COBEM 87 — IX Congresso Brasileiro de Engenharia Mecânica, Florianópolis, SC (Dezembro, 1987)
9th Brazilian Congress of Mechanical Engineering

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 Mecânica
Universidad de Magalhanes - 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 behaviour of thermal discharges in shallow waters.

INTRODUCTION

The waste water from the cooling systems of eletric power generation plants is normally discharged into lakes and rivers. In the case of 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 species is strongly dependent of the temperature and the equilibrium betwen the species may shift to a situation where an undesiderable specie may become predominant. The increase in the water temperature, associated with the plume velocity may block the natural travelling path of certain species difficulting 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 exercized.

Due to the environmental concern, licensing requirements for instalation of new generation power plants become more and more restrictive, pressing 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 assesment 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| |3| |4| |5| have followed this route, culminating with the complete models available nowadys |6| |7| |8|. The techniques described in |2| |3| and |4| are three-dimensional formulations with a parabolized coordinate following the main flow. The variable bottom is not take into account precisely, since the models deal with uniform depths. In |5| this limitation is removed using a coordinate system along the depth according to |9|. This means that is 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 a 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

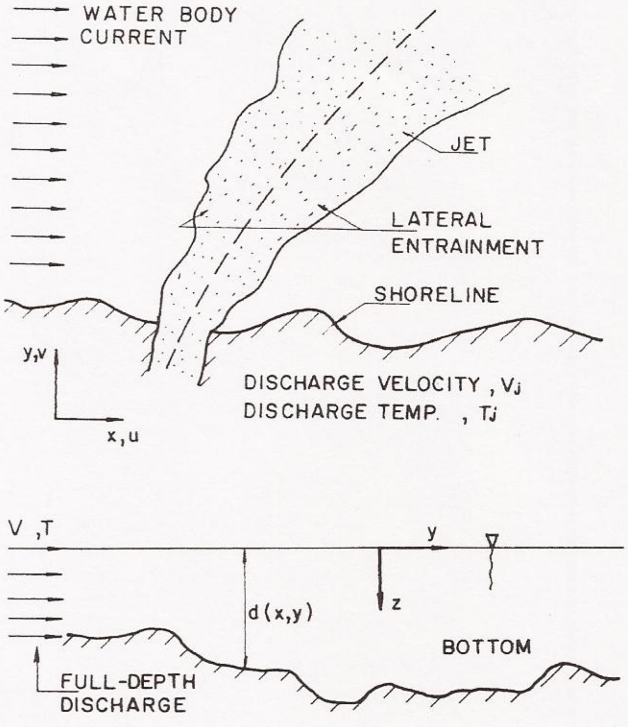


Fig. 1 Top view of the discharge jet (a) and variable depth of the computational domain (b)

---

*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.

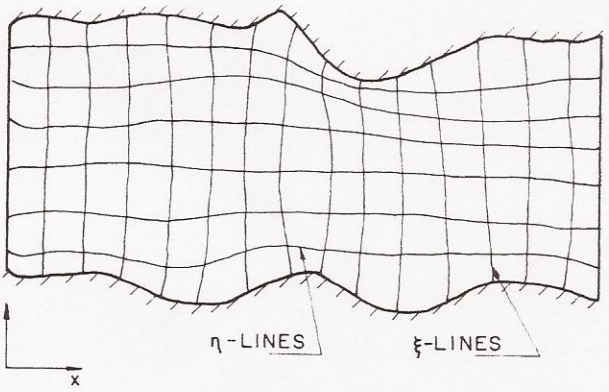


Fig. 2 Nonorthogonal boundary-fitted mesh

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-diretion and transformed to the new ξ-η coordinate system. For a general scalar the conservation equations take the form

$$\frac{1}{J}\frac{\partial}{\partial t}(\rho\phi d) + \frac{\partial}{\partial \xi}(\rho U\phi d) + \frac{\partial}{\partial \eta}(\rho V\phi d) + \hat{P}^{\phi} =$$

$$\frac{\partial}{\partial \xi}\left(C_1\frac{\partial\phi}{\partial\xi} + C_2\frac{\partial\phi}{\partial\eta}\right) + \frac{\partial}{\partial \eta}\left(C_4\frac{\partial\phi}{\partial\eta} + C_5\frac{\partial\phi}{\partial\xi}\right) + \hat{S}^{\phi} \qquad (1)$$

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's coefficients involve the diffusion coefficients of momentum and energy. $\hat{P}^{\phi}$ and $\hat{S}^{\phi}$ are given in Table 1, where S_B^{ϕ} and S_S^{ϕ} are the diffusion fluxes of momentum and energy in the bottom and surface. Details of the mathematical model and the corresponding transformation to the ξ-η plane can be found in |12|.

Table 1 Pressure and Source Terms for ϕ

ϕ	1	u	v	T
$\hat{S}^{\phi}$	0	$\frac{S_B^u + S_S^u}{J}$	$\frac{S_B^v + S_S^v}{J}$	$\frac{S_B^T + S_S^T}{c_p\, J}$
$\hat{P}^{\phi}$	0	$(P_\xi y_\eta - P_\eta y_\xi)d$	$(P_\eta x_\xi - P_\xi x_\eta)d$	0

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 ϕ 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 S.O.R method.

NUMERICAL PREDICTIONS

To asses 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 firt 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 |6| |7| |8|. It solves the three-dimensional equations of motion and energy. The turbulent viscosity can be either determined through a algebraic model or through the k-ε 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 volumes for the vertical direction |9|. 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 dicretization 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 difusivity was taken equal to 0.1 m²/s, as suggested in |10| |8| and used in several other works in this field. The turbulent Prandtl

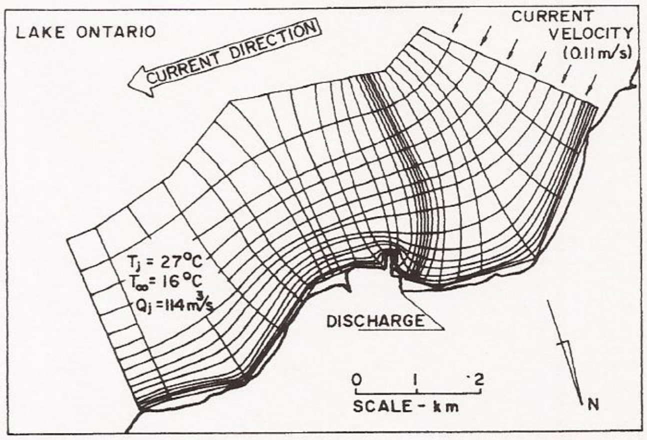


Fig. 3 Grid used to simulate the Pickering NGS A case

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 little difference in the isotherm shapes for level 1 and 2 and for level 3, 4 and 5. The strong thermal gradient occurs betweem 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|.

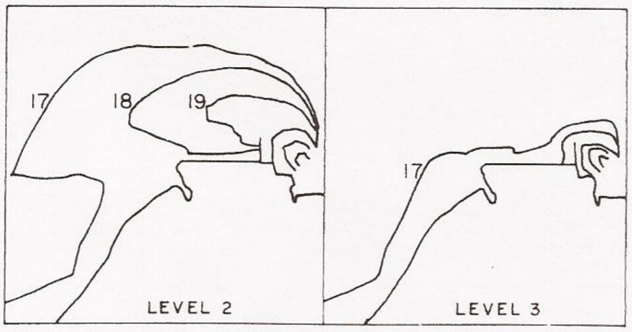


Fig. 4 Isotherms - Pickering NGS A - From |16|

Comparing the results one sees that the temperature field calculate 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 buoyance effects lifts the plume towards the surface causing a stronger penetration of the plume into the lake.

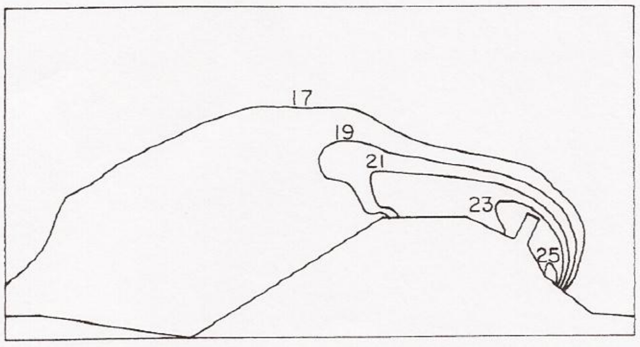


Fig. 5 Isotherms - Pickering NGS A - present work

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 obtanded during 1971 and 1972 with a boat equipped with thermistors positioned at every 0.5 m vertically.

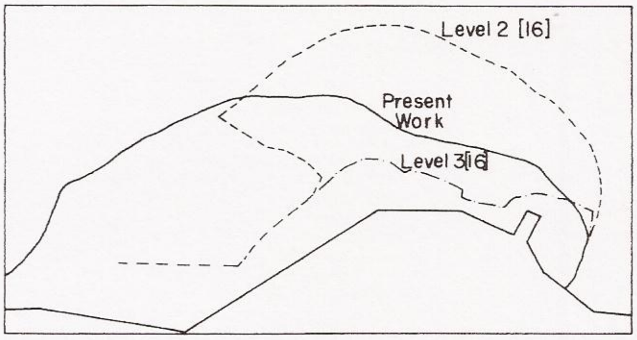


Fig. 6 Isotherms comparison

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 |1|, 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.

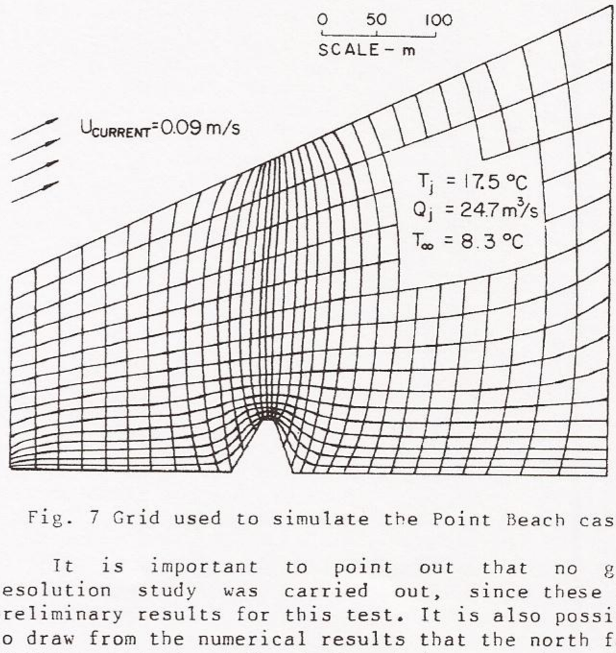


Fig. 7 Grid used to simulate the Point Beach case

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 permited 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 against the distance along the line seen in the insert of Fig. 9. 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.

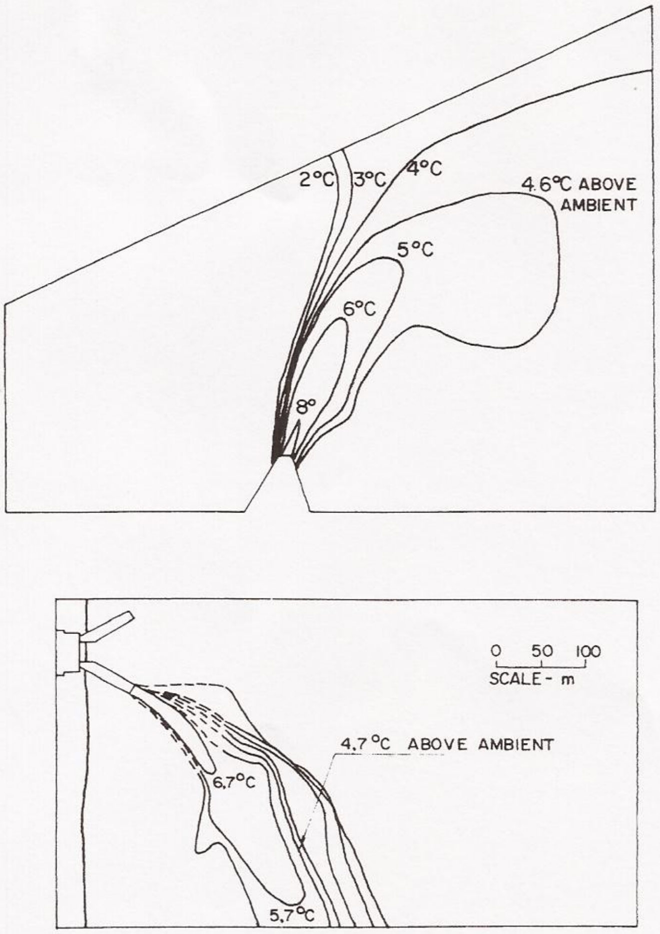
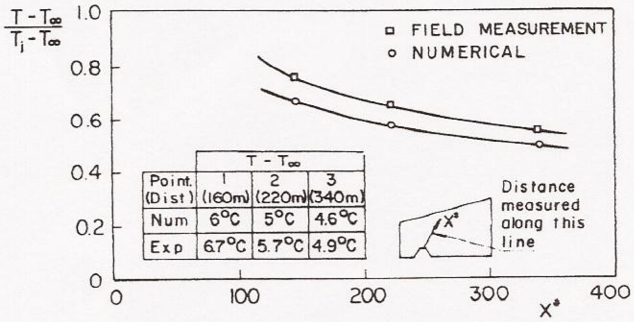


Fig. 8 Isotherms obtained in the present work (a) and from field measurements (b)



	$T - T_\infty$		
Point. (Dist)	1 (160m)	2 (220m)	3 (340m)
Num	6°C	5°C	4.6°C
Exp	6.7°C	5.7°C	4.9°C

Fig. 9 Temperature along the jet "centerline"

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.

ACKNOWLEDGES

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

REFERENCES

|1| Dunn, W.E., Policastro, A.J. and Paddock, R.A., Surface Thermal Plumes: Evaluation of Mathematical Models for the Near and Complete Field, Argonne National Laboratory Report No. ANL/WR-75-3,1975.

|2| Raithby, G.D., Prediction of Dispersion by Surface Discharge, Report of the Basin Investigation and Modelling Section, Canada Centre for Inland Waters, Burlington, Ontario, 1976.

|3| McGuirk, J. and Rodi, W., Mathematical Modelling of Three-Dimensional Heated Surface Jets, J. Fluid Mechanics, Vol. 95, Part 4, pp. 609-633, 1979.

|4| Raithby, G.D. and Schneider, G.E., The Prediction of Surface Discharge Jets by Three-Dimensional Finite Difference Model, J. Heat Transfer, Vol. 102, No. 1, pp. 138-145,1980

|5| Elliot, R.V. and Raithby, G.D., A Three-Dimensional Finite Difference Model for the Prediction of Thermal Discharges, Third Conference on Waste Heat Management and Utilization, Miami Beach, Florida, May 1981.

|6| Raithby, G.D. and Elliot, R.V., The Prediction of a Thermal Discharge Entering a Cross Flow, The International Symposium on Refined Modelling of Flows, Paris, Sept. 7-10, 1982.

|7| Raithby, G.D., Huget, R. and Van Doormaal, J.P., A Three Dimensional Model for Prediction of Hydraulic Flows in Complex Geometries, Thermal Science Ltd. report to Ontario Hydro, May 1982.

|8| Raithby, G.D., Predictions of Thermal Plumes at the Bruce Nuclear Power Development, Thermal Science Ltd., report to Ontario Hydro, May 1984.

|9| Phillips, N.A., A Coordinate System Having Some Special Advantages for Numerical Forecasting, J. of Meteorology, Vol. 14, pp. 184-185, 1957.

|10| Csanady, G.T., Turbulent Diffusion in the Environmental, D. Reidel Publishing Company, Dordnecht, Holland, pp.106-107, 1973.

|11| Maliska, C.R. and Raithby G.D., A Method for Computing Three-Dimensional Flows using Nonorthogonal Boundary-Fitted Coordinates, Int. J.Num. Meth. in Fluids, Vol. 4, pp.519-537, 1984.

|12| Perez, J.O., Simulação Numérica de Descargas Térmicas em Corpos d´Agua Rasos de Geometria e Profundidade Variáveis, Diss. de Mestrado, Dep. Eng. Mec. Univ. Fed. Santa Catarina, junho 1987.

|13| Maliska, C.R. and Raithby G.D., Calculating Three-Dimensional Flows using Nonorthogonal Grids, Numerical Methods in Laminar and Turbulent Flow, C. Taylor, J.A. Johnson and W.R. Smith, pp. 656-666, Pineridge Press, 1983.

|14| Raithby, G.D. and Torrance K.E., Upstream-weighted Differencing Schemes and Their Applications to Elliptic Problems Involving Fluid Flows, Computer and Fluids, Vol.2, pp.191-206,1974.

|15| Maliska, C.R., A Solution Method for Three-Dimensional Parabolic Fluid Flow Problems in Nonorthogonal Coordinates, Ph.D. Thesis, University of Waterloo, Canada, 1981.

|16| Elliott, R.V., Use of the Raithby Mathematical Model to Study the Effects of Tempering on Thermal Plume Behavior, Report no. 86186, Ontario Hydro, July 1986.