Technical University of Denmark

Design of cooling water intake structure CFD modelling case study

Hjelmager Jensen, Jacob; Jensen, Bjarne; Pedersen, Claus; Fredsøe, Jørgen

Publication date:
2014

Citation (APA):
Hjelmager Jensen, J., Jensen, B., Pedersen, C., & Fredsøe, J. (2014). Design of cooling water intake structure CFD modelling case study. Abstract from 34th International Conference on Coastal Engineering, Seoul, Korea, Republic of.

DESIGN OF COOLING WATER INTAKE STRUCTURE CFD MODELLING CASE STUDY

Jacob Hjelmager Jensen, Technical University of Denmark, jahj@mek.dtu.dk
Bjarne Jensen, Technical University of Denmark, bjaje@mek.dtu.dk
Claus Pedersen, DHI SE-Asia, clp@dhigroup.com
Jørgen Fredsøe, Technical University of Denmark, jofr@mek.dtu.dk

INTRODUCTION

Kapar power plant is located on the west coast of Peninsular Malaysia, approximately 50 km to the west of capital city Kuala Lumpur producing an astonishing 2500 MW. The power station lies between the dynamic river mouths of Kapar Besar and Serdang Kecil facing the Malacca Strait - see Figure 1. The power plant operation is aggravated by continuous siltation from the seaward progression of river mouth mudflats which "engulf" power plant coastal structures including pumphouses and intake structures. To alleviate cooling water shortage a comprehensive re-design of the intake structures have been proposed. DHI (SE Asia) was contracted in 2008 to investigate various hydraulic issues related to the new structure using a full 3D CFD (Computational Fluid Dynamics) modeling tool. CFD tools are becoming a serious and feasible alternative to physical modeling when solving problems of such complexity.

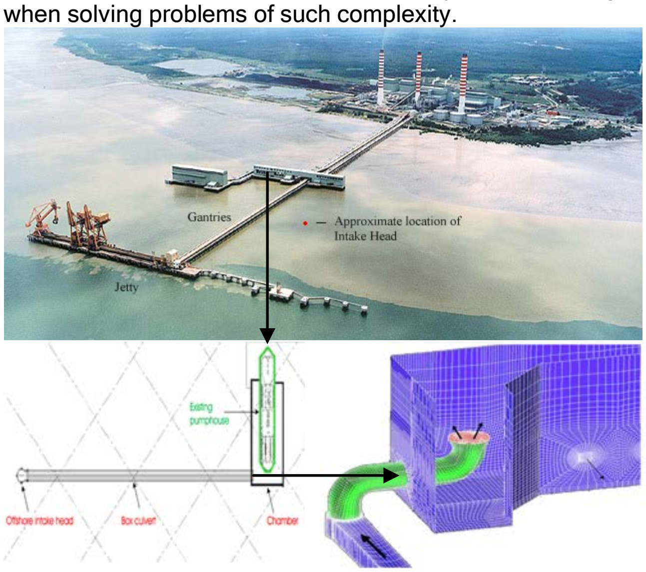

Figure 1 – Top: View of Kapar Power Plant. Bottom Left: Chamber and Pumphouse and Culvert to New Intake. Bottom Right: 3D Model of Chamber and Culvert Outlet

PROJECT DESCRIPTION

A seaward relocation of intake heads has been proposed as a mean to alleviate siltation problems created by the dynamic river mouth morphology. The proposed structure is shown in Figure 1 and consists of *i*) A chamber embracing the existing intake structure and pump house and *ii*) A 240m long box culvert and offshore intake head that connect the chamber with the open sea. Water will be flowing by gravity through the proposed offshore intake head and the 240m long box culvert into the chamber. Water will be pumped from chamber and into the power plant cooling circuit as before. A CFD model has been established to resolve the complex geometries (see Figure 1) and to capture the hydraulics comprising a combination of open and confined flows. The applied CFD model includes free-surface flows, and a 3D sediment transport model is coupled with the flow.

PROBLEM ASSESSMENT

Cavitation: The pumps situated inside the existing pump house have an optimal operating pressure head range. Pumps may not function to their optimal if pressure (water level) drops below this range and in worst case, air entrainment and cavitation may occur. As water flows by gravity alone in the new structure, a water level difference due to head loss between the open sea and the chamber will exist. The CFD model was used both to quantify water level differences due to head loss (marine growth was considered as well as) and as an optimization tool to minimize head losses in individual parts of the intake structure). Simulations shows that internal surface undulations generated by turbulence, wall reflections and water level set-down experienced during upstart of the system (acceleration) are of importance – see Figure 2.

Sedimentation. The waters off Kapar power plant experiences high concentrations of suspended sediments due to its proximity to mudflats, large rivers and its exposure to wave action. The intake water therefore contains sediments and fines. To which degree fine sediments are re-circulating (i.e. bypassing the chamber and sucked further into existing intakes) is determined.

CONCLUSSION

A CFD model for the hydraulic intake structures of the Kapar power plant has been established. The model has been used for hydraulic design optimization (i.e. used as an adaptive tool rather than just a test tool) and to provide various detailed input to structural design, environmental and operational engineers.

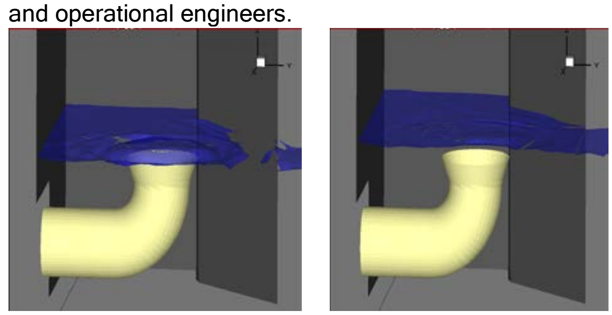

Figure 2 – Water Levels inside Chamber during Pump Upstart and after the System has Attained a State of Equilibrium