The Egyptian Int. J. of Eng. Sci. and Technology
Vol. 17, No. 1 (Jan. 2014)

EIJEST

THERMAL CONCENTRATION IN LOW FLOW STREAMS*

Yasser M. Shawky, Ahmed M. Nada, Fahmy S. Abd-Elhaleem+

Hydraulics Research Institute, National Water Research Centre, Delta Barrages, Egypt,

ABSTRACT

This paper employs physical modelling technique to test an outfall structure, as a new approach to streams with low flow velocity and poor water mixing. From available data at the Hydraulics Research Institute (HRI), a thermal power plant close to the left bank of El-rayah El-tawfiki was taken as a case study (Banha power plant) to test a series of outfalls with different combinations. A physical model was constructed inside the northern hall of HRI. One surface outfall and a series of multi-port diffuser outfalls with different parameters were tested. The tested outfall parameters comprised number of diffuser rows, nozzle diameter, space between nozzles and their number. The tests were carried out with one or two rows of the diffuser in operation. The results indicate that the surface outfall leads to a thermal plume with temperature rise more than 5˚C above ambient water outside the mixing zone. For the multi-port diffuser outfall there is high capability of diluting the temperature rise to less than 5^{o}C outside the mixing zone and confining the plume in the third of the width of the stream. The diffuser which has two rows in operation has achieved almost the same mixing zone area as in case of diffuser with only one row in operation near the bed level. The results indicate that it is better to use a diffuser with one row only near the bed level to avoid high cross flow and surface turbulence which may obstruct fish boats operating nearby.

KEY WORDS: Thermal; Power Plants; Multi – Port Diffuser Outfalls.

CONCENTRATION THERMIQUE EN FAIBLE DEBIT FLUX

RÉSUMÉ

Cette technique de modélisation physique du papier pour tester une structur émissaire , comme nouveau approche de flux avec la vitesse d'écoulement faible et pauvre mélange d'eau à partir des données disponibles à l'Institut de Recherches Hydrauliques (HRI) , une centrale thermique près de la rive gauche de El - Rayah El - Tawfiki a été prise comme une étude de cas , (Banha centrale) , pour tester une série de déversoirs avec différentes combinaisons . Modèle A physical a été construit à l'intérieur de la salle nord de HRI .Un émissaire de surface et a series de multi-port émissaires de diffusion avec des paramètres différents ont été testés testés.Les paramètres de rejet comprennent : nombre de lignes de diffuseurs , buses de diamètre , l'espace entre les buses et le nombre . Les tests ont été effectués avec un ou deux rangées de diffuseur dans les résultats opération.Le indiquent que l'émissaire de surface conduit à un panache thermique avec la température augmente de plus de 5 ˚ C au-dessus de l'eau ambiante en dehors de la zone de mélange . Pour le multi-port diffuseur outfallthere est élevée capacité de dilution de l' élévation de la température à moins de 5 ° C en dehors de la zone de mélange et de confinement de la plume dans le tiers de la largeur du diffuseur stream. The qui a deux lignes de fonctionnement a atteint presque l' même zone de zone de mélange comme dans le cas de diffuseur avec une seule ligne en exploitation près les résultats lit de level.The indiqué qu'il est préférable d'utiliser adiffuser avec une seule rangée près du niveau du lit pour éviter l'écoulement de croix haute et de la turbulence de surface qui peut obstruer les poissons bateaux opérant à proximité.

MOTS CLÉS : Thermique, Centrales Electriques ; Multi - Port Diffuseur Emissaires.

* Received1/10/2013, accepted: 4/12/2013, Ref. No. 165, (Original paper)

+ Contact author (fahmyhri@gmail.com)

1. INTRODUCTION

On shore or open channel intake/outfall structures can be used in rivers or streams with relatively high flow velocities. The high flow velocity helps in achieving a good mixing of the effluents from the plant outfall with fresh water and reduces the stratification in front of the outfall. As a consequence, the water temperature rise of the stream is minimized and water quality standards can be achieved. Moreover, the recirculation of effluents from the plant outfall to its intake is avoided.

The situation is more complicated in streams with relatively low flow velocities in which poor water mixing of the hot water with fresh water occurs [9]. In this case, water temperature rise of the stream at the plant vicinity may violate the water quality standards in Egypt which may impact the aquatic life in the stream [6]. Hot water recirculation at the plant intake may occur. According to Law No. 48, 1982 of the MWRI and Law No. 9, 1994 of the Ministry of Environment, (MI), the mixing zone area should be minimized to be less than one third of the width of the stream and the maximum absolute water temperature at the plant vicinity should not exceed 35 °C. The water temperature outside the mixing zone should not exceed 5 °C above the ambient. Because of the low stream velocities, the power plant fails to satisfy the environmental requirements and consequently it cannot be achieved or constructed. Another alternative is to test a new technique for the outfall structure to solve these issues and to make the plant feasible [7]. Hopefully, the new technique presented in this research can be used in the power plants outfalls built on streams with relatively low flow velocity and poor water mixing to increase its efficiency [5].

The objective of this research is to develop a technique to be deployed in the outfall structures of the thermal power plants built on streams with low flow velocities and shallow water depth [1] and [2]. This technique is to be tested hydraulically by employing a physical model study at the Hydraulics Research Institute (HRI). Other objectives that can be fulfilled from this research are to promote the applied research at HRI and introduce a methodology to overcome the water quality and recirculation problems of the effluents in stream with low flow velocities and shallow water depths [3].

2. METHODOLOGY AND MEASUREMENTS

A well designed hydraulic model may be the best tool to check the flow patterns and behavior at the vicinity of the thermal power plants. Also testing the optimal design and orientation of the plant outfall structure built on streams with low velocities and shallow water depths is targeted and needed. The hydraulic model gives direct insight into the physical processes which cannot be obtained otherwise. The similarity laws should be applied to check the model similarity with the prototype. Model should be calibrated against prototype data measurements. Different plant outfalls can be simulated and tested in the physical scale model as follows:

1- **Outfall (1):** On shore outfall structure (surface type).
2- **Outfall (2):** Multi-port diffuser with one row of nozzles.
3- **Outfall (3):** Multi-port diffuser with two rows of nozzles.
4- **Outfall (4):** Two different nozzles diameters should be tested.
5- **Outfall (5):** Two different nozzles space along the diffuser should be tested.

In all tested outfalls in the model, the intake / outfall discharges of the plant should be the same. The water temperature along the model reach at the intake/outfall vicinity can be measured by thermometers connected to a data logger. The water temperature measurements stored in the data logger can be retrieved through the computer after each test for data processing and analysis.

2.1 Undertaken Activities

To achieve the aims of this research, the following activities were carried out:

- Selection of a reach along the river which is characterized by its low flow velocities and discharges. A reach of El-Rayah El-Tawfiki near Banha City and upstream Gamgara Regulator was selected. The selection considered the availability of the bathymetric and hydrometric measurements at HRI. The reason behind this selection is that during the winter season, El-Rayah El-Tawfiki has a very low flow velocities and discharges which make it exposed to high thermal pollution from the thermal power plants [11].

- Identification of thermal power plant characteristics to be simulated in the selected reach with low flow velocity at Banha Power Plant site. The plant characteristics include the plant intake/outfall discharge and the design excess water temperature at the outfall above the ambient water. In 2012, a physical model study was carried out at HRI for this plant. Interested reading on the construction of the physical model used by this study can consult reference [8]. The discharge needed for the plant cooling system which was tested in the study was 13.7 m3/s, (two units in operation). An outfall of multi-diffuser of 24 nozzles arranged in two rows with nozzle diameter of 700 mm and nozzles space of 2 m was tested and the results indicated its capability of diluting the thermal concentration in El-Rayah El-Tawfiki downstream of the plant. In this current research, a discharge of 9.13 m3/s (one unit in operation) was taken instead of 13.7 m3/s to give a wide range for testing more small nozzles diameter and more large nozzles space than in the above - mentioned study. This means that the optimal outfall configurations deduced from this research is not applicable for the plant and it serves only the research objectives.
- An undistorted physical scale model with a scale of 1:50 was constructed at HRI to simulate the flow conditions of El-Rayah El-Tawfiki and the Banha Power Plant. The model simulates 2 Km length of El-Rayah El-Tawfiki and the power plant cooling system. Figure (1) shows a general layout for the model details. Different outfall structures of the power plant including the normal outfall structure (surface outfall) and different design of outfalls consists of multi-port diffusers were simulated to recommend the optimum design of the outfall structure based on the model results.

2.2 Measurements of Thermal Concentration

A system of 20 thermometers was installed on a moving wooden bridge along the model length in order to measure the water temperature at each measuring cross section. The thermometers are connected to a data logger to save the temperature measurements. The measurements extend from the outfall structure to the end of the model. Thirty three measuring cross sections were selected to cover the modeled reach. These cross sections were divided into 3 groups, group I consists of 9 cross sections, group II consists of 10 cross sections and group III consists of 14 cross sections. The varying distance between the successive cross sections in each group depends on the resolution needed at some specific location along the model area. As an example, the highest resolution was made at the vicinity of the outfall structure. The distance between each two successive cross sections for Group I, II, and III is 25 m, 12.5 m and 5 m respectively.

2.3 Model Scenarios

A series of model scenarios for the plant outfalls configurations that consists of surface and multi-port diffuser outfalls with different configurations were tested to determine its capability of diluting the thermal concentration downstream of the plant outfall. In all scenarios, the critical flow conditions in El-Rayah El-Tawfiki and near Banha Power Plant were taken into consideration. From the historical records of MWRI data for El-Rayah El-Tawfiki, the minimum flow discharge conditions during the winter time of 25.11 m^3/s with the corresponding water level of (11.10) m above mean sea level were considered representing the critical flow conditions in El-Rayah El-Tawfiki. In all scenarios, the operation mode of the plant was taken as one cooling water pump, (CW pump), in operation with temperature difference, (Δt), of 10 ^oC. The inflow of water needed for the plant cooling process is 9.13 m^3/s, whereas the river flow at the plant location is 25.11 m^3/s, i.e. the water abstraction by the plant is 36 % of the main river flow. Table (1) presents a summary of the model scenarios of surface and multi-port diffuser outfalls with different configurations tested in this research.

The tested outfalls parameters comprise the number of diffuser rows, the nozzles diameters, the space between nozzles and nozzles number. The scenarios were carried out under the condition of operating one or two rows of the diffuser with the variation of the other parameters as follows:
Nozzles diameters: 700 mm and 450 mm
Nozzles number : 6, 12, and 24
Nozzles space : 2m and 4m
For all tests:-

Stream discharge = 25 m^3/s,

Stream water level = 11.1m,

Outfall discharge = 9.13 m^3/s, and Temperature difference (Δt) = 10 ^oC.

3. MODEL RESULTS AND ANALYSIS

The model scenarios results indicated that the outfalls which have diffusers with nozzles near the water surface or have high jet velocities in shallow water depth did not give good results. Despite of its efficiency for diluting the thermal concentration downstream of the plant outfall, they generate high cross flow velocities and water surface turbulence and some recirculation to the intake direction. This can be attributed to the presence of the nozzles near the water surface especially in case of high jet velocities which means that there is no sufficient water depth for mixing and consequently a high turbulence and high cross flow can be generated [4]. The recirculation of hot water to the plant intake can take place, especially in case of low ambient water velocities. This recirculation can affect the plant efficiency. The high cross flow velocities in the ambient due to the high jet velocities from the plant outfall can affect the stability of the small fish boats near the plant outfall. So, these diffusers which have high jet velocities or nozzles near the water surface were excluded from the upcoming comparison which can determine the optimal diffuser configurations that verify the best results of reducing the thermal concentration and the mixing zone dimensions downstream of the plant outfall.

The surface plant outfall which was tested earlier was added to the comparison to show the gradual improvement of thermal concentration downstream of the plant outfall due to the change of the outfall structure from surface outfall to multi-port diffuser outfall as follow:
Scenario 1: surface discharge outfall
Scenario 2: diffuser with 24 nozzles in two rows – nozzle diameter (700 mm)
Scenario 4: diffuser with 12 nozzles in one row near the bed level – nozzle diameter (700 mm)
Scenario 5: diffuser with 12 nozzles in two rows– nozzle diameter (700 mm)
Scenario 8: diffuser with 24 nozzles in two rows– nozzle diameter (450 mm)
Scenario 10: diffuser with 12 nozzles in one row near the bed level – nozzle diameter (450 mm)
Scenario 11: diffuser with 12 nozzles in two rows– nozzle diameter (450 mm)

The comparison between these diffusers was carried out to determine the mixing zone dimensions downstream of the outfall for each diffuser, (area - maximum length parallel to the shore line – maximum width perpendicular to the shore line). The dimensions of thermal contours which have temperature rise from 5 to 9 ^oC above the ambient were determined for each diffuser. Table (2) shows the mixing zone dimensions for the above scenarios.

From Table (2), it can be seen that in the test No. 2 which has a diffuser of 24 nozzles with nozzle diameter of 700 mm and nozzle space 2m, the mixing zone occupied an area of 254 m long and 21 m wide downstream of the plant outfall. With the decrease of nozzle diameter to 450 mm instead of 700 mm, an improvement of the mixing zone dimensions as it occupied an area of 122 m long and 12 m wide. This means that the decrease of nozzles diameter to 450 mm led to a mixing zone of area that represents about 35 % of the mixing zone area for the case of diffuser with nozzles diameter of 700 mm. This indicates the importance of nozzle diameter on the decrease of thermal concentration downstream of the plant outfall as the decrease of nozzle diameter leads to the decrease of thermal concentration downstream of the plant outfall. With the decrease of nozzles number to 12 instead of 24

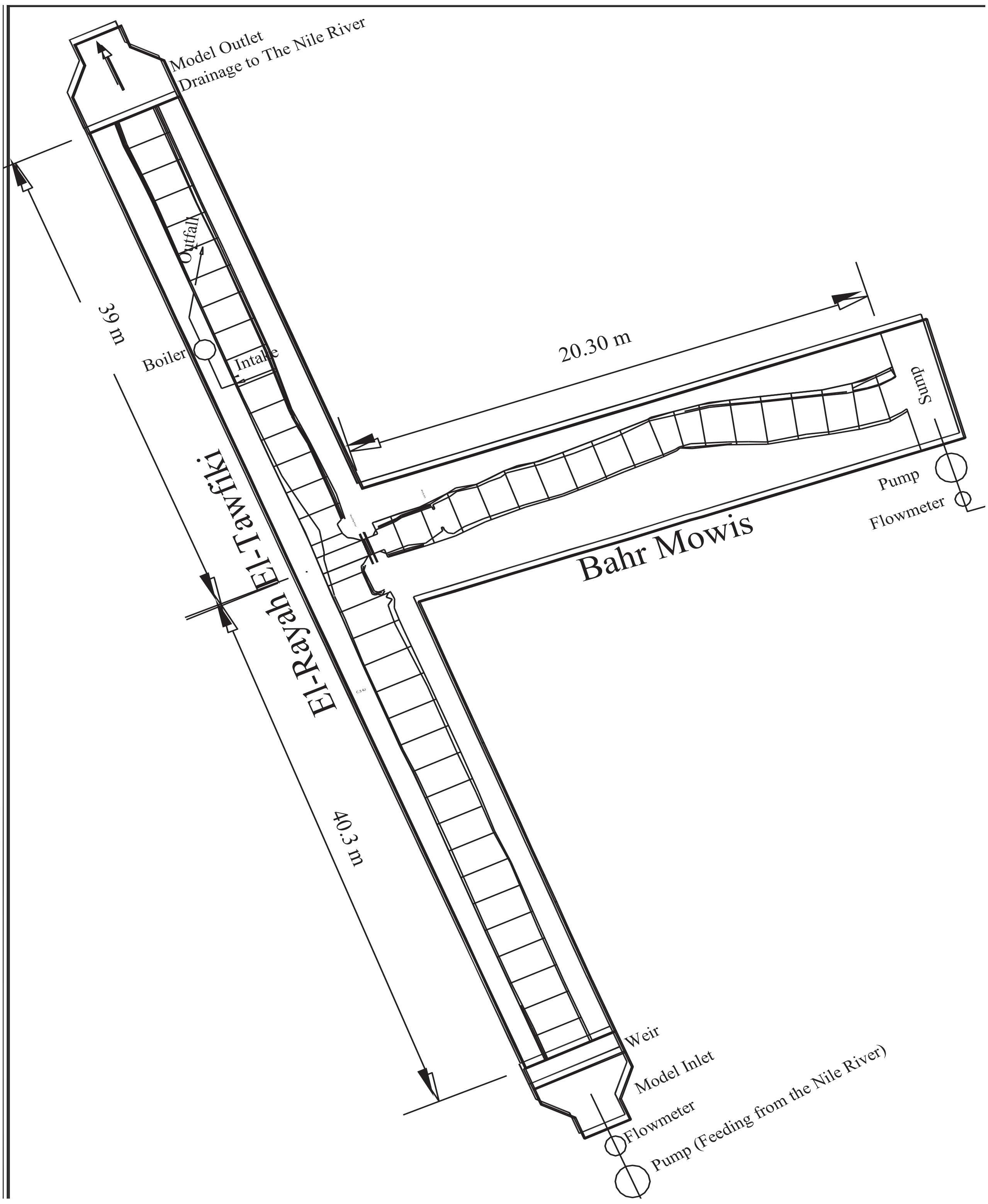

Figure (1): General layout of Banha power plant physical model

Table (1): Model scenarios

Model Scenarios	Nozzles Diameter (mm)	Nozzles Spacing (m)	Nozzles Number	Diffuser Rows Condition
Scenario1	Surface outfall			
Scenario2	700	2	24	Two Rows
Scenario3	700	2	12	One Row (Upper Row)
Scenario4	700	2	12	One Row (Lower Row)
Scenario5	700	4	12	Two Rows
Scenario6	700	4	6	One Row (Upper Row)
Scenario7	700	4	6	One Row (Lower Row)
Scenario8	450	2	24	Two Rows
Scenario9	450	2	12	One Row (Upper Row)
Scenario10	450	2	12	One Row (Lower Row)
Scenario11	450	4	12	Two Rows

Table (2): Mixing zone dimensions for the optimum scenarios conditions

Scenario No.	Diffuser Rows	No. of Nozzles	Nozzles Diameter (mm)	Nozzles Spacing (m)	Mixing Zone Dimensions		
					Area (m^2)	Length (m)	Width (m)
1	Surface Outfall structure				4100	265	45
2	2	24	700	2	2640	254	21
4	1 (near the bed)	12	700	2	940	140	12
5	2	12	700	4	1860	222	14
8	2	24	450	2	658	122	12
10	1(near the bed)	12	450	2	245	100	8
11	2	12	450	4	299	123	10

and nozzle spacing to 4 m instead of 2 m, the mixing zone area has occupied an area of 100 m long and 8 m wide which represents 55 % of the mixing zone area for the case of the diffuser with nozzles number of 24 with 450 mm diameter and nozzles spacing of 2m. This indicates that the high effect of nozzles space and number on the thermal concentration downstream of the plant outfall as the increase of nozzles spacing and the decrease of nozzles number led to the improvement of jets dispersions and diffusion and consequently the decrease of thermal concentration.

Figures from (2) till (4), show the relationship between the temperature rise and the mixing zone area, length and width downstream of the plant outfall respectively.

Figure (2) shows that the mixing zone area has decreased from 528 m^2 to 42 m^2 for the contour line of temperature rise of 9 ^oC above the ambient water temperature and from 4100 m^2 to 245 m^2 for the contour line of temperature rise of 5 ^oC above the ambient water temperature due to the change of surface outfall (Scenario1) to diffuser outfall of 12 nozzle with 450 mm diameter arranged in one row near the bed level (scenario 5). The dissolved Oxygen in El-Rayah El-Tawfiki had high value due to the use of surface outfall discharge and it was necessary to decrease the occupied mixing zone area related to this high value by using multiport diffuser outfall to meet the average water quality index (AWQI) criteria [11]. Figure (3) explains how the maximum mixing zone length parallel to the shore line decreased from 50 m to 12m for the contour line of temperature rise 9 ^oC and from 265 to 100 m for the contour line of temperature rise 5 ^oC for the two above scenarios. Also,

From Figure (4), the maximum mixing zone width perpendicular to the shore line decreases from 16 m to 4m for the contour line of temperature rise 9 ^oC and from 45 to 8 m for the contour line of temperature rise 5 ^oC for the two above scenarios. This decrease of mixing zone dimensions and areas was carried out gradually by the change of diffuser configurations as described in the above scenarios until the optimum mixing zone dimensions were reached (Scenarios 10 and 11).

Some thermal cross sections were deduced from the collected data for each tested outfall diffuser to indicate the change of thermal distribution downstream of the plant outfall. Figures from (5) till (7), show these cross sections at a distance of 10, 85, and 335 m downstream of the plant outfall axis. From these figures, it can be seen that the diffuser which has 24 nozzles arranged in two rows with a nozzle diameter of 700 mm had the maximum temperature rise distribution. An improvement in the thermal concentration was observed due to the decrease of the diffuser nozzle number from 24 to 12 and the increase of nozzle distance to be 4 m instead of 2m, as the average value of temperature rise decreased from 5 ^oC to 4.5 ^oC at cross section No.3 (335 m downstream of the plant outfall axis). More improvements of thermal concentration were observed when the diffuser nozzles diameter was reduced from 700 mm to 450 mm as the average temperature rise decreased from 4.5 ^oC to 3 ^oC at cross section No.3. These results indicate that the thermal concentration downstream of the plant outfall decrease with the decrease of nozzles number and diameter and with the increase of nozzle spacing [9], [10]. Upon these results, it was decided to use one from two diffusers which gave the best results in reducing the temperature rise downstream of the plant outfall (Scenarios 10 and 11). So, the diffuser which has 12 nozzles of 450 mm diameter each arranged in one row near the bed level is considered to be the best diffuser (scenario 10) and comes next to it directly the diffuser which has the same configuration but with nozzles arranged along two rows (scenario 11).

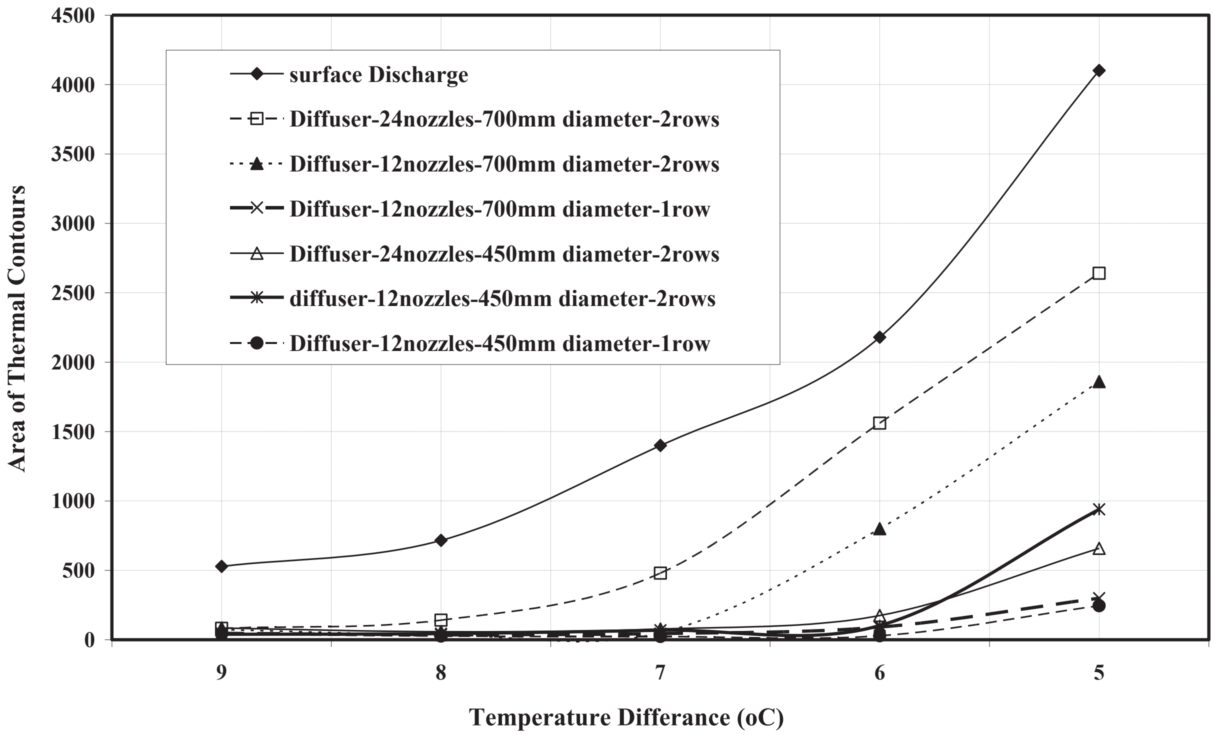

Figure (2): Relationship between mixing zone area and temperature rise above ambient

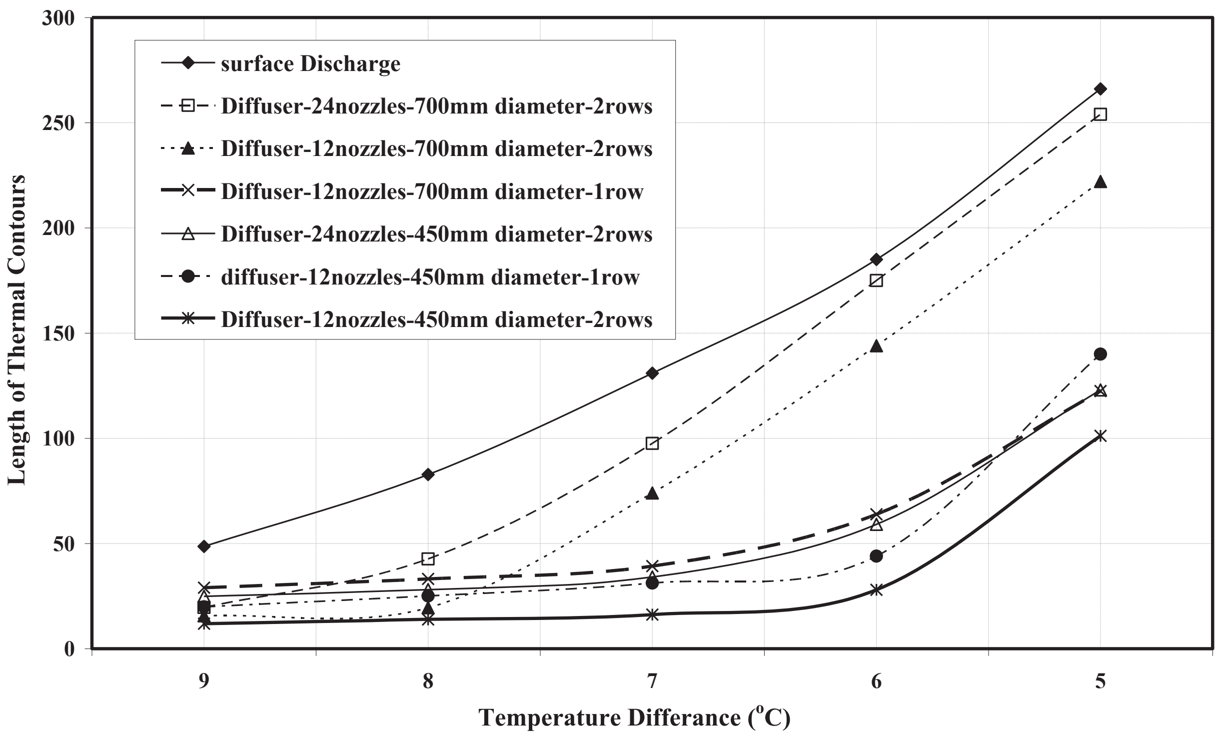

Figure (3): Relationship between mixing zone length and temperature rise above ambient

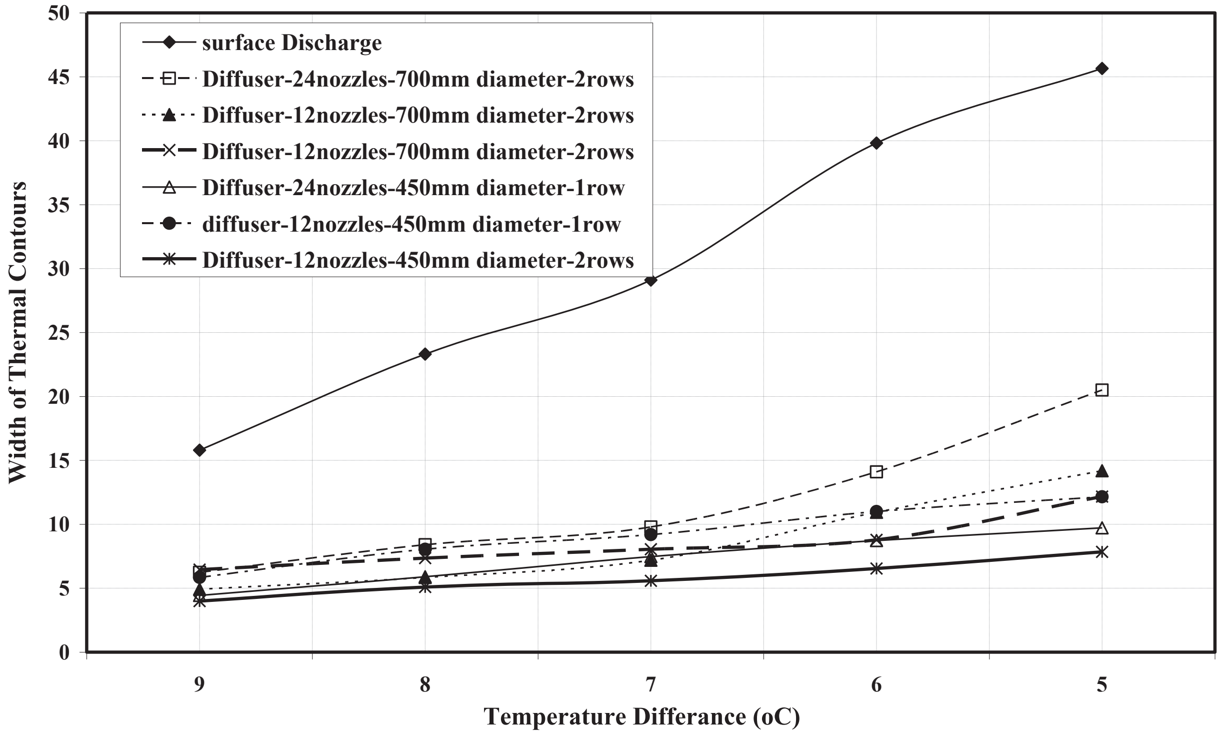

Figure (4): Relationship between the mixing zone width and temperature rise above the ambient

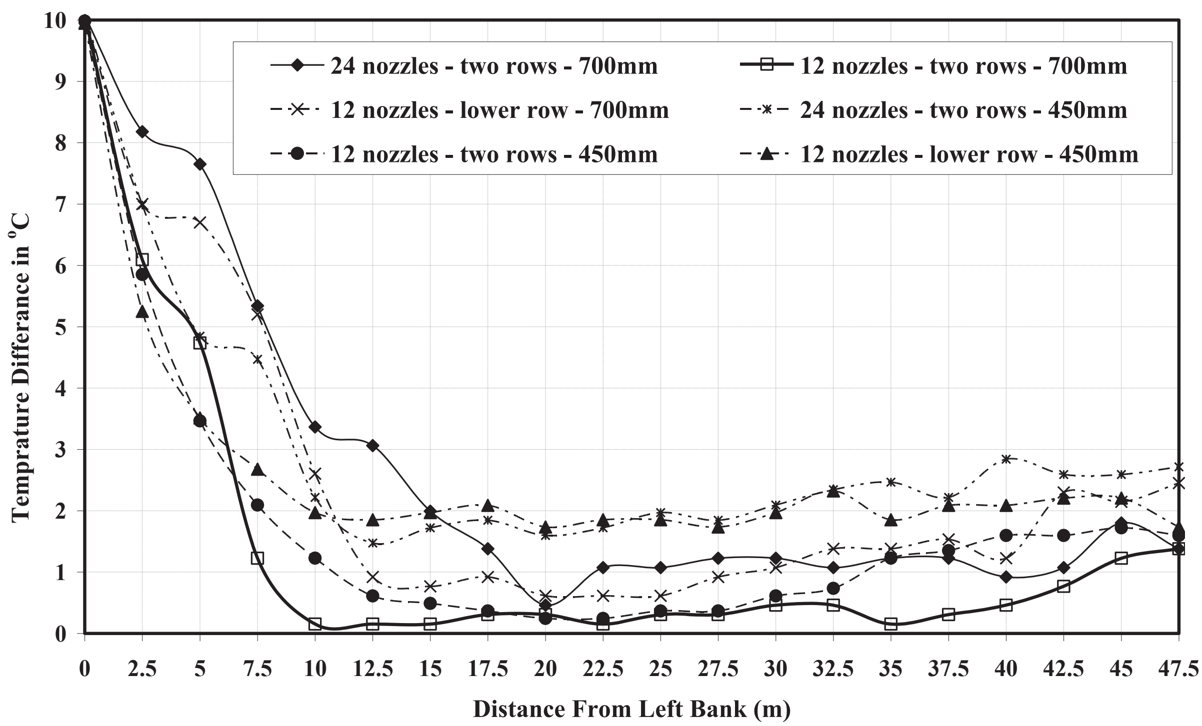

Figure (5): Thermal Cross Section at 10 m downstream of the outfall axis

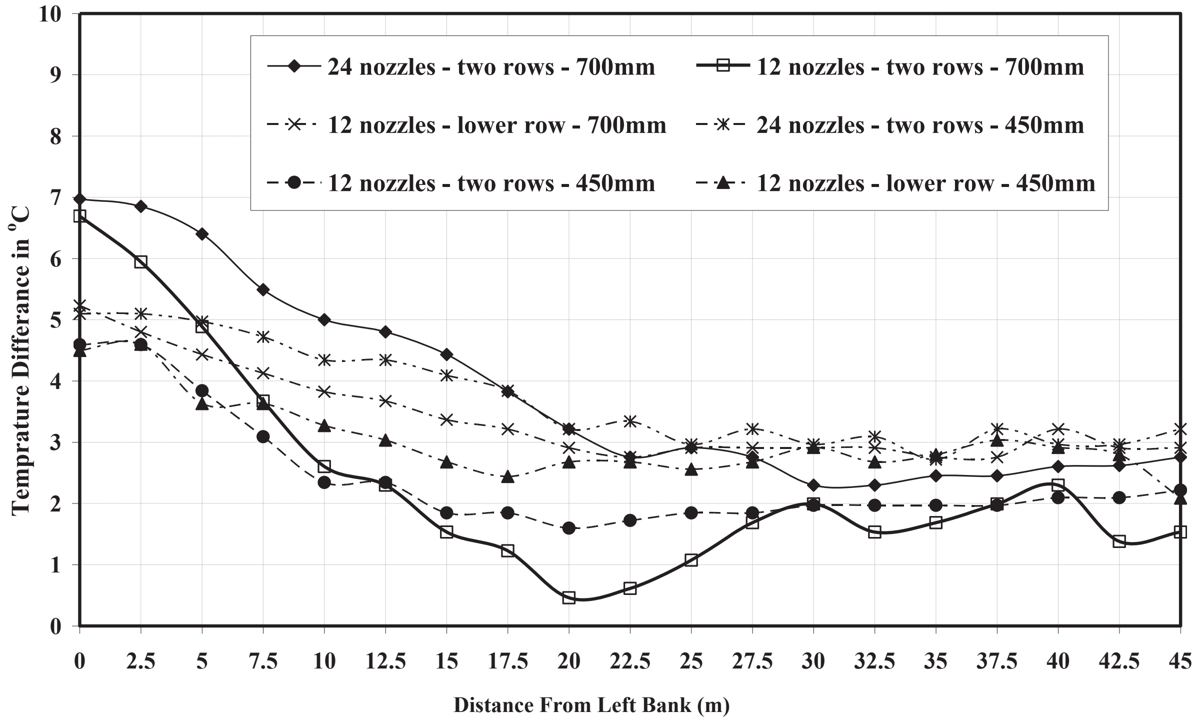

Figure (6): Thermal cross section at 85 m downstream of the outfall axis

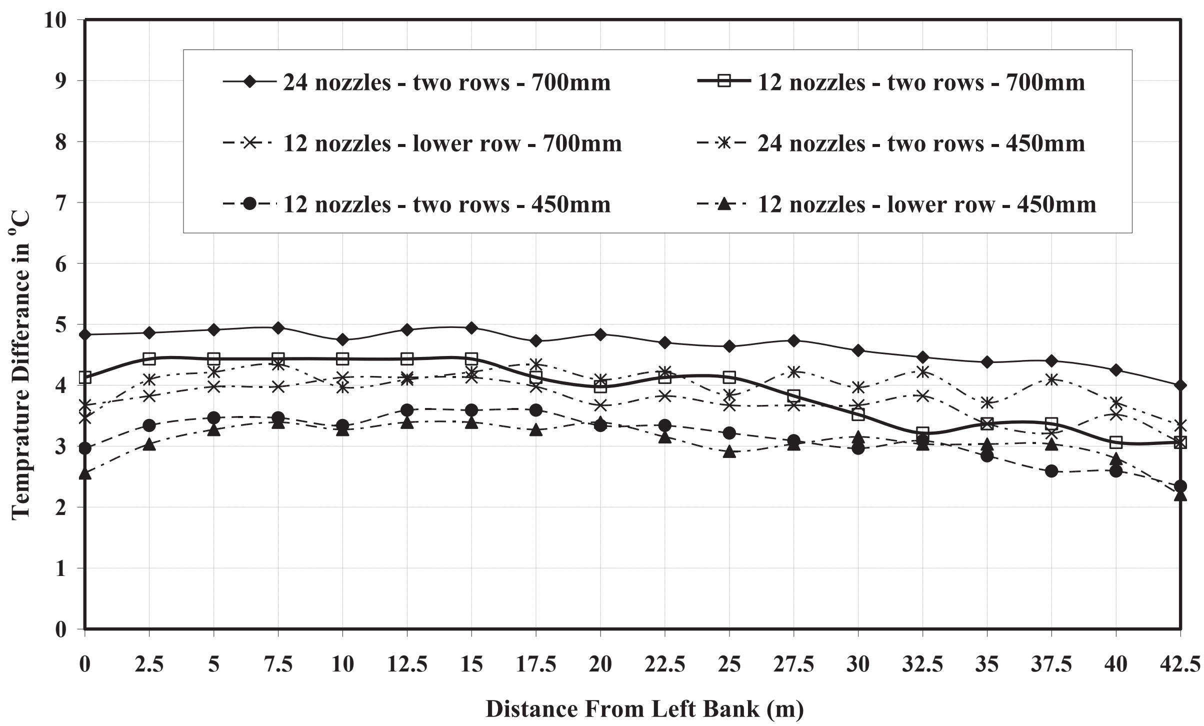

Figure (7): Thermal cross section at 335 m downstream of the outfall axis

4. CONCLUSIONS AND RECOMMENDATIONS

4.1 Conclusions

The obtained results indicated that the diffusers which have two rows in operation has achieved almost the same mixing zone area as in case of diffuser with only one row in operation near the bed level. Although the diffuser which has two nozzles rows increases the jets diffusion due to the division of original plume into many separate jets in the vertical and horizontal directions, the results indicated that it is better to use diffuser with one row only near the bed level in case of high nozzle velocities to avoid high cross flow and surface turbulence which may affect the fish boats near the plant location.

From the study results, it can be concluded that a diffuser with the same discharge can give the best results of diluting the thermal concentration downstream of the plant outfall and decrease the mixing zone area by decreasing the diffuser nozzle number and diameter and increasing the nozzle spacing [10]. Upon these results, the diffuser which has 12 nozzles with nozzle diameter of 450 mm arranged in one row near El-Rayah El-Tawfiki bed level was chosen to be the optimum solution for diluting the thermal concentration downstream of the plant outfall without any recirculation of hot water to the plant intake and also without high cross flow or turbulence which may affect the small fishing boats near the plant. This indicates the importance of using the physical models for deducing the thermal pollutions downstream of the plant outfalls and consequently determines the optimal layout and configuration of the diffusers outfalls as the design equations are not enough in that case.

4.2 Recommendations

Based on the study results, it can be recommended that:

- A multi-port diffuser for the plant outfall structure gives good results of diluting the thermal concentration downstream of the plant and confined the temperature rise to be less than 5 ^{o}C outside the mixing zone. The optimal diffuser has 12 nozzle with nozzle diameter 450 mm arranged in one row near the ambient bed level. The space between the nozzles is 2 m [as mentioned earlier and to serve the purpose of this research for checking a lot of diffusers arrangements, a discharge of 9.13 m^3/s was considered the dominant discharge needed for the plant cooling system – case of one unit of the plant is in operation).
- More intensive experimental studies need to be carried out to study the thermal concentration distribution in the vertical and lateral directions, as the concentration in these two directions is important in case of multi-port diffuser discharging hot water into lake or sea.

REFERENCES

[1] Adams, E. E.,“ Submerged Multi-port Diffusers in Shallow Water with Current,” Thesis presented to the Massachusetts Institute of Technology, at Cambridge, Mass., in partial fulfilment of the requirements for the degree of Master of Science,1972.

[2] Adams, E. E., and Stolzenbach, K.D., “Comparison of Alternative Diffuser Designs for the Discharge of Heated Water into Shallow Receiving Water,” Proc. Conf. on Waste Heat Management and Utilization, Vol. I., Miami Beach, pp. 26-171, 1977.

[3] Adams E. E. and Trowbridge, J.H., “Near-Field Performance of Staged Diffusers in Shallow Water” M.I.T. Energy Lab. Report No. MIT-EL-79-015, April 1979.

[4] Ahmed K. M., and Rajaratnam, N., “Dilution of Multiple Non-Buoyant Circular Jets in Cross Flows” J. Envir. Eng., ASCE, Vol. 124, No. 1, January 1998.

[5] Buhler, J., “Model Studies on Multi-port Out-falls in Unstratified Stagnant or Flowing Receiving Water” Thesis presented to the University of California, at Berkeley, Calif., in partial fulfillment of the requirements for the degree of Doctor of Philosophy, 1974.

[6] Essam, A.H., Ahamed, F.K., Fawzy, M. and Salah El-Deen, M., “River Nile Environmental Quality in the Vicinity of Thermal Power Stations”, Nile 2002 Conf., Aswan, Egypt, 1993.

[7] Gerhard, H. J., “Multi-port Diffusers for Heat Disposal (Summary)” Journal of the Hydraulic Division, ASCE, Vol. 108, No. HY 12, pp. 1425-1465, December, 1982.

[8] HRI, Hydrothermal Study for Banha Power Plant (750 MW) Modified outfall structure, Report No. 187/2011, September, 2011.

[9] Miller D.S. and Brighouse B.A., “Thermal Discharge” A guide to power and process plant cooling water discharge into river, lakes and seas, 1984.

[10] Shawky, Y. M., “Near Field Performance of River Multi-port Diffuser” Thesis presented to the University of Minufiya, in partial fulfillment of the requirements for the degree of Doctor of Philosophy, 2001.

[11] Shawky, Y. M., Nada A., and Fahmy Salah Abdelhaleem, “Environmental and Hydraulic Design of Thermal Power Plants Outfalls – Case Study: Banha Power Plant, Egypt” Ain Shams Engineering Journal, Vol. 4, pp. 333-342, Spet. 2013.