American Journal of Engineering and Applied Sciences

Original Research Paper

Dilution of Single Port Submerged Diffuser Clogged by a Free Rotating Propeller

[1]Ahmed Ashmawy, [1]Mahmoud Aboelnasr,
[2]Entesar Abdallh El-Ghorab, [3]Mofreh Hashim and [1]Hamdy Abotaleb

[1]*Department of Mechanical Power Engineering, Faculty of Engineering, Ain Shams University, Cairo, Egypt*
[2]*Hydraulics Research Institute, National Water Research Center, Qalubiya, Egypt*
[3]*Mechanical and Electrical Research Institute, National Water Research Center, Qalubiya, Egypt*

Article history
Received: 03-03-2020
Revised: 14-04-2020
Accepted: 04-05-2020

Corresponding Author:
Ahmed Ashmawy
Department of Mechanical Power Engineering, Faculty of Engineering, Ain Shams University, Cairo, Egypt
Email: Ahmed_ashmawy@nwrc.gov.eg

Abstract: Thermal pollution raises the water source temperature and thus causes a change in the physical, chemical, and biochemical properties of water. That makes limitations for designing cooling systems, which increase its cost. Single port submerged diffuser is used for the disposal of hot water. It is a form of once-through cooling systems, and it is between the surface discharge and the multi-port submerged diffuser in comparison with cost and dilution of temperature. A new method was used to increase the dilution of Single-port submerged diffuser discharging hot water in ambient temperature. The diffuser was clogged by a free rotating propeller, to study its effect on temperature distributions experimentally, a model with 18 m long and 2 m was constructed. The results were confirmed using numerical simulations. An experimental test was carried on for three different Reynolds number Ratios Re_r (0.4, 0.21, and 0.13). Adding propeller at (L = 0.1), the plume centre temperature ratios were decreased by 22.62%, 46.23% and 48.57%. There was an agreement between experimental and numerical results for temperature distributions through the model. Clogging single port submerged diffuser increase dilution of the thermal plume.

Keywords: Thermal Pollution, Propeller, Dilution, Clogging, Single-Port Submerged Diffuser, Temperature Distribution

Introduction

Many industries use cooling systems, which use water as a cooling fluid, like thermal power stations and many other applications. Most of these industries use a once-through cooling system, which causes thermal pollution. Thermal pollution raises the water temperature and thus causes a change in its physical, chemical, and biochemical properties, which affect aquatic life and disturb the environment. That makes limitations for designing a cooling system, which increases its cost. Single port submerged diffuser is in between the surface discharge and the multi-port submerged diffuser in comparison with cost and dilution of temperature. George and Panayotis (1989), studied dilution of water discharged from around vertical jet blocked by a thin concentric disc in salted water, the results showed that blocking the outlet of single round port submerged diffuser with a disc increase dilution rate in the region behind the disc. Lilun and Jiin-Jen (1996) studied the mixing process of jet blocked with a pierced disc, and the results showed that using jet obstructed by pierced disc increases the dilution rate of injected fluid. Wen-Xin *et al.* (2006b) studied the buoyant jet from a square diffuser blocked with a square disc discharge in static ambient. In this research work, a new method was used to increase the dilution of Single-port submerged diffuser discharging hot water in ambient temperature. The propeller is rotating due to the water speed discharged from the diffuser. The effect of the propeller on temperature was studied experimentally, and the results were confirmed with numerical simulations.

Materials and Methods

An experimental model was built in hydraulic research institute to study the effect of clogging Single port submerged diffuser by a free rotating propeller.

Figure 1 shows a single port submerged diffuser clogged with a propeller. The experimental model consists of three parts entrance, main body, and exit. The entrance consists of a receiving tank (3 m long, 0.5 m wide, 1.5 m height), to dissipate the energy of water and excessive turbulence, a weir with (0.4 m height), and (2 m wide) and screen box field with large stones were used. Main body; model bed made from concrete and it is (18 m) long and (2 m) wide, model exit; model exit consists of a revolving tailgate used to control the water level in model manually by adjusting the height of the gate. Two pumps were used to deliver water to the model; the main pump used to deliver ambient water to the model, it has the flowing specification (Discharge (Q) = 0.07 m^3/s, Head (h) = 10 m.), electromagnetic flow meter was used to measure ambient water flow rate. The second pump used to deliver ambient water to the electric water heater and then pumped to the model, it has the flowing specification (Discharge (Q) = 0.012 m^3/s, Head (h) = 12 m). Ultrasonic flow meter was used to measure hot water flow rate, it is inserted on two-inch pipe, which delivers hot water to the model.

Electric water heater was used as a source for hot water. The temperature of the water is controlled by electric switches and thermostats.

Flow similarity (Amer, 2005)**,** Reynolds number (Re) model ≥ 2000, and this means that internal friction force can be neglected so that the gravity force is dominant and that can be expressed by Froude number for the flow in model F_r <1. Thus similarity between open channel flow and rigid boundaries model is attained. Figure 2 shows the model schematic diagram.

CFD Three-dimensional unsteady state simulation was used to confirm the result of experimental work. Finite volume was used as a discretization method for partial differential equations.

Experimental Procedures

Experimental work was accomplished to study temperature distributions of single port submerged diffuser discharging hot water in ambient temperature clogged by a free rotating propeller. The diffuser consists of two parts; first part was a horizontal pipe with (0.25 m long and 0.05 m diameter), the second part was (0.25 m long and 0.05 m diameter) and inclined with an angle of (30°) with horizontal direction and (20°) with the vertical direction. The propeller was placed at the outlet of the diffuser at a distance $\left(L = \frac{l}{d_{pipe}}\right)$ equal (0.1). Water depth above diffuser was one-time pipe diameter, the difference between the ambient water and hot water discharged was (Δ T = 10°C). Three Reynolds number ratios (Re_r) between ambient water and hot water flow were considered (0.4, 0.21, and 0.13). Hot water temperature and ambient water temperature were continuously measured and, hot water temperature adjusted as necessary. The surface temperature distribution was measured across the model using thermistors after steady-state conditions had been reached. The surface temperature distribution was measured upstream and downstream from a single port submerged. Temperature readings were taken at the selected positions for each experiment. At each run, data acquisition was set to measure temperature three times at each position. Each time scanned in two minutes.

Fig. 1: Single port submerged diffuser clogged with a propeller

DOI: 10.3844/ajeassp.2020.182.190

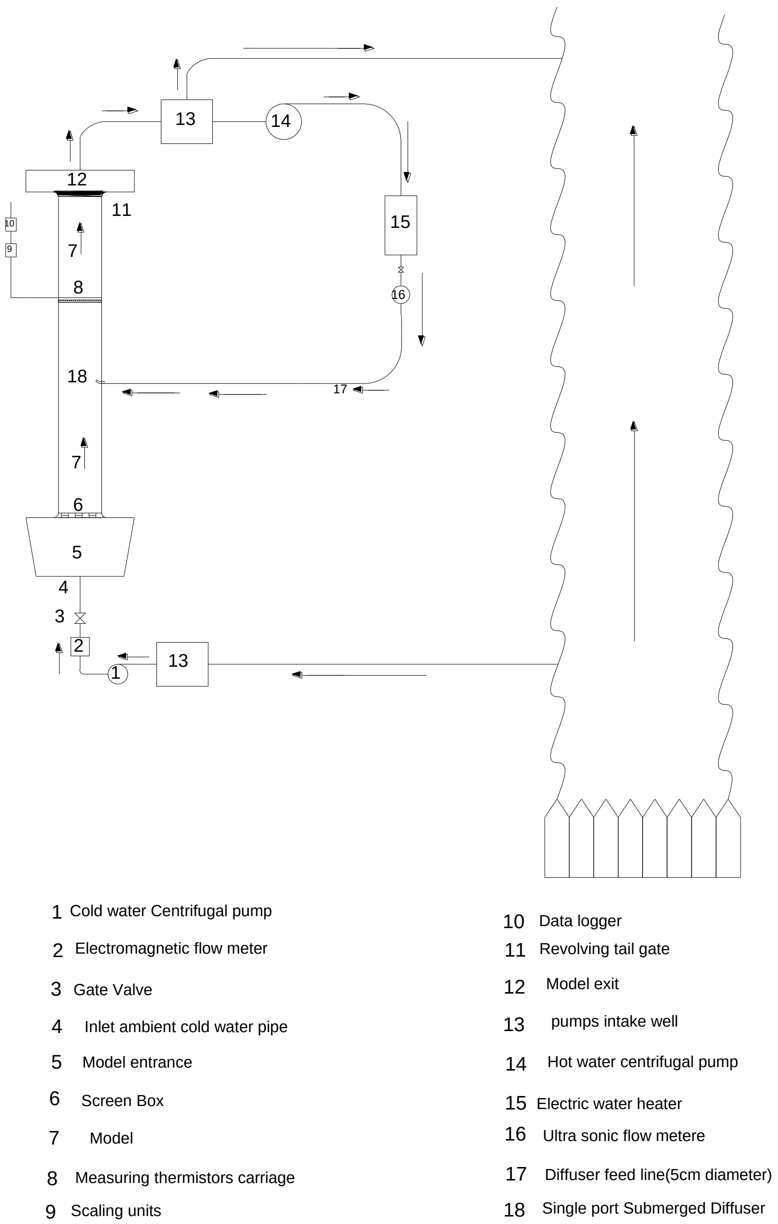

Fig. 2: Model schematic diagram

Numerical Model

Three-dimensional unsteady state simulation was used to confirm the result of experimental work. Finite volume was used as a discretization method for partial differential equations. Piecewise linear function was used to describe density as a function of temperature. (k-ε) turbulence model was used in this research work as a turbulence model. Pressure Implicit with Splitting of Operators algorithm (PISO algorithm) was used for pressure-velocity calculation. Meshing is part of the simulation process, which has influences on the accuracy, convergence, and speed of the solution. Mesh generator is used to create structured tetrahedral mesh cells with advanced size function at curvature and Proximity. An independent mesh study was made, four different mesh with different size were created to study mesh size effect on the solution. Using the boundary condition of one of the experimental tests and compare the results with the result of the simulation for each mesh size. Table 1 shows the number of cells and nodes for each mesh. Mesh (c) was used for the rest of the study. Figure 3 shows mesh (C) for flow domain and propeller.

DOI: 10.3844/ajeassp.2020.182.190

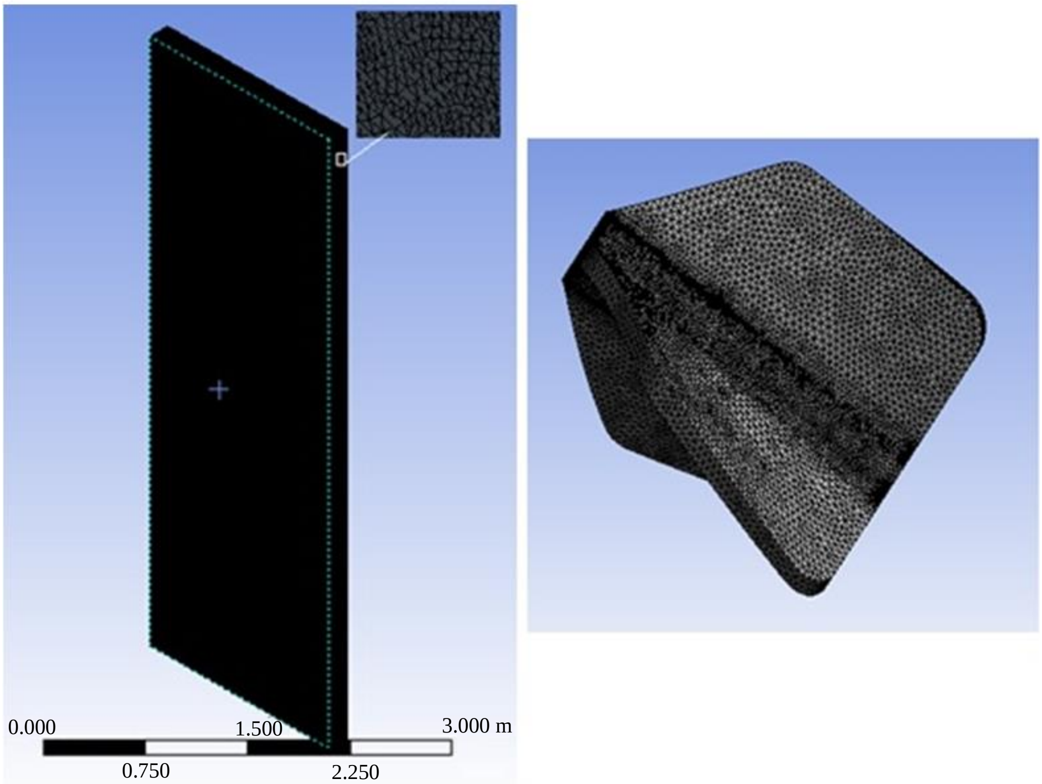

Fig. 3: Mesh (C) for flow domain and propeller

Table 1: Number of cells and nodes for each mesh

Mesh	Number of cells	Number of nodes	Plume centre temperature	Actual plume centre temperature	Error (%)
A	1123538	220578	0.91	0.71	28.17
B	9909369	1863560	0.86	0.71	21.12
C	15695699	3554809	0.785	0.71	10.5
D	19976060	3692854	0.78	0.71	9.86

Results and Discussion

The diffuser was tested without a propeller. The temperature distribution was measured on the water surface for Re_r (0.4, 0.21, and 0.13). Temperature difference ratios were plotted in upstream and downstream of the diffuser as ($\Delta T_f/\Delta T$), in the longitudinal direction and transverse direction. Figure 4 shows temperature distribution for Rer (A at Re_r = 0.4, B at Re_r = 0.21 and C at Re_r = 0.13). As Reynolds number ratio decrease, plume centre temperature increased by 14% and 17.5%, the average temperature at the end of measurements increased 104.4% and 155.5%. Temperature distributions through the model increased, temperature dilution decreased. Table 2 shows the Plume centre temperature ratio, average temperature ratio at the end of measurements.

Placing propeller at (L = 0.1), Temperature distribution was measured on water surface for Re_r (0.4, 0.21 and 0.13). The temperature difference was plotted in upstream and downstream of the diffuser as ($\Delta T_f/\Delta T$), in the longitudinal direction and transverse direction. Figure 5 shows Temperature distribution for diffuser with propeller for Re_r (A at Re_r = 0.4, *B* at Re_r = 0.21 and C at Re_r = 0.13). When adding propeller, plume centre temperature decreased by 22.62%, 46.23%, and 48.57%, temperature ratios at the end of measurements decreased by 22.63%, 46.43%, and 48.57%. Temperature distribution through the model was reduced due to adding propeller. Clogging single port submerged diffuser with propeller increases dilution of temperature. Table 3 shows the plume centre temperature ratio, average temperature ratio at the end of measurements for diffuser with a propeller. Figure 6 shows a comparison for plume centre temperature dilution through the model for the diffuser with and without propeller. The temperature dilution increase due to the rotation of the propeller, which increases mixing and reduce temperature distributions.

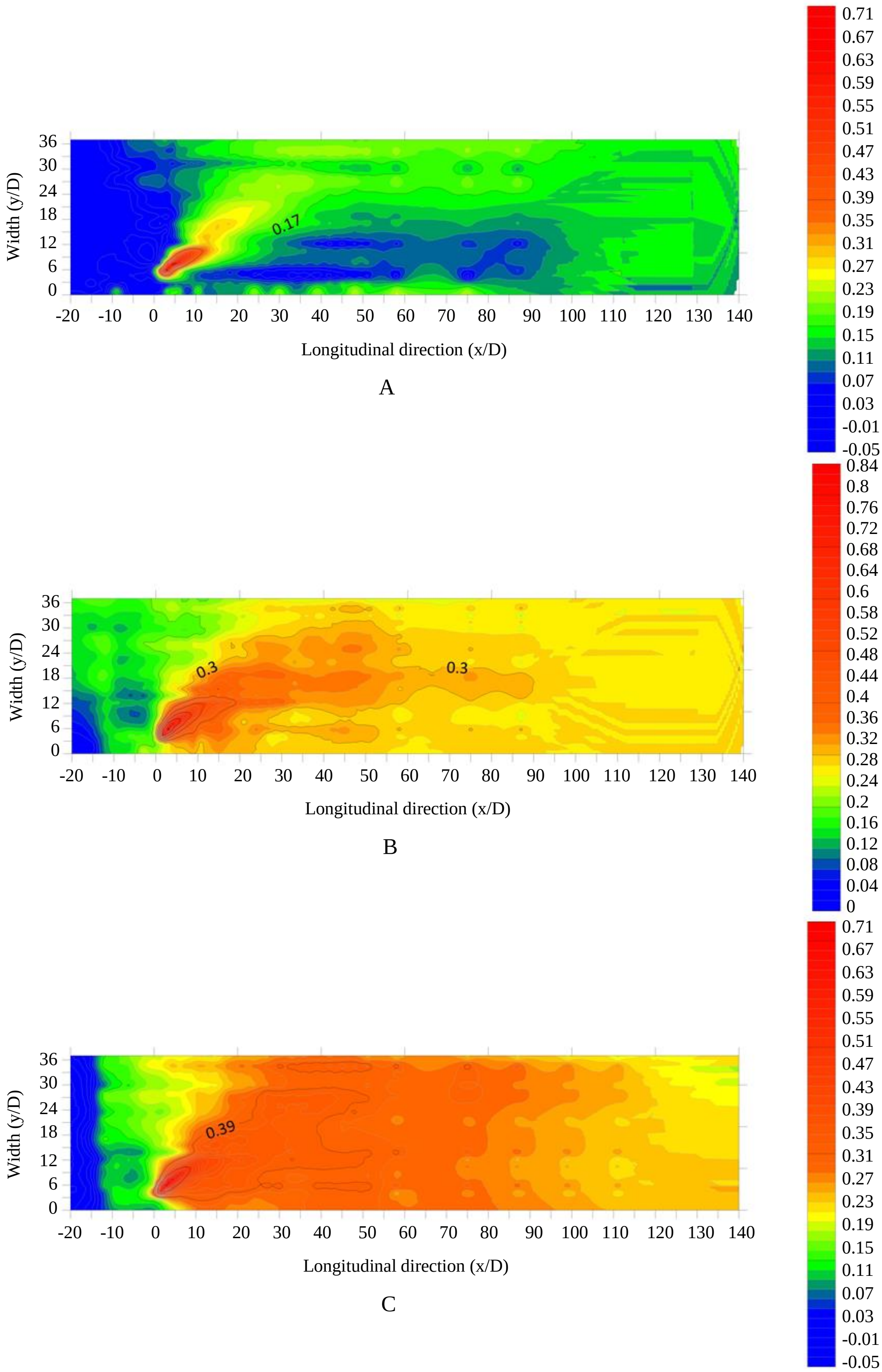

Fig. 4: Temperature distribution for Re_r (A at $Re_r = 0.4$, B at $Re_r = 0.21$ and C at $Re_r = 0.13$)

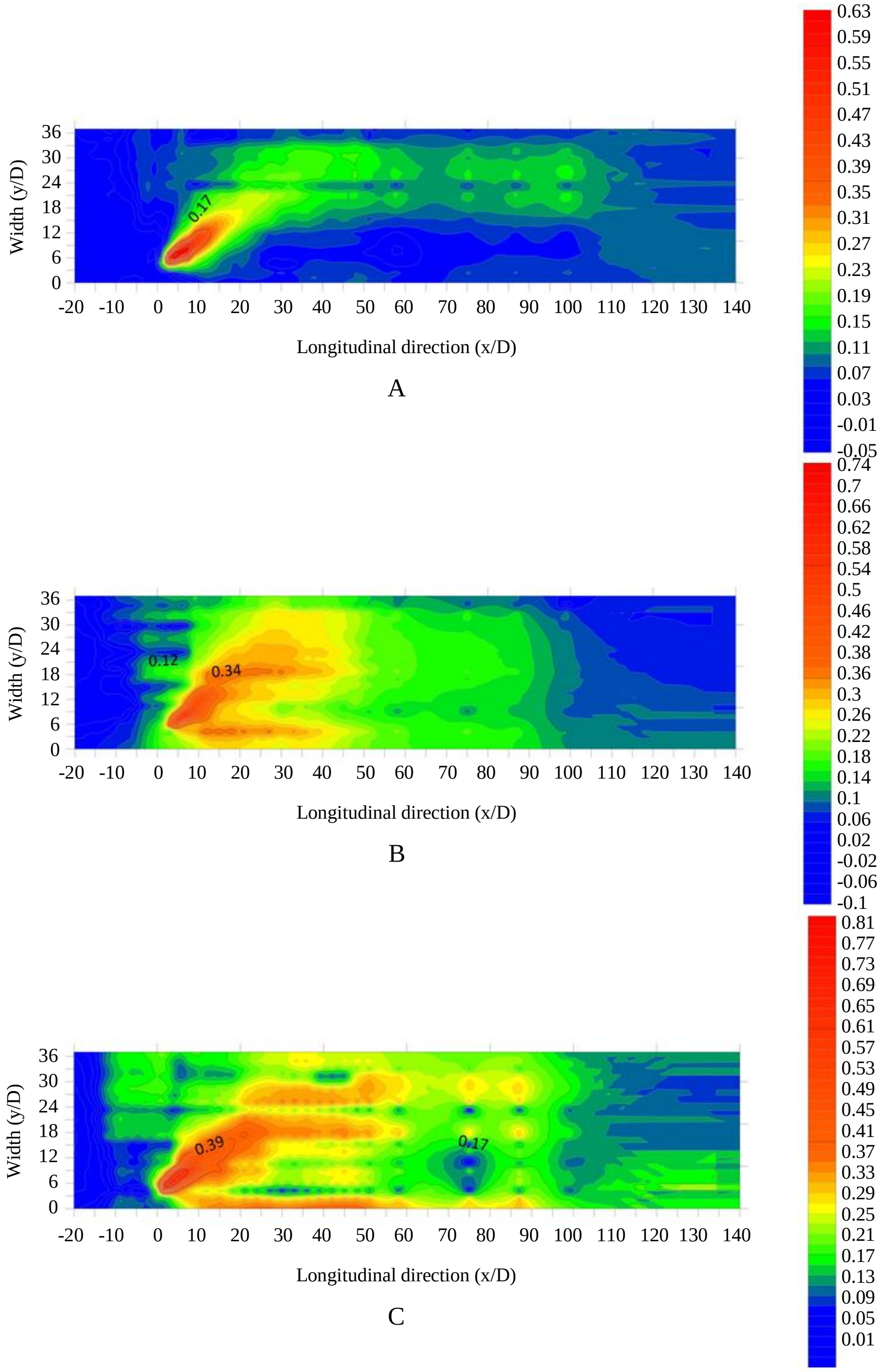

Fig. 5: Temperature distribution for diffuser with propeller for Re_r (A at Re_r = 0.4, B at Re_r = 0.21 and C at Re_r = 0.13)

Table 2: Plume centre temperature ratio, average temperature ratios at the end of measurements.

Reynolds number ratios	Plume centre temperature ratio	Average temperature ratio at the end of measurements
0.4	0.71	0.137
0.21	0.83	0.28
0.13	0.86	0.35

Comparing experimental results and numerical results for the diffuser with a propeller at (L = 0.1), it was found that there was an agreement between temperature distributions through the model for experimental results and numerical simulations. The difference between plume centre temperatures ratios for experimental results and numerical simulations when adding propeller, were 15.4%, 13.7%, and 8.64%. The numerical simulation confirmed that adding propeller increases plume dilution. Figure 7 shows a comparison between temperature distribution through the model for Re_r = 0.4 for experimental result and numerical simulation result for diffuser with a propeller at (L = 0.1), Fig. 8 shows a comparison between temperature distribution through the model for Re_r = 0.21 for experimental result and numerical simulation result for the diffuser with a propeller at (L = 0.1), Fig. 9 shows a comparison between temperature distribution through the model for Re_r = 0.13 for experimental result and numerical simulation result for the diffuser with a propeller at (L = 0.1).

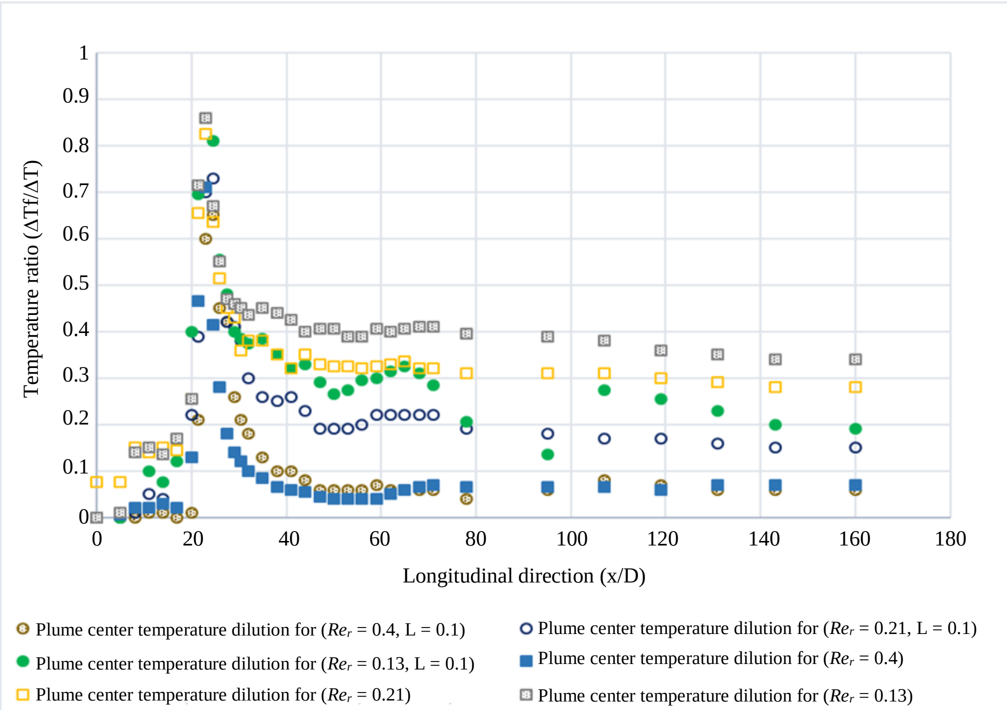

Fig. 6: Comparison for plume center temperature dilution for diffuser with and without propeller

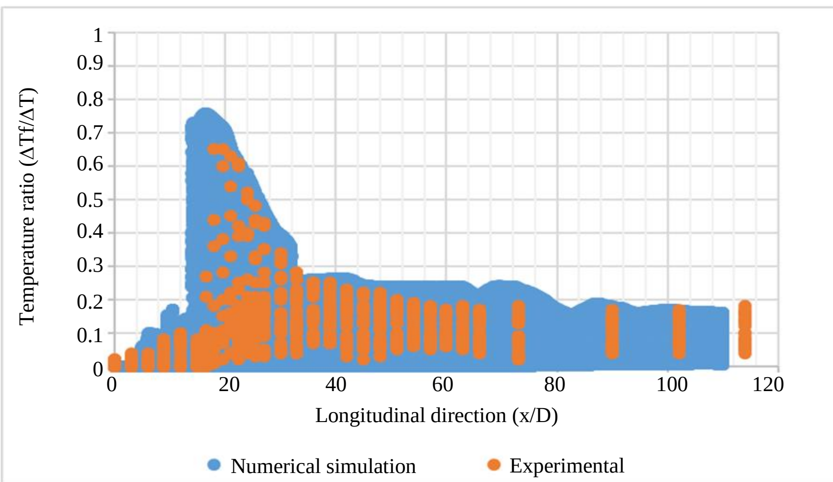

Fig. 7: Comparison between temperature distribution through the model for Re_r = 0.4 for experimental result and numerical simulation result for the diffuser with a propeller at (L = 0.1)

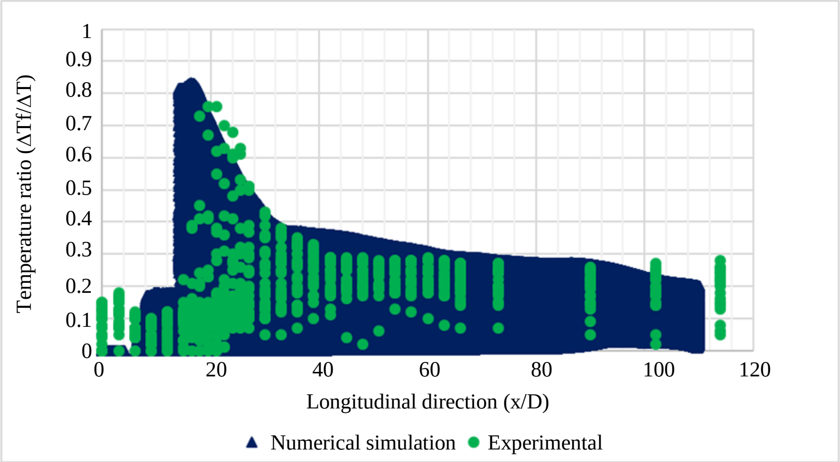

Fig. 8: Comparison between temperature distribution through the model for Re_r = 0.21 for experimental result and numerical simulation result for the diffuser with a propeller at (L = 0.1)

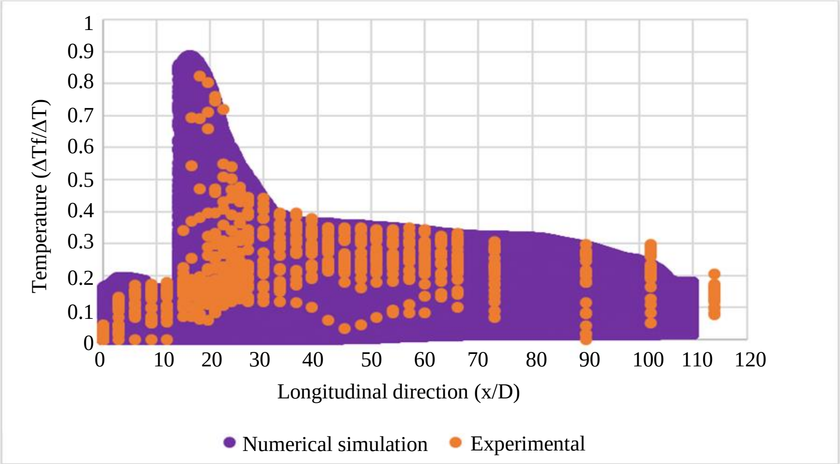

Fig. 9: Comparison between temperature distribution through the model for Re_r = 0.13 for experimental result and numerical simulation result for the diffuser with a propeller at (L = 0.1)

Table 3: Centerline temperature ratio, average temperature ratio at the end of measurements for diffuser with propeller

Reynolds number ratios	the centerline temperature ratio	Average temperature ratio at the end of measurements
0.4	0.65	0.106
0.21	0.73	0.15
0.13	0.81	0.18

Conclusion

- For diffuser without propeller as Reynolds number ratio decrease, centerline temperature ratios increased by 14% and 17.5%, the average temperature at the end of measurements increased 104.4% and 155.5%
- Adding propeller at (L = 0.1), the plume center temperature ratios decreased by 22.62%, 46.23%, 48.57%
- Adding propeller at (L = 0.1), average Temperature ratios at the end of measurements decreased by 22.63%, 46.43%, 48.57%
- The difference between plume centre temperatures ratios for experimental results and numerical simulations when adding propeller, were 15.4%, 13.7%, 8.64%
- The numerical simulation confirmed that adding propeller increases temperature dilution
- Clogging single port submerged diffuser increase dilution of the thermal plume due to the rotation of the propeller, which increases mixing and reduce temperature distributions

Acknowledgement

All authors are appreciatively acknowledging the Hydraulics Research Institute (HRI), National Water

Research Center (NWRC), Ministry of Water Resources and Irrigation (MWRI), Egypt for collaboration, and facilitating the experimental modeling.

Author's Contributions

All authors contributed to design the study, write, and revise the manuscript.

Ethics

This article is original and contains unpublished material. The corresponding author confirms that all of the other authors have read and approved the manuscript and no ethical issues involved.

Nomenclature

Re_c:	Reynolds number for ambient water flow
Re_h:	Reynolds number for hot water flow
Re_r:	Ratio between Reynolds number for ambient water and hot water flow
ΔT:	Difference between hot water temperature and ambient water temperature
ΔT_f:	Difference between the temperature at any point and ambient water temperature
Fr:	Froude number
L:	The distance between the exit of diffuser and propeller
L:	Ratio between the distance of propeller and diffuser diameter
Plume centre temperature:	Maximum temperature of the plume.
Average temperature ratio:	The sum of temperatures divided by the number of temperatures

References

Amer, A., 2005. Open Channel. Environmental Hydrology for Arid and Simi-arid, Post Graduated Regional Training Course, Egyptian committee of International Hydrological Program.

Boeriu, P., 2002. Physical Models. Hydraulic Engineering in River Basin, Post Graduated Regional Training Course, Hydraulic Research Institute, National Water Research Center, Egypt.

Daniela, M., C. Alan, B.M. Mouldi, and M. Michele, 2019. Computational simulation of round thermal jets in an ambient cross flow using a large-scale hydrodynamic mode. J. Hydraulic Res.

George, C.N. and C.Y. Panayotis, 1989. Axial dilution in obstructed round buoyant jet. J. Hydraulic Eng., 115: 71-81. DOI: 10.1061/(ASCE)0733-9429(1989)115:1(71)

Huai, W. and S. Fang, 2006. Numerical simulation of obstructed round buoyant jets in a static uniform ambient. J. Hydraulic Eng., 132: 428-431. DOI: 10.1061/(ASCE)0733-9429(2006)132:4(428)

Huang, H., R.E. Fergen, J.R. Proni and J.J. Tsai, 1998. Initial dilution equations for buoyancy dominated jets in current. J. Hydraulic Eng., 124: 105-108. DOI: 10.1061/(ASCE)0733-9429(1998)124:1(105)

Lilun, W. and L. Jiin-Jen, 1996. Enhanced mixing through perforated discs on round buoyant jet. J. Coastal Eng.

Miller, D.S. and B.A. Brighouse, 1984. Thermal discharge a guide to power and processes plant cooling water discharges into rivers, lakes and seas. Britch Hydromechanics Res. Assoc.

Tang, H.S., J. Paik, F. Sotiropoulos and T. Khangaonkar, 2008. Three-dimensional numerical modeling of initial mixing of thermal discharges at Real-Life Configurations. J. Hydraulic Eng., 134: 1210-1224. DOI: 10.1061/(ASCE)0733-9429(2008)134:9(1210)

Wendt, J.F., 2009. Computational Fluid Dynamics an Introduction. 3rd Edn., Springer-Verlag, Berlin, ISBN-10: 3540873074, pp: 332.

Wen-Xin X., L. Zhi-Wei, Q. Zhong-Dong, Z. Yu-Hong and H. Jie, 2010. Numerical simulation of horizontal buoyant wall jet. J. Hydrodynamics, 22: 58-65. DOI: 10.1016/S1001-6058(09)60028-7

Wen-Xin, H. and F. Shen-Guang, 2006. Rounded flowing states of obstructed buoyant jet. J. Applied Math. Mech., 27: 1133-1139. DOI: 10.1007/s10483-006-0814-z

Wen-Xin, H., F. Shen-Guang and D. Hui-Chao, 2006a. Behavior of obstructed square buoyant vertical jets in static ambient (I) - verification of mathematical model and numerical method. J. Applied Math. Mech., 27: 645-652. DOI: 10.1007/s10483-006-0510-y

Wen-Xin, H., F. Shen-Guang and D. Hui-Chao, 2006b. Behavior of obstructed square buoyant vertical jets in static ambient (II) - analysis on behavior of flow field. J. Applied Math. Mech., 27: 653-659. DOI: 10.1007/s10483-006-0511-y