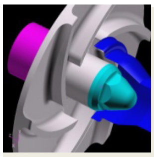
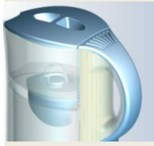


▶ **ANALYSIS & DESIGN**
ASSESSMENT OF PRODUCT PERFORMANCE & DESIGN DEVELOPMENT



▶ **RESEARCH & DEVELOPMENT**
FEASIBILITY OF NEW CONCEPTS & UNDERSTANDING CORE PHYSICS

▶ **PRODUCT DEVELOPMENT**
WORKING AS PART OF YOUR MULTIDISCIPLINARY TEAM - FROM CONCEPT TO MARKET

- Environmental Analysis
- Dispersive Flows
- Performance Assessment

Intelligent

Fluid Solutions

SPECIALISTS IN THE APPLICATION OF FLUIDS ENGINEERING & CFD FOR ENGINEERING & DESIGN

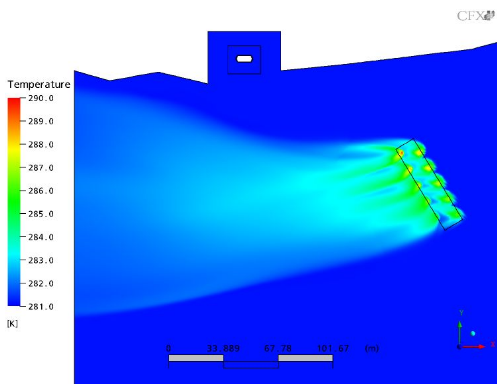
Assessment of Pollutant Dispersal from a Power Station

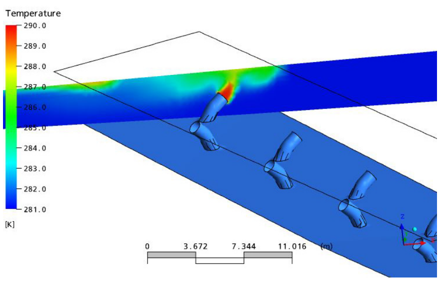
Power stations often require large volumes of coolant water and hence are sited close to bodies of water such as rivers and lakes. However, large natural bodies of water often contain complex marine ecosystems that are sensitive to the local environmental conditions.

Although significant effort is made to limit the discharge of artificial pollutants, thermal pollution ie the discharge of higher temperature water, can also cause harm. The increase in water temperature can lead to migration of species, creates a potential for new incoming species and can radically alter the balance of the existing marine ecosystem.

The aim of the work programme was to use CFD techniques to assess the performance of a power station outflow design that aimed to limit thermal environmental impact. The power station was situated adjacent to a tidal bay.

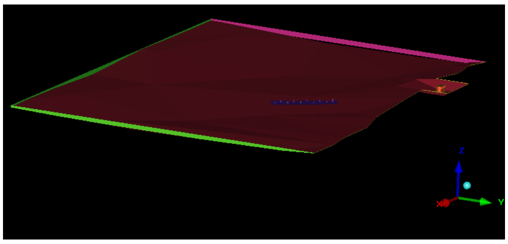
The objective of the project was to determine the geometrical extent of the region in which the water temperature was elevated by more than 3°C during a "worst case" outflow condition. This level of temperature increase is commonly used as an indicator for the potential for environmental damage.





Project Summary

Power stations take in large volumes of coolant water from nearby water reservoirs and discharge water at elevated temperatures. This thermal pollution can cause significant damage to sensitive marine ecosystems. This work programme aimed to use CFD techniques to assess the performance of a power station outflow manifold and configuration at a worst case condition. The local geographical relief and the geometry of the outflow manifolds were used to construct a CFD model of the system. The results from the analysis indicated that the excessive temperature increase is restricted to a region within approximately 70m of the outflow. In addition, the elevated temperature is concentrated at the tidal bay surface with the bay bed remaining largely unaffected.



Getting the Most From Engineering Analysis

Using advanced engineering analysis to predict device performance and hence gain confidence and reduce technical risk

INTELLIGENT ENGINEERING

Experience & Technology

Geographical data of the tidal bay adjacent to the power station and CAD geometry of the outflow and inflow manifolds were used to develop a CFD model of the power station discharge system. The investigation was limited to a near-field analysis encompassing a region approximately 200m upstream and downstream of the outflow.

The ANSYS suite of CFD products was used throughout this project, ANSYS-ICEM CFD software was used to develop the CFD models and ANSYS-CFX was used to obtain the flow solutions.

The domain of interest was large from a CFD viewpoint and included a large range of length scales ie smaller scales of the order of the outflow discharge pipe diameter as well as larger scales of the order of the main dimensions of the bay. Thus a hybrid meshing approach was used in which the local field was meshed using tetrahedral elements whilst the far-field used efficient hexahedral elements.

The worst case condition was assumed to be during the period 7 ½ minutes before the Low Water point and the 7 ½ minutes after the Low Water point. A full time-dependent analysis was performed that included the free surface of the tidal bay.

The outflow discharge temperature was assumed to be 8.6°C higher than the bay ambient temperature.

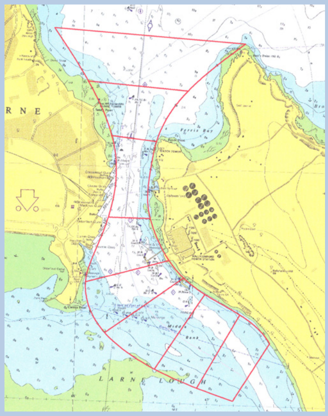
An initial steady-state analysis was performed in order to develop a reasonable set of initial conditions for the time-dependent analysis.

The analysis results suggested temperature increases greater than 3°C were restricted to a region within 70m of the outflow manifold. In addition the region of increased temperature was concentrated close to the surface of the tidal bay with a large vertical temperature gradient. The bay bed was largely unaffected.

The results indicated that although the discharge flow did result in a slight increase in water temperature at the inlet manifold, the increase was not significant. Thus the level of flow recirculation was low.

The analysis indicated that region of potential environmental impact was relatively small and concentrated to a small area in the neighbourhood of the outlet.

The advent of inexpensive computer power and commercially available analysis software has made possible the full virtual modelling of product performance. However, the technology is still developing in terms of accuracy and reliability. Thus CFD is not a substitute for traditional engineering methods, but rather compliments the existing processes



Intelligent Fluid Solutions

Experts in Fluids Engineering

Contact Us to Find Out How We Can Help You

UK Branch:
Dr. Jim Shaikh
jim.shaikh@intelligentfluidsolutions.co.uk
+44 (0) 208 859 9633

SA Branch:
David Hartwanger
david.hartwanger@intelligentfluidsolutions.co.uk
+27 (0) 43 748 5518