Study of a submerged discharge by means of different methodologies
G. Agate, E. Ghisolfi, S. Brambilla
ENEL SpA, Centro Ricerca Idraulica e Strutturale, Via Pozzobonelli 6, 20162 Milano, Italy

Abstract

Because of the issuing of a new law concerning a new methodology of measuring the effects of thermal discharges in the sea, ENEL SpA carried out studies in order to modify the superficial discharges of a few power plants into submerged discharges as the existing superficial ones were unable to diffuse the thermal plume according to the new law requirements. This paper shows the design guidelines and the methodology applied at ENEL-CRIS in order to predict the plume dispersion both in the near field and in the far field in one of the examined power plants, located in a bay with a narrow mouth to the sea.

1 Introduction

Thermoelectric power plants need a large quantity of cooling water that is generally pumped, through a plant intake, from a natural water body (sea, lake or river) and, after the cooling process, is returned to the water body itself by means of a discharge. The returned water has an additional heat which gives a temperature rise generally from 7 to 10 °C over the ambient water temperature.

The discharged water produces a hot water plume which spreads in the natural water body. As a consequence the heat is dispersed both in the water body and in the atmosphere.

The behaviour of the plume under different meteo-marine conditions must be foreseen in order to assure, when the plant is working, the fulfilment of the limits established by the law.

It is also important to determine if recirculation of hot water from the outlet to the intake takes place. This will drop in fact the thermodynamic efficiency of the plant.
In the present paper the methodology used at ENEL-CRIS to verify the design of a new submerged discharge is described. The design of the discharge was carried out by the Engineering Department of ENEL. The new discharge allowing a suitable dilution of the cooling water is located in a bay whose extension is about 10 km² with an average depth of 10 m. The bay has two narrow mouths which allow limited exchanges with the sea: the Levante mouth and the Ponente mouth which are, respectively, about 200 m and 400 m in length.

The dilution power of a multi-port submerged discharge can be defined via heuristic or experimental formulas, see for instance Miller [1] and Adams [2], or, more carefully, via physical models or 3D mathematical models. In the first case the dilution predicted refers to an off-shore discharge and the effects of the possible recirculation are not taken into account. In the second case the models can be built in order to point out the physical phenomena that take place either in the near field or in the far field. In fact the dilution problem is connected with two scales: the small scale in which the dilution is mainly caused by the turbulent effects due to the jet; and the large scale in which the effects of the discharge produce a modification in water circulation and the plume dispersion is mainly due to the effect of ambient turbulence. In the present case we are interested in knowing the dilution capability of the discharge, the water circulation caused in the bay, the water exchange with the sea and the recirculation of heated water from the bay to the plant intake.

2 New discharge design and objectives

The new discharge system, designed by ENEL-Engineering Department is a submerged multiport diffuser in staged configuration which provides an efficient means of creating initial rapid dispersion of thermal discharges. In the staged diffusers configuration that was chosen the ports are unsymmetrically disposed along the axis (diffuser line) and all the ports have an opening angle of 20° with respect to the diffuser line. The vertical angle of discharge between the port centrelines and a horizontal plane is 20°. The main characteristics of the new discharge system are reported in table A while the discharge geometry is shown in figure 1.

The intake of the plant is also located in the bay. The main characteristics of the intake are reported in table B.

In order to verify the observance of the law the dilution of the plume has been studied splitting the interested area into two zones: the near field and the far field. The goal was to reach a temperature rise of about 3 °C on the free-surface of the sea just over the diffuser by means of turbulent mixing of the jet with ambient water. The thermal energy, remained in the plume, should be dissipated through the heat exchange with atmosphere and the water exchange at the mouths. It is also important that the heated water recirculation from the outlet to the intake will be maintained within acceptable values.
### Table A: New discharge system characteristics

<table>
<thead>
<tr>
<th>Characteristic</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>discharge system length</td>
<td>360 m</td>
</tr>
<tr>
<td>number of ports</td>
<td>20</td>
</tr>
<tr>
<td>distance between ports</td>
<td>40 m</td>
</tr>
<tr>
<td>port diameter</td>
<td>1.1 m</td>
</tr>
<tr>
<td>total discharge area</td>
<td>19.0 m²</td>
</tr>
<tr>
<td>height of the port centres above the bottom</td>
<td>2 m</td>
</tr>
<tr>
<td>flow speed at the port exit</td>
<td>3.156 m/s</td>
</tr>
<tr>
<td>total discharge flow rate</td>
<td>60 m³/s</td>
</tr>
<tr>
<td>thermal rise</td>
<td>10 °C</td>
</tr>
<tr>
<td>equivalent slot width</td>
<td>0.053 m</td>
</tr>
<tr>
<td>densimetric Froude number</td>
<td>80.26</td>
</tr>
</tbody>
</table>

### Table B: Intake characteristics

<table>
<thead>
<tr>
<th>Characteristic</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>height of intake port</td>
<td>6.0 m</td>
</tr>
<tr>
<td>width of intake port</td>
<td>32.0 m</td>
</tr>
<tr>
<td>total discharge area</td>
<td>192.0 m²</td>
</tr>
<tr>
<td>minimum depth</td>
<td>3.0 m</td>
</tr>
<tr>
<td>maximum depth</td>
<td>9.0 m/s</td>
</tr>
<tr>
<td>total intake flow rate</td>
<td>60 m³/s</td>
</tr>
</tbody>
</table>

### 3 Near field study

The near field has been studied with the aid of different methodologies: a physical model, an expert system and a three-dimensional mathematical model.

The dilution power of the discharge was evaluated by means of CORMIX Expert System. CORMIX (Cornell Mixing Zone Expert System) was developed at DeFrees Hydraulics Laboratory at Cornell University and is described in Jirka [3], Doneker [4] CORMIX User’s guide [5] and Tsanis [6]. This code is a series of software subsystems for the analysis, prediction and design of aqueous toxic or conventional pollutant discharges into water bodies.

In this case the results obtained are referred to an off-shore discharge and, therefore, the dilution power predicted is theoretical and higher than the real one.

CORMIX results were validated against physical model measurements. See ENEL-CRIS [7] and Brambilla [8]. The physical model was built at ENEL-CRIS laboratory in a pool 23 m long, 3.7 m wide and 1.9 m high. The model has a geometrical scale of 1:39. Because of the relative small size of the pool width the number of nozzles, reproduced in the physical model, were reduced to 6 and 10. The temperature rise along the centreline of the plume predicted by the expert system and the physical model were in good agreement. Figure 2 shows the comparison for the 10 nozzle case.

In order to evaluate the stability of the sea bed and the disturbance generated by the discharge to the sea trade, a 3D mathematical model was
developed using the CFD code FLOW3D [9] developed by AEA Thecnology Harwell Laboratory. This code is a multi-purpose code which performs the classical integration of the Navier-Stokes equations (continuity and momentum equations) and energy equation, via a finite volume algorithm on an eulerian grid. The mathematical model uses primitive variables: the velocity components \( u, v, w \) in \( x, y \) and \( z \) directions and pressure \( p \). The solution is achieved using the pressure-correction method. The convective terms can be modelled by different schemes (upwind, QUICK) and the use of several turbulence models is allowed (standard k-\( \varepsilon \) model, low Reynolds k-\( \varepsilon \) model, RNG k-\( \varepsilon \) model and Reynolds stress model). In these specific computations the standard k-\( \varepsilon \) model was used.

The numerical simulations were carried out considering two nozzles and the computational domain was reduced considering the symmetry of the problem with respect to the diffuser line. See ENEL-CRIS [10]. The computational domain (see figure 3) was selected 200 m long, 150 m wide and with height 13.9 m in the vicinity of the nozzles and 9.5 m far from them. At the inlet side a 0.05 m/s velocity was imposed while at the nozzles the 3 components of the velocity were imposed.

Figures 4 and 5 show the velocity field at the depth of 13.85 m and 0.05 m; figure 6 depict the velocity field in the vertical cross-section along the nozzle axis. In figure 7 the thermal rise on the centreline plume is shown comparing the physical model results and the 3D mathematical model.

4 Far field study

The far field has been studied by means of a three-dimensional mathematical model based on the TRIMDI computational code developed at ENEL and described in Ghisolfi [11] and Di Monaco [12]. This code integrates the 3D Navier-Stokes equations, simplified according to the hydrostatic pressure distribution, and the heat balance equation. These equations are solved via a finite volume algorithm. TRIMDI takes into account the spatial variation in the water density due to the variations in the temperature (or pollutant concentration) so that stratification phenomena may be reproduced. The continuity condition is imposed at the free surface.

The computational domain is 7,000 x 10,000 m\(^2\) and is subdivided into about 10,000 cells. The domain covers the bay and a wide part of the sea in front of it as can be seen in ENEL-CRIS [13].

In the vertical direction the computational domain is subdivided in 15 superimposed horizontal layers. Their thickness varies from 0.15 m for the upper layer to 1.5 m to the lower one. The thickness of these layers is constant except for the surface one whose local thickness depends upon the instantaneous free-surface elevation. In this way the following variables are obtained as average values along the thickness of each layer: orthogonal velocity components \( u, v, w \), temperature \( T \), pressure \( p \) and density \( \rho \), plus the free-surface elevation \( \zeta \).
In the sea a longshore current of 0.05 m/s is imposed and the thermal exchange with the atmosphere is imposed equal to 20 W/m²°C.

The discharge system is schematised as two horizontal slots located 2 m over the bottom, ejecting water in opposite directions. The discharge and the horizontal momentum of the ejected water are assigned equal to the real ones. In the bay is also located the intake at 450 m distance from the outlet with 192 m² area positioned at 3 to 9 m depth.

Figure 8 shows the computed thermal perturbation on the surface layer. The result has allowed to observe a self generated circulation from the bay to the open sea. Cold water enters in the bay passing through the lower part of the mouths and heated water leaves the bay from the top of the mouths. Figures 9 and 10 show the velocity and temperature values computed on a vertical line located in the middle of the mouths. The velocity field has allowed to note the development of an anticlockwise circulation in the bay. This fact helps the outflow of water at Ponente mouth (the outgoing water being 35% more than the incoming water) and the inflow water at Levante mouth (the inflow is double than the outflow). The results of the computations have also pointed out that 20% of the thermal energy is dissipated in the atmosphere; 70% leaves the bay from Ponente mouth and 10% from Levante mouth.

5 Conclusions

In this paper the methodology used at ENEL-CRIS to evaluate the dilution capability of a submerged multiport discharge has been presented. This multiport diffuser has been designed as a possible solution for substituting the old superficial one in order to fulfil new stringent limits established by the law.

As the new multiport discharge would be located in a bay it was necessary to split the estimation study in two parts. In the first part the near field was studied and the physical phenomena that take place in the region nearest to the discharge were investigated. The dilution power predicted with different methodologies (expert system, physical and mathematical models) was in agreement with the project target.

In the second part the far field was studied. The simulation of the phenomena connected with the dispersion of the heated water plume in the bay was performed and the exchange of water with the open sea through two narrow mouths was investigated as well by means of a 3D mathematical model.

This experience proved that the integrated coupling of different methods can be very useful in the study of problem driven by different scales. The followed approach could be applied in similar cases.

Acknowledgements

The authors thank Mr P. Molinaro and Mr. G. Angelico for their valuable suggestions during the development of the studies.
References


Figure 1: Geometry of the submerged multiport diffuser.
Figure 2: Temperature rise along the centerline of the plume. Comparison between physical model and CORMIX expert system.

Figure 3: 3D mathematical computational domain.
Figure 4: Velocity field at the depth of 13.85 m.

Figure 5: Velocity field at the depth of 0.05 m.

Figure 6: Velocity field in a vertical section 2.5 m from the diffuser line.
Figure 7: Temperature rise along the centerline of the plume. Comparison between physical model and 3D mathematical model.

Figure 8: Computed thermal perturbation on the surface layer.
Figure 9: Computed orthogonal velocity at the mouth.

Figure 10: Computed temperature at the mouth.