rivers. In the case of the coal fired power plants, besides the environmental impact of the hot water over the biota, there is the potential danger of chemical pollution from ash deposits or from some other pollution source in the plant.

There are several environmental consequences due to thermal discharges in water bodies. Primarily the hot water may destroy some kinds of living organisms encouraging the development of others. The metabolic rate of the living species is strongly dependent of the temperature and the equilibrium between the species may shift to a situation where an undesirable specie may become predominant. The increase in the water temperature, associated with the plume velocity may block the natural travelling path of certain species, thus the struggle for food. Also, eggs and small organisms are swept into the intake and exposed to elevated temperatures. The time of residence inside the cooling system is an important parameter when analyzing the environmental impact. If the water body also serves as a water supply for the nearby communities severe control of pollutant dispersion should be exercised.

Due to the environmental concern, licensing requirements for installation of new generation power plants become more and more restrictive, pressuring the power generating companies towards the development of more reliable methods for predicting the thermal impact of water discharges. This need motivated advanced studies during the last decade in order to develop such a methods.

The predicting methods are normally classified in three categories: phenomenological, integral and numerical methods. A complete review of the existing methods up to 1975, including an assessment of some of them, is found in [1]. There, it was conclude, based on the poor perspectives for generality of the phenomenological and integral methods, that the viable route for predicting thermal discharges, taking into account all the physical aspects of the plume interaction with the ambient water, would be through the use of distributed models, leading to finite difference and finite element techniques.

The models described in [2] and [4] have followed this route, culminating with the complete models available nowadays [6] [7] [8]. The techniques described in [2] and [4] are three-dimensional formulations with a parabolized coordinate system along the depth according to [9]. This means that the vertical direction a boundary conforming coordinate was used. The shoreline irregularities could not be taken into account precisely since the other two-coordinate remained Cartesian. In [6] [7] this last restriction was removed by using an orthogonal coordinate system fitting the shoreline.

In this paper it is presented a finite volume model for the solution of the depth-averaged equations which models the full-depth thermal discharge in water bodies with arbitrary shoreline. The irregularities of the shoreline is taken into account employing a boundary-fitted nonorthogonal coordinate system. The model is used for simulating real discharge problems and the results are compared with numerical and field data.

PROBLEM FORMULATION

Fig. 1 depicts the problem with the related nomenclature. The jet is assumed to have a full-depth discharge.

---

*Member of ABCM
discharge and buoyance effects are not considered. The surface and bottom interactions can be included by supplying the stresses at $z=0$ and $z=d$ in the $x-y$ plane. Lateral entrainment and jet bending due to lateral currents are properly considered in the elliptic formulation in the $x-y$ plane. An algebraic constant turbulent viscosity is used as recommended in [10] [8].

To deal with the irregular shape of the shoreline a boundary conforming coordinate system is adopted as shown in Fig. 2.

![Fig. 2 Nonorthogonal boundary-fitted mesh](image)

With the assumptions described above the conservation equations for $u, v, p$ and $T$ are written in the $(x, y, z)$ Cartesian coordinate system. The equations are integrated in the $z$-direction and transformed to the new $\xi-\eta$ coordinate system. For a general scalar the conservation equations take the form

\[ \frac{1}{\partial \xi} \left( \frac{\partial (\rho \phi)}{\partial x} + \frac{\partial (\rho \nu \phi)}{\partial y} + \frac{\partial (\rho \phi)}{\partial z} \right) + f^\phi = \frac{\partial (\rho u \phi)}{\partial x} + \frac{\partial (\rho v \phi)}{\partial y} + \frac{\partial (\rho \phi)}{\partial z} + p^\phi \]

\[ \frac{3}{\partial \eta} \left( \frac{\partial (\rho \nu \phi)}{\partial x} + \frac{\partial (\rho \nu \phi)}{\partial y} + \frac{\partial (\rho \phi)}{\partial z} \right) + \frac{2}{\partial \eta} \left( \frac{\partial (\rho u \phi)}{\partial x} + \frac{\partial (\rho u \phi)}{\partial y} + \frac{\partial (\rho \phi)}{\partial z} \right) + \frac{2}{\partial \eta} \left( \frac{\partial (\rho v \phi)}{\partial x} + \frac{\partial (\rho v \phi)}{\partial y} + \frac{\partial (\rho \phi)}{\partial z} \right) + \frac{2}{\partial \eta} \left( \frac{\partial (\rho \phi)}{\partial z} \right) = 0 \]  

(1)

In the above equation $U$ and $V$ are the contravariant velocity components [11], $d$ is the variable depth of the receiving water body and the $C_i$'s coefficients involve the diffusion coefficients of momentum and energy. $P^\phi$ and $S^\phi$ are given in Table 1, where $S^\phi$ and $S^\phi$ are the diffusion fluxes of momentum and energy in the bottom and surface. Details of the mathematical model and the corresponding transformation to the $\xi-\eta$ plane can be found in [12].

Table 1 Pressure and Source Terms for $\phi$

<table>
<thead>
<tr>
<th>$\eta$</th>
<th>$\xi$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$p^\phi$</td>
<td>$0$</td>
</tr>
<tr>
<td>$u$</td>
<td>$S^\phi + S^\phi$</td>
</tr>
<tr>
<td>$v$</td>
<td>$S^\phi + S^\phi$</td>
</tr>
<tr>
<td>$T$</td>
<td>$S^\phi + S^\phi$</td>
</tr>
</tbody>
</table>

BRIEF DESCRIPTION OF THE NUMERICAL METHOD

The numerical method to solve the set of partial differential equations is described in details in [11] and [13]. For the sake of compactness, only the major features will be repeated here. The approximate equations is obtained by integrating Eq.(1) over the elemental volume, which is equivalent of performing momentum, mass and energy balances at discrete level. For the boundary elements the fluxes required by the integration are given according to the existing boundary condition at that boundary. For the internal elements the value of $\phi$ and its derivative at the control volume interfaces are calculated using the Upstream-weighted Differencing Scheme [14].

The pressure-velocity coupling is handled using the PRIME method, as described in [15]. In this method a Poisson-like equation is solved for pressure and this pressure field is used to correct the velocities to satisfy mass, and as the new field for the next iteration. The velocity estimates are calculated in a point-by-point fashion, sweeping the whole domain only once for each coefficients update. As a result only a linear system for pressure is solved in each iteration cycle using the SOR method.

NUMERICAL PREDICTIONS

To assess the numerical model several test problems were realized, trying to verify important terms of the equation set. For the sake of brevity these results are not shown here and can be found in [12]. In this work only the numerical results obtained in simulating real discharge problems are presented. Two problems are solved. The first one deals with the thermal discharge at Pickering NGS A, located at Lake Ontario, Canada, for which there is a numerical simulation for comparison [16], and the second one compares the numerical results obtained in this work with the field data measured at Point Beach Power Plant, located at Lake Michigan, USA [1].

Results for Pickering NGS A. Before giving the details of this case it is worthy to comment some features of the method used in [16] to simulate this problem. The numerical model adopted is described in [16] [7] [8]. It solves the three-dimensional equations of motion and energy. The turbulent viscosity can be either determined through an algebraic model or through the $k-c$ formulation. The three-dimensional grid is obtained firstly generating a two-dimensional boundary-fitted orthogonal coordinate for the surface. The three-dimensional elemental volumes are obtained by dividing the local depth, for each horizontal grid corner, in the desired number of elements for the vertical direction [8]. This means that the top view of the domain discretization for any depth level is the same.

For the simulation described in [16] a 30X16X5 discretization is adopted. The grid is not shown here due to space restriction. Boundary and discharge conditions are depicted in Fig. 3, where it is also shown the shoreline and the grid adopted in this study. The turbulent diffusivity was taken equal to 0.1 m²/s, as suggested in [10] [8] and used in several other works in this field. The turbulent Froude
number was taken equal to 0.7. Nor surface neither
bottom stresses are accounted for in both tests.

Fig. 4 shows the isotherms for the level 2 and 3
(level 1 is the surface) obtained in [16] using the
3-D model. It is important to say that there is slight
difference in the isotherm shapes for level 1 and 2
and for level 3, 4 and 5. The strong thermal gradient
occurs between levels 2 and 3, which is the region in
the vertical direction occupied by the thermal plume.
Fig. 5 brings to the reader the isotherms obtained in
this work and Fig. 6 shows the 17°C isotherm obtained
in this work and obtained in [16].

Fig. 4 Isotherms - Pickering NGS A - From [16]

Comparing the results one sees that the
temperature field calculated with the depth-averaged
equations falls in between the temperature fields for
level 2 and 3 of [16]. This was expected since in
this formulation the velocity field, as well as the
temperature field, represents the depth-averaged
results. In the three-dimensional model the buoyancy
effects lifts the plume towards the surface causing a
stronger penetration of the plume into the lake.

Fig. 5 Isotherms - Pickering NGS A - present work

Far from the discharge the 17°C isotherm no
longer lies between the isotherms of [16]. This
behavior is, probably, due to a recirculation that was
captured by the present model close to point A, which
is absent in the 3-D results.

As an overall outcome of this first test one
could say that the present results follow the expected
trends and agree qualitatively with the results of a
general 3-D model. This encourages the use of the
model developed here for predicting full-depth thermal
discharges in shallow waters, where well-mixed
conditions prevail.

Results for Point Beach Power Plant. The main
goal of this test is to compare the numerical results
with the experimental ones. The Point Beach Power
Plant results are extensively used by researchers to
compare their mathematical models. The field data was
obtained during 1971 and 1972 with a boat equipped
with thermistors positioned at every 0.5 m vertically.

Fig. 6 Isotherms comparison

The computational domain and the grid adopted in
this study is shown in Fig. 7, where the corresponding
parameters are also given. The simulation carried out
refers to the "Application no. 2", as described in
[11], pg. 505, where the bathymetry for the site is
also presented. A moderate ambient current is
prescribed in the west face of the computational
domain. The outfall protudes into the lake at an
angle of 60° to the shore, as is sketched in Fig. 7.

0 50 100
SCALE - m

U_current=0.09 m/s
T_1=17.5°C
Q_discharge=247 m³/s
T_ambient=8.3°C

Fig. 7 Grid used to simulate the Point Beach case

It is important to point out that no grid
resolution study was carried out, since these are
preliminary results for this test. It is also possible
to draw from the numerical results that the north face
of the computational domain is not far enough, since
the influence of the discharge is being "felt" there.
As a result, the boundary condition employed for this
boundary is not the appropriate one. Since one is
using a slip condition, allowing no flow through the
north boundary, the jet is not allowed to penetrate,
as it would, if mass were permitted to cross this
boundary.

Fig. 8 shows the isotherms obtained in this work
and the ones from the field measurements. The shape
of the isotherms agree well, with the discharge not
influencing the ambient water as far as the real
discharge does. The reason for this behavior was
already explained in the presentation of the first
test.

Fig. 9 presents the dimensionless temperature
gainst the distance along the line seen in the insert of
Fig. 5. The maximum error in predicting the plume
temperature at the point that the line crosses the
isotherms is about 15%. It is expected that refining
the test, like moving the north boundary far from the
discharge will improve the results.

The results obtained with this simulation again
indicate that the model follow closely the physics of
the phenomenon, being possible to predict the full-
depth discharges in shallow waters.
CONCLUDING REMARKS

This work presents a numerical method for the solution of the depth-averaged Navier-Stokes and energy equations in boundary-fitted coordinates for thermal discharge problems. The model described here is the first goal in a research project which is being conduct to develop numerical models to simulate complex environmental flows.

The results predicted here encourages the use of this methodology in the route for the development of more complete models.

ACKNOWLEDGMENTS

This work was performed under a contract with Centrais Elétricas do Sul do Brasil S/A - ELETROSAUL, through the Assessoria de Fontes Alternativas de Energia (AFAE) of the Diretoria de Engenharia e Contrações. The financial support is acknowledged.

REFERENCES


