DHI SOLUTION

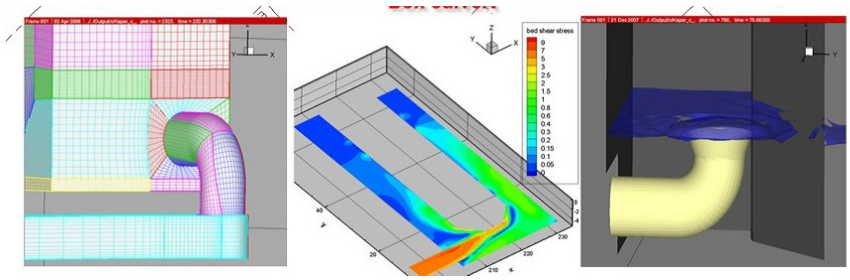
HYDRAULIC STRUCTURES IN POWER PLANTS

Effective CFD tools for modelling flows and loads

Cooling water systems for power, desalination and industrial plants extract and discharge water from the surrounding environment. Components of such a system typically include an intake head and pipeline to receive the cooling water, an intake structure where the water is screened and pumped, and an outfall structure to discharge the warmer water from the plant. Using Computational Fluid Dynamics (CFD) modelling, we can provide assessments of each of these components in order to verify and optimise their hydraulic design.

A COMPREHENSIVE ANALYSIS

Computational Fluid Dynamics (CFD) is utilised to describe the complex flow dynamics in hydraulic structures of a cooling water system. It offers the ability to virtually test a selection of scenarios and design options, while providing a highly detailed description of fluid velocity and pressure fields. Such insights, which are difficult to attain in physical experiments, help in designing systems which are reliable and efficient.



From left to right—Details of a CFD model for an intake pipe and intake structure of a power plant—Assessment of shear stresses on the bottom of the intake structure for the evaluation of sediment re-suspension—Flow dynamics near the pump intake

FORCES AND FLOW ON THE INTAKE HEAD

Intake head structures are designed to help ensure a uniform flow path into the cooling water system as well as limit the introduction of aquatic life. Our solutions provide descriptions of the flow field into and around the intake structures in order to assess if they will meet design requirements. Forces from waves and currents can also be analysed to evaluate the structural integrity for a specific location.

SUMMARY

CLIENT

- Engineering design contractors
- Plant operators

CHALLENGE

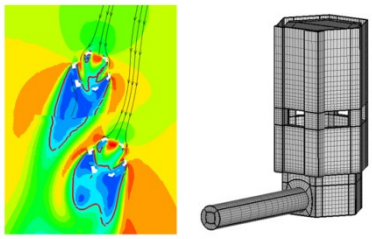
- Design optimisation and verification of intake head, pipeline, intake and outfall structures
- The inefficient and environmentally adverse operation of power, desalination and industrial plants

SOLUTION

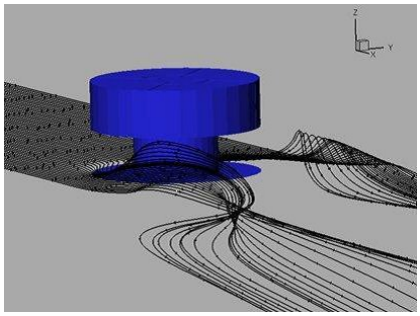
- Computational Fluid Dynamics (CFD) for virtual testing of design options in various test scenarios
- Detailed description of flow velocity, pressure fields and loads

VALUE

- Optimised design, which is energy efficient and sustainable
- Ensured structural integrity to wave and current forces
- Quantification of the environmental impact of the outfall plume
- Confidence in the final design in extreme and emergency scenarios



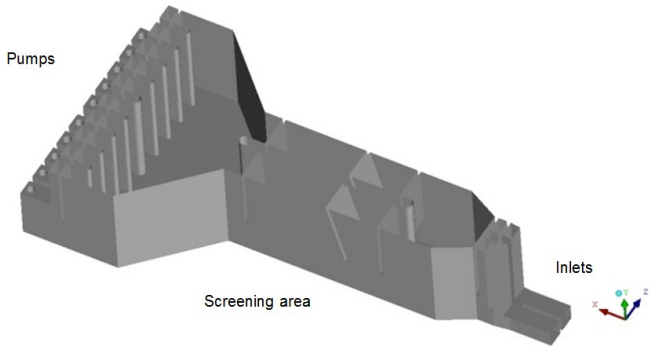
Flow field contours for two intake heads in close proximity (left) and 3D CFD model mesh for one (right) structure



Streamlines around a velocity cap intake

ASSESSMENTS OF THE INTAKE STRUCTURE

An intake structure receives cooling water from the surrounding coast/waterways and removes debris and contaminants via screens. Water is then pumped to the plant via a series of pumps.

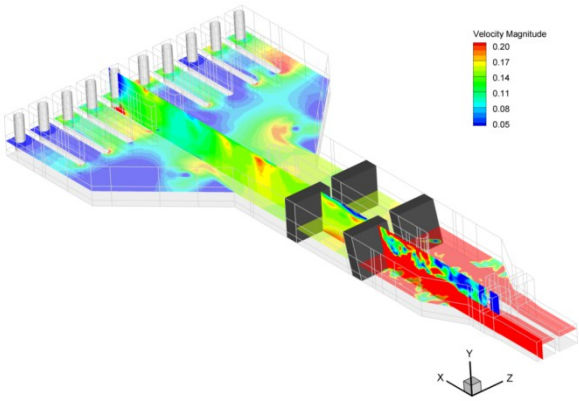


Typical geometry for an intake structure with inlets receiving sea water, a screening area and a common bay leading to an array of pumps where the water is extracted

Understanding the 3D flow dynamics within an intake structure is essential in order to avoid problematic circulation and cavitation. Such flow characteristics result in the inefficient use of the intake pumps, resulting in faster wear. We offer services which provide an overview of the flow field within an intake structure for a variety of test scenarios. These include:

- clogging of bar and drum screens
- malfunctioning/maintenance of certain intake pumps
- closure of stop logs
- various tidal levels
- fouling in the intake pipes

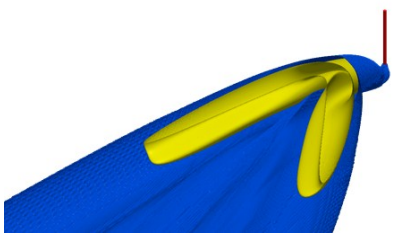
Various design solutions can be tested, such as different configurations of flow guides, baffles and anti-vortex devices. Depending on the requirements of the project, our CFD solutions can vary in complexity from transient free-surface calculations to steady state single-phased simulations.



Modelled intake structure with vertical and horizontal velocity magnitude contours for a test scenario. Dark shaded elements represent the screens used to filter the inflow

DISPERSION OF OUTFALL PLUMES

Understanding the complexity of the buoyant cooling water from the outfall structure is essential in mitigating potentially adverse environmental impacts. The near-field mixing processes of strongly buoyant sub-surface plumes are complex and are poorly described in most commercially available software. Therefore, a more comprehensive description of the physics provided by CFD is required for a more robust analysis.



Model contours showing buoyant plume from an offshore outfall

SOLUTIONS – ONE OF MANY

With decades of experience assessing the hydraulic performance of cooling water systems, we provide state-of-the-art 3D CFD modelling. We also offer:

- physical modelling experiments
- design reviews
- scour assessments
- MetOcean studies
- larger scale plume dispersion studies

Our extensive experience, combined with our unique ability to combine services enables us to provide valuable and pragmatic engineering solutions.

Contact: Henrik Kofoed-Hansen - hkh@dhigroup.com or Xerxes Mandviwalla - xem@dhigroup.com
For more information visit: www.dhigroup.com