International Journal of Current Engineering and Technology

E-ISSN 2277 – 4106, P-ISSN 2347 – 5161
Available at http://inpressco.com/category/ijcet

*Research Article*

# Thermal Dispersion Model for Cooling Water of Thermal Power Plant System

**Shaikh Mohamed Ashfaq**[†*]

[†]Department of Chemical Engineering, Z.H.C.E.T [AMU], Aligarh, India

Accepted 10 July 2015, Available online 16 July 2015, **Vol.5, No.4 (Aug 2015)**

***Abstract***

*The existing of thermal power stations on rivers results in numerous environmental dilemma. Thermal contamination has come to mean the unfavorable effects of abnormal temperature changes in a natural body of water, caused by the release of industrial cooling water. The discharging of the cooling system of the thermal power stations into rivers may be surface discharge, submerged point discharge and submerged multi-port diffusers. In this research paper, the study is focused on the most common surface discharge method and isotherm map, centre line of temperature decay, and isotherm areas will be studied. Three dimensional (3D) model of the Cooling circuit thermal plume model by considering far field assessment and near field assessment for the thermal power plant cooling water system will be analyzed. In order to quantify the discharge-intake recirculation effect (the temperature increase at the intake point due to the effect of the discharge). The study will cover the stretch of the river indicated for the far field. It is understood that the level of accuracy of the model will be much more precise for the near field than for the far field by using computational fluid dynamics (CFD) 3D model. In case of any discrepancy in the cooling performance of the cooling tower, CFD analysis is used to propose modification to improve the cooling performance.*

***Keywords:*** *CFD; 3D; Plume; Centre line of temperature decay, Thermal pollution.*

## 1. Introduction

The cooling water of the thermal power plant is considered one of the most pollution sources of rivers, especially for the liquid natural gas stations, which require large amount of refrigeration water. Thermal pollution affects both the physical and chemical characteristics of the flowing water. Also, thermal pollution effect on morphology of the rivers due to the growth of the water plants depending on the water temperature increase and sedimentation process. the chemical reactions is approximately doubled for each 10°C temperature rise, which increase the sedimentation due to changes of water properties, changes in flocculation, and ion exchange. The discharging of the cooling system of the thermal power stations into rivers may be surface discharge, submerged point discharge and submerged multi-port diffusers. In this research paper, the study was focused on the most common surface discharge method.

Submerged discharges and multiport diffusers are finding wider use as means of disposing of waste heat from once-through condenser cooling water systems for power plants. These discharges appear to be superseding surface shoreline discharges for large power plant applications. A reason for employing submerged discharges is that they may create greater initial dilution of the effluent than surface discharges.

Argonne National Laboratory (ANL) has been studying the effects of power-plant waste-heat discharges into the Great Lakes since 1970 Frigo, A.A.(1972). The initial effort involved the study of thermal plumes from power plants employing shoreline surface discharges, ANL began an extensive field program to measure the physical phenomena related to surface shoreline waste-heat discharges into the Great Lakes. Approximately 75 sets of thermal-plume data along with related physical parameters suchas lake currents, ambient diffusivities, and meteorological.

In the present study development of the Cooling circuit thermal plume model by considering far field assessment and near field assessment for the thermal power plant cooling water system will be analyzed using computational fluid dynamics (CFD). In order to quantify the discharge-intake recirculation effect (the temperature increase at the intake point due to the effect of the discharge). The study will cover the stretch of the river indicated for the far field.

*Corresponding author: **Shaikh Mohamed Ashfaq**

## 2. CFD Modeling of the Thermal dispersion

*2.1 Geometry*

The surface discharge cooling system is considered the simplest way for discharging the cooling water into rivers, but the prediction of the plume through river is very difficult. Since the relative cost of the surface discharge of the cooling system of the thermal power plant is very low compared with the other two types, the most cooling systems of the thermal power plants are surface discharge type. In the present study three dimensional model of the cooling water system is considered. The modelled simulation domain was 70 m long and 50 m wide and 8m deep. Flow rate used for at hot fluid inlet was 10 $m^3/s$ and at other inlet 5 $m^3/s$. The software code flow 3D used for geometry /mesh generation. **Fig1.** Shows the overall view of the three dimensional (3D) model generated for cooling water.

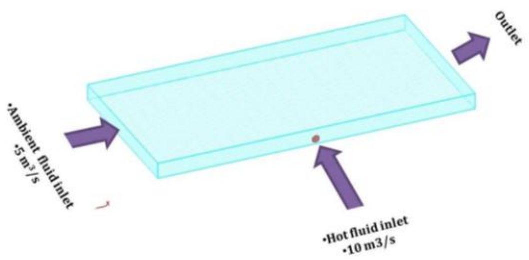


**Fig. 1** Three dimensional view of the model generated for cooling circuit thermal plume

*2.2 Modeling*

General Details of the CFD model are as follows

2.2.1 Modeling Approach

Flow from the hot fluid into the river is categorized as surface discharge flow. Thermal dispersion are developed by using energy balance equation, density elevation, fluid source, gravity, heat transfer and viscosity turbulence model have been used in flow 3D. The software code 'Flow 3D' was used for this analysis. The standard k - ε model is used in this simulation. It is a semi-empirical model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε). In the derivation of the k - ε model, it is assumed that the flow is fully turbulent, and the effects of molecular viscosity are negligible. The standard k - ε model is therefore valid only for fully turbulent flows. Thermal model to account the temperature effect in the form of boussinesq equation for the heat transfer by natural convection was setup along with density and turbulence models to obtain the results. For the Cooling circuit thermal plume geometry, a grid with approximately 219520 cells was constructed with cell size ranging from 0.5 m to 0.01 m in order to get adequate resolution of the geometric features. No-slip conditions were prescribed on all solid surfaces, and all the walls in cooling circuit thermal plume geometry.

2.2.2 Assumptions

The following assumptions were made while studying the performance of the thermal dispersion model: Latent of heat of the vaporization was ignored initial in order the simplified the model. To get the initial picture of the model and it will be incorporate to give the overall problem. The physical properties such as the specific heat, thermal conductivity and viscosity are assumed to be constant with temperature. The ambient fluid was assume to be river and specified by inlet velocity.

Wind speed and RH humidity are ignored. Assuming the contribution by this term is small. Surface discharge of the hot water fluid. The inflow is continuous and steady; Inflow of 5 $m^3/s$ from one side (ambient stream) and the flow 10 $m^3/s$ (hot stream) from other end. Considering a ΔT of 5 $^0$C and surrounding fluid temperature was taken 20 $^0$C. The free surface level of the water has no major variation.

## 3. Results and Discussion

The following subsections discuss the results of the CFD simulation for present thermal load and respective figures are included to illustrate the thermal plume profile in the simulation domain. The contours of the temperature are coloured by the temperature magnitude and the temperature scale ($^0$C) is shown on the top side on each figure. The colour bar located to the top side of each temperature contour shows the temperature magnitude ranging from 20.0 $^0$C as blue to 22$^0$C as red.

*3.1 Temperature Contours*

Thermal plume stem formation begins at the time 10 second. After that plume appears to be shifted in the upper direction i.e. in the flow direction at time 19 second.

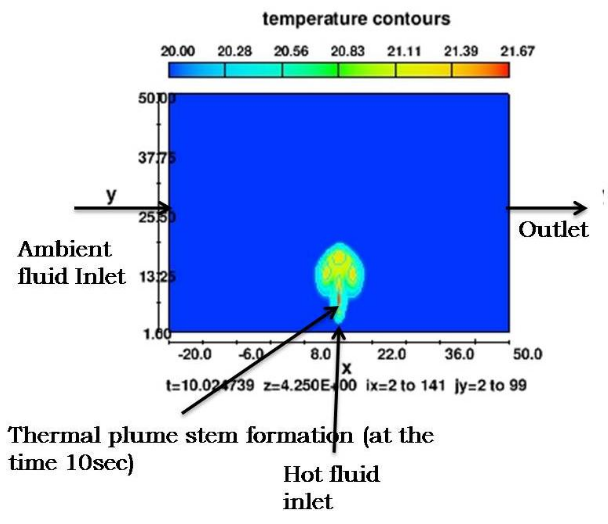


**Fig. 2** Temperature contour after 10 Second

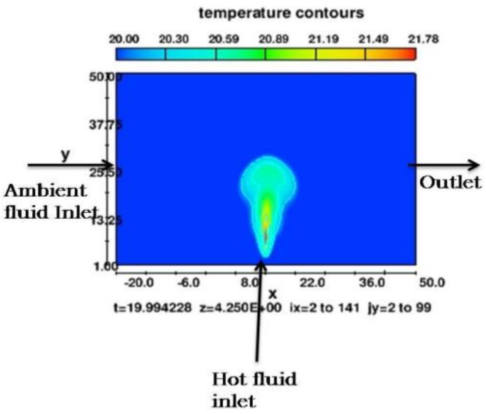


**Fig.3** Temperature contour after 20 second

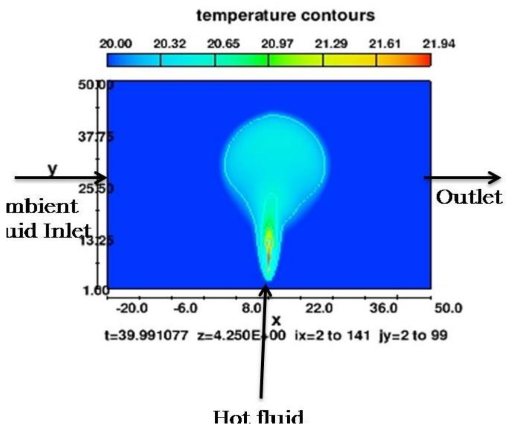


**Fig. 4** Temperature contour after 40 second

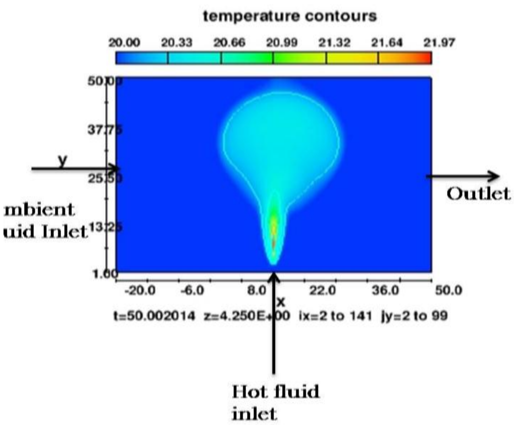


**Fig. 5** Temperature contour after 50 second

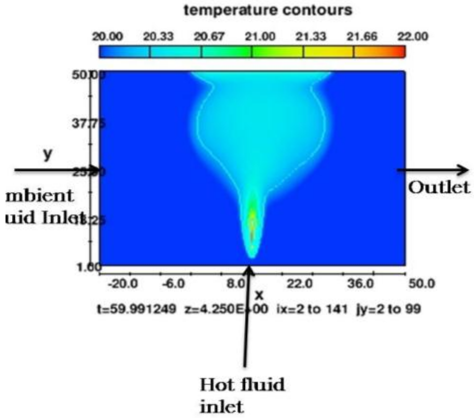


**Fig. 6** Temperature contour after 60 second

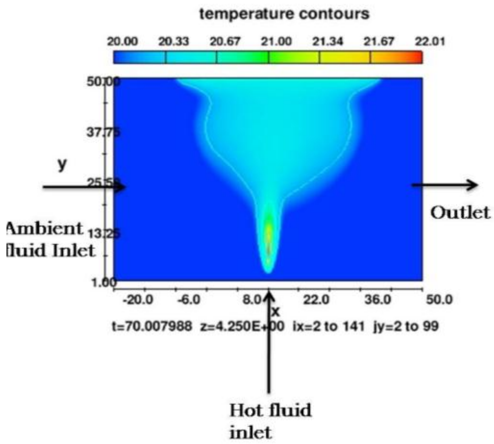


**Fig. 7** Temperature contour after 70 second

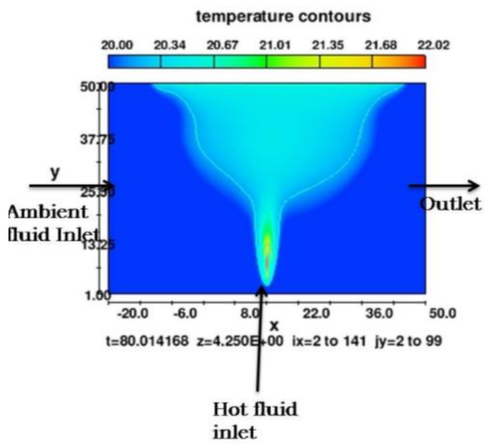


**Fig. 8** Temperature contour after 80 second

Thermal plume has classical "mushroom "spherical head tail structure in the flow direction at time 30 second. The head cools through the time by thermal diffusion we can see that the hotter isotherm in the head is disappearing one after another in the flow direction at time 59 second. Thermal diffusion causes the temperature to decrease with height and stem radius to increase in the flow direction at time 80 second.

*3.2 Validation model*

The Zion station is located in northeastern Illinois on the western shore of Lake Michigan about 5.1 km south of the Illinois-Wisconsin state line. It consists of two units (Unit 1 to the south. Unit 2 to the north) employing pressurized water reactors. Each unit is capable of a gross generating capacity of 1100 MWe. Each unit has its own discharge structure located 232 m from shore, about 47 m on either side of the plant centerline, in about 4.5 m of water. The measurement and data acquisition system for relatively rapid measurement of the three dimensional temperature structures of the thermal plumes from the submerged discharge developed by ANL.

Cooling water system and discharge structures of Zion nuclear power station has the following details and temperature contour Frigo, A.A.(1972). The discharge flow rate =50 $m^3/s$. Temperature of the discharge water =28.6 $^0C$. Ambient water temperature= 19.6 $^0C$. River stream flow rate =2.2 cm/s. Fig 9 show isotherm plot at the surface using data acquisition technique.

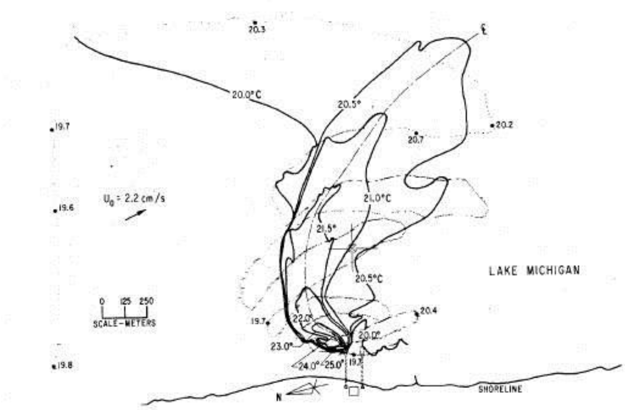


**Fig. 9** Temperature contour Zion power plant

Fig 10 to Fig 13 show the temperature contour for the Cooling water system and discharge structures of Zion nuclear power station obtained by using CFD. The discharge flow rate =50 $m^3/s$. Temperature of the discharge water =28.6 $^0C$. Ambient water temperature= 19.6 $^0C$. River stream flow rate =2.2 cm/s.

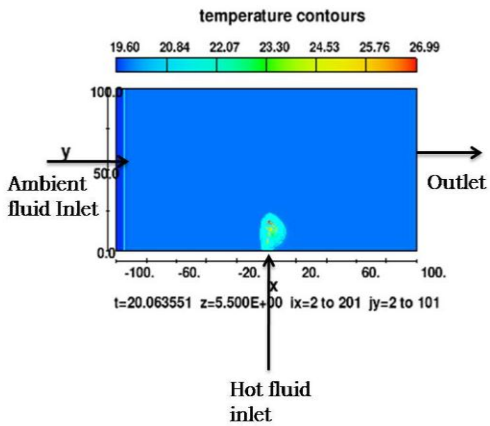


**Fig. 10** Temperature contour after 20 second

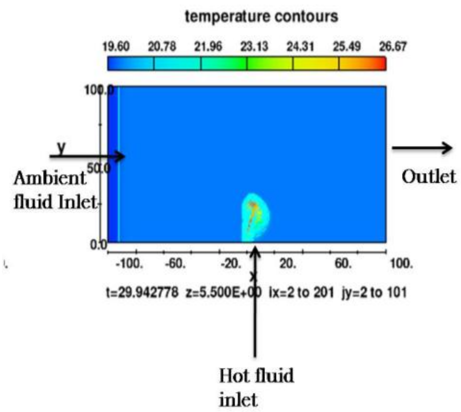


**Fig. 11** Temperature contour after 30 second

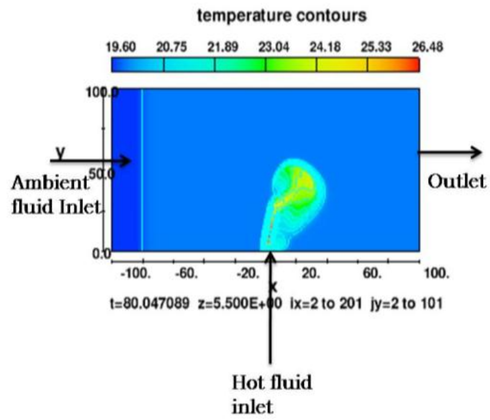


**Fig. 12** Temperature contour after 80 second

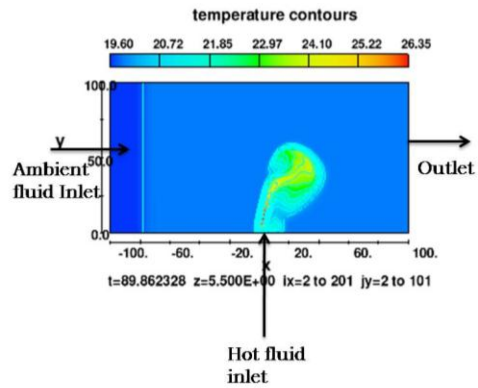


**Fig. 13** Temperature contour after 90 second

### *3.3 Centre line of Temperature decay*

Miller and Brighouse developed the following equation for the centre line of Temperature decay from field and laboratory analysis as follows:

$$\frac{\Delta T}{\Delta To} = 3.774\left(\left[\frac{S}{(ho * bo)^{0.5}}\right]\right)^{-0.405} F^{-0.373} A^{-0.133} \alpha^{-0.084}$$

in which

| | |
|---|---|
| ΔT/ΔTo | excess temperature ratio; |
| S | distance from the outlet on the centre line of the plume; |
| F0 | densimetric Froude number (the ratio of inertia to buoyancy forces acting on the discharge); |
| α | angle between the centre line of the outlet and the river measured in the downstream direction of the flow (in radian); |
| ΔT | difference between local temperature at any point and normal ambient temperature; |
| ΔTo | difference between discharged water temperature at the outlet and normal ambient temperature; |
| Ho | outlet flow depth; m |
| To | discharged water temperature; |
| Bo | half width of the outlet; |
| A | aspect ratio =ho/bo |

**Table 1** Parameters obtained from theoretical equation and CFD

| S. No | Parameters | Condition | |
|---|---|---|---|
| | | Model 1 | Zion model |
| 1 | Theoretical Temperature | 20.63$^0$C | 22.94$^0$C |
| 2 | CFD Simulation Temperature | 20.67$^0$C | 22.97$^0$C |

Since due to temperature dependent viscosity, hot material has lower viscosity than the ambient fluid. Thermal diffusion causes the temperature to decrease with height and stem radius to increase. The local temperature at any point is found from the theoretical calculation is 20.63$^0$C and simulation local temperature 20.67$^0$C. From the temperature contour we can see the temperature is dispersing in both directions and jet length is 8.25m. The dispersion of the hot water in the river water has density difference in the range of 1 kg/m3 and droplet size of the dispersed phase is in the range of the less than 1 micron.

The local temperature at any point is found from the theoretical calculation is 22.94$^0$C and simulation local temperature 22.97$^0$C. From the temperature contour we can see the temperature is dispersing in directions of the ambient current and jet length is 0.246m. The dispersion of the hot water in the river water has density difference in the range of 1 kg/m3 and droplet size of the dispersed phase is in the range of the less than 1 micron.

## Conclusion

Following the CFD analysis, the following conclusions can be made.

1. The temperature contour for cooling water system model was estimated by CFD modelling.
2. The theoretical and CFD temperature decay are nearly same for model 1.
3. For Zion nuclear power system contours was obtained by CFD.
4. The dispersion model for cooling water system may be concluded after finalizing the variation in physical properties.

## Nomenclature

| | |
|---|---|
| ΔT/ΔTo | excess temperature ratio; |
| S | distance from the outlet on the centre line of the plume; |
| F0 | densimetric Froude number (the ratio of inertia to buoyancy forces acting on the discharge); |
| α | angle between the centre line of the outlet and the river measured in the downstream direction of the flow (in radian); |
| ΔT | difference between local temperature at any point and normal ambient temperature; $^0$C |
| ΔTo | difference between discharged water temperature at the outlet and normal ambient temperature; $^0$C |
| ho | outlet flow depth; m |
| To | discharged water temperature; $^0$C |
| bo | half width of the outlet; m |
| A | aspect ratio =ho/bo |
| D | Diffusion coefficient, $m^2/s$ |
| U | Velocity, m/s |
| K | Turbulence kinetic energy , $m^2/s^2$ |
| ε | Dissipation rate, $m^2/s^3$ |

## Acknowledgment

The author would like to thanks Almighty Allah for giving the wonderful life. The author is also grateful to his parents and AMU.

## References

Huang, W. and M.L. Spaulding (1995). A three dimensional numerical model of estuarine circulation and water quality induced by surface discharges. *ASCE Journal of Hydraulic Engineering*, 121 :( 4), pp. 300-311.

Tamai, N., Wiegel, R.L(1969)., Horizontal surface discharge of warm water jets, *Proc. of ASCE Journal of power division*, Vol. 95, no po2, pp 253-276.

Frigo, A.A.(1972), Prediction of surface plume areas associated with heated discharges into large lakes – a phenomenological model, Proc. 15th Conf., University of Wisconsin, *Int1. Assoc. for Great Lakes Research*, pp 583-587.

Jirka, G. H., E. E. Adams, and K. D. Stolzenbach (1981), Buoyant surface jets, *J. Hydraulic. Div.*, Proc. ASCE, 107(HY11), 1467– 1487.

M. Waldichuk (1965), Water exchange in Port Moody, British Columbia, and its effect on waste disposal, *Journal of Fisheries Research Board of Canada*, Vol. 22, pp. 801- 822.

G.L. Mellor and T. Yamada (1982), Development of a turbulence closure model for geographical fluid problems, *Review of Geophysics*, Vol. 20, pp. 851-875.

V. Casulli and E. Cattani(1994), Stability, Accuracy and Efficiency of a semi-implicit method for three dimensional shallow water flow, *Computers Math. Application*, Vol. 27, pp. 99-112.

A.J Johnston, C.R. Phillips and R.E. Volker (1994), Modeling horizontal round buoyant jets in shallow water, *Journal of Hydraulic Engineering*, Vol. 120, pp. 41-59.

G.A. Bemporad(1994), Simulation of round buoyant jet in stratified flowing environment, *Journal of Hydraulic Engineering*, Vol. 120, pp. 529-543.

M. Abbaspour, A. H. Javid, P. Moghimi, and K. Kayhan(2005), Modeling of thermal pollution in coastal area and its economical and environmental assessment, *International Journal of Environmental Science and Technology*, Vol. 2, no. 1, pp. 13–26.

S. A. Lowe, F. Schuepfer, and D. J. Dunning(2009), Case study: three-dimensional hydrodynamic model of a power plant thermal discharge, *Journal of Hydraulic Engineering*, Vol. 135, no. 4, pp. 247–256.

D. G. Kim and H. Y. Cho(2006), Modeling the buoyant flow of heated water discharged from surface and submerged side outfalls in shallow and deep water with a cross flow, *Environmental Fluid Mechanics*, Vol. 6, no. 6, pp. 501–518.

G. H. Jirka(1982), Multiport diffusers for heated disposal: a summary, *Journal of the Hydraulics Division*, Vol. 108, no. 12, pp. 1425–1468.

S. H. Chieh (1987), Two-dimensional numerical model of thermal discharges in coastal regions, *Journal of Hydraulic Engineering*, Vol. 113, no. 8, pp. 1032–1040.

J. M. Beckers and J. J. Van Ormelingen(1995), Thermo hydrodynamical modeling of a power plant in the Zeebrugge Harbour, *Journal of Hydraulic Research*, Vol. 33, no. 2, pp. 163–180.

Policastro, A. J.t and Tokar, J. V.(1972), Heated Effluent Dispersion in Large Lakes: State-ofthe- Art of Analytical Modeling: Part 1. Critique of Model Formulations, *Argonne National Laboratory Repor*t ANL/ES 11

Frigo, A. A., and Frye, D. E.(1972), Physical Measurements of Thermal Discharges into Lake Michigan: 1971, *Argonne National Laboratory Report* ANL/ES-16.

Frigo, A. A., Frye, D. E., and Tokar, J. V.(1974), Investigations of Heated Discharges from Nuclear Power Plants on Lake Michigan: 1972, Argonne National Laboratory Report ANL/ES-32.

Kassem Salah El-Alfy (2004), Eighth *International Water Technology Conference, IWTC8 2004, Alexandria*, Egypt, pp433-447

Fong, D. A. (1998), Dynamics of freshwater plumes: Observations and numerical modeling of the wind-forced response and alongshore freshwater transport, Ph.D. thesis, Joint Program in Oceanography., Mass. Inst. Of Technol./Woods Hole Oceanography. Inst., Woods Hole, Mass.