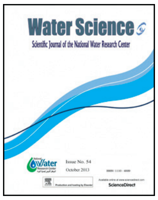
Water Science

ScienceDirect

Water Science 29 (2015) 54–67

journal homepage: www.elsevier.com/locate/wsj



Power plant intakes performance in low flow water bodies

Yasser M. Shawky, Mohamed B. Ezzat*, Mohamed M. Abdellatif

Hydraulics Research Institute (HRI), National Water Research Center, Delta Barrage, Egypt

Received 29 May 2014; received in revised form 29 December 2014; accepted 1 January 2015
Available online 21 April 2015

Abstract

This research aims to study the hot water recirculation at the power plants intakes due to the discharge from the plant cooling system into a low flow receiving water body. To achieve this objective, a 3Dnumerical model was employed to study the effect of the main parameters in this phenomena such as the plant intake length (*L*), the distance between the plant intake and outfall (*S*), the water depth under the intake skimmer wall (*h*) and the water depth just upstream the intake (*D*) on the recirculation of hot water to the plant intake. Eight scenarios were tested and two mathematical formulas accounting for the effect of these parameters on the hot water concentration at the plant intake were deduced. Physical model tests were carried out to verify the accuracy of the two deduced formulas. The study results indicated that the measured thermal concentrations in the physical model tests coincide with those calculated by the two above-mentioned mathematical formulas.

Keywords: Power plant; Intake; Recirculation; Outfall; Numerical model; Physical model

1. Introduction

The greatest single source of man-made heat addition to the world's rivers, lakes and oceans comes from electricity generating plants which have large cooling requirements. Widespread influence of thermal water depends on several factors such as the volume and temperature of hot water, receiving water body temperature (near outfall), flow or mass circulation of water around the outfall (Pitchaikani et al., 2010). Heat transfer mechanisms occurring in receiving water bodies are part of the physical properties that may be affected by tides, waves, river discharge, salinity, water depth, bathymetry, heat sources, and hydraulic structures in the water body (Huboyo and Zaman, 2007).

The possibility of recirculation of hot water into cooling water intakes is of particularly concern to hydraulic engineers in building thermal power stations because of its adverse impacts on the plant efficiency and operation. Thermal recirculation occurs when the thermal water of the outfall discharged water causes an increase of water temperature at the intake, which in turn will provide a further temperature increase at the outfall and a continuous

* Corresponding author. Tel.: +20 1006785908; fax: +20 42189539.
E-mail address: mohbahgat@gmail.com (M.B. Ezzat).
Peer review under responsibility of National Water Research Center.

Production and hosting by Elsevier

http://dx.doi.org/10.1016/j.wsj.2015.01.001
1110-4929/

temperature rise at the intake and outfall. By this, the efficiency of the cooling system reduces with time which will ultimately reduce the power plant efficiency. There are basically two types of intakes generally grouped as onshore and offshore intakes. It is desirable for the intake to be well low the water surface (maximum 0.8 of water depth to avoid the entry of sediment into the intake structure). For economic considerations, it is obviously desirable that the intake should be as close to the shore as possible (Jirka, 1979). This research focuses on the transport and advection phenomena of the plume, in particular the recirculation of effluents to the plant intake in water bodies with low velocity currents (current velocity is less than 0.1 m/s).

2. Purpose of the study and methodology

The main purpose of this study is to investigate the effect of the thermal power plant intake and water body parameters on the hot water recirculation at the plant intake. It well focuses on the effect of the following parameters: the plant intake length measured from the shore line (L), the distance between the plant intake and outfall (S), the water depth under the intake skimmer wall (h) and the water depth just upstream of the intake (D). The two parameters L and S are representing the surface area confined between the intake and outfall. So, they were taken as one term (L/S). Also, the two parameters h and D are representing the available height for the intake to abstract water from the ambient with respect to the whole water ambient depth. So, they were taken as one-dimensional less term (h/D). For this purpose, a hydro-thermal 3D numerical model was employed to investigate the effect of the above-mentioned parameters on the hot water recirculation to the plant intake. The model was well calibrated with field measurements representing the average Suez Gulf characteristics at the location of three power plants that use its water for its cooling system (Old and New Suez Power Plant, Attaka Power Plant and El-Ain El-Sokhna Power Plant) (HRI Report 159, 2010). Other parameters as the ambient current and wave direction affect the hot water recirculation at the plant intake. According to the Egyptian reference wind charts and Atlas wind maps, the maximum wind forces affecting the Suez Gulf district are in the winter season and are recognized by prevailing wind directions of (North and North West). As observed, the maximum wind intensities at the proposed project are coming from the directions (North-North West) to (West-North West), which have a very slight effect on wave propagation at the area as these wind forces are more or less blowing from land direction. On the other hand, almost small wind is blowing from the sea side, which is also exposed to very narrow and limited water fetch. Subsequently, it was decided to carry out all tests in this study under calm wind and low ambient current condition. Fig. 1 shows the overall view of the Suez Gulf Power Plants and Fig. 2 shows the Suez Gulf study area. For model development and calibration, reference is made to HRI Report 160 (2010). The three-dimensional (3D) hydrothermal numerical model was used in which different scenarios were tested to account for the effect of the intake parameters in the study area. Also, physical model for the Suez Gulf and New Suez Power Plant was used as a case study and the transport and advection of the plume was simulated to confirm the final results of the numerical model.

3. Numerical model setup

The pollutant transport task comprises the calibration and application of a dimensional model of the study area to reproduce the distribution of the flow. The relevant discharge plume can be simulated and recirculation of excess temperature is deduced (Dhanus et al., 2012).

The numerical model (Delft3D Software Package of Delft Hydraulics – The Netherlands) was used in this study. The model describes the behavior of the plume, the subsequent mixing processes and area of impact and capable of reproducing the most important processes that affect the dissipation and dilution of excess heat introduced by the plant to the receiving water body. The simulation of intake/outfall configuration was done in the model by increasing the resolution of the model grid through refining the grid cells at the vicinity of the plant. Two domains with different grid resolutions were generated. A detailed model with fine grid resolution was produced at the plant vicinity including the intake/outfall configuration of the plant. The detailed model simulates enough area of the long shore and off shore direction. The grid size of the detailed model is 10 m × 10 m. A courser grid resolution with grid size of 30 m × 30 m was carried out to cover the remaining study area which is named as the “overall model”. The Domain Decomposition technique, D.D. was used to make online coupling between the two domains, detailed and overall models. D.D. is a technique, in which a model is divided into several smaller model domains (Vollebregt, 1997). The subdivision is based on the horizontal and vertical model resolution required for adequately simulating physical processes. Then, the