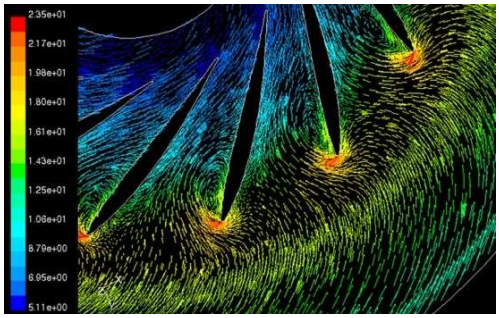
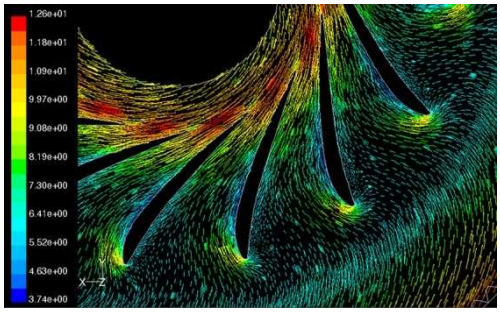
CFD

COMPUTATIONAL FLUID DYNAMICS

e3k™ engineering3000 new product design & development





Since 1995, e3k have been developing and expanding their expertise in Computational Fluid Dynamics (CFD) analysis. e3k is well positioned to analyse a wide range of fluid flow regimes.

Computational Fluid Dynamics (CFD) has grown from an academia-based research method requiring substantial computing resources, into a highly adaptive and accessible computer aided engineering and design tool. The development of faster, more powerful computers has made it possible to run CFD models effectively on a PC or laptop computer.

CFD can be fully integrated into the design process – the same 3D CAD models built for prototype construction and Finite Element Analysis (FEA) can be imported into the CFD software for accurate computer analysis at an early stage of development.

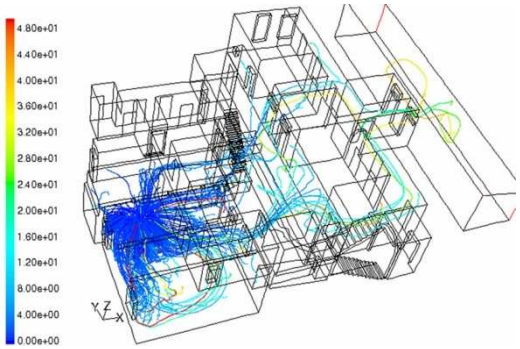
Essentially CFD can provide an invaluable phase between pure mathematical analysis and experimental testing. By creating 2D or 3D images and data for analysis, CFD can enable a concept to be developed through to prototype stage with speed and efficiency, minimising the time and expense required for experimental testing. CFD can also be effectively used to simulate flows in situations where experimental testing would not be possible – e.g. after an event or when a prototype would be overly large, complex or expensive.

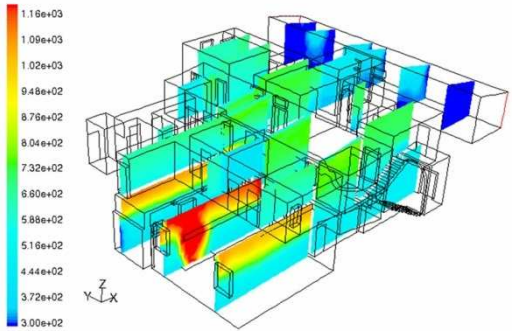
The applications for CFD are endless – from renewable energy devices and industrial and mining machines to race car aerodynamics. Any machine or object experiencing fluid flows can be modelled. e3k utilises the software ANSYS CFX to produce accurate models of fluid behaviour for a vast range of flow problems. The software allows engineers to study diverse flow situations including free surface flows, acoustics, combustion behaviour, fluid mixing and turbo machinery.

Examples of CFD studies undertaken include:

Modelling of a building fire

A CFD study was undertaken on a building fire within an accommodation complex. Results from the study were integrated into the police investigation enabling determination of cause and qualification of the overall building hazard. The same simulation technology has since been successfully validated by e3k against experimental data in a full-scale mock building burn.



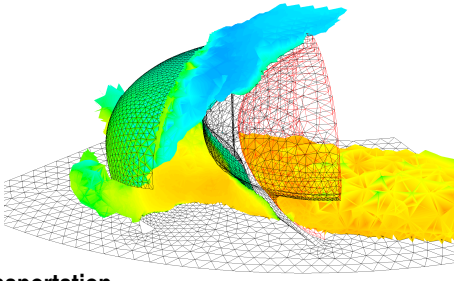


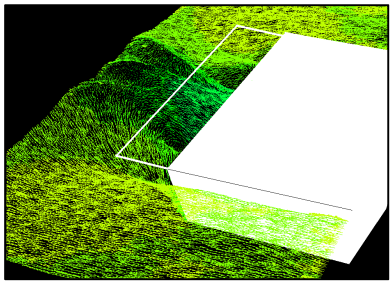
Ore skip passing through a mine shaft

An investigation of the lateral motion of an ore skip within a vertical mine shaft due to aerodynamic and other effects was carried out using CFD. This analysis looked at the forces on the sides of the skip as it travelled towards the surface. The model included details of the openings in the top of the skip and the air velocity down the shaft.

Dynamic modelling of a hydro Pelton wheel bucket interaction and nozzle optimisation

Dynamic analyses were carried out to determine the interactions between the water jet and the Pelton buckets. During each CFD analysis, the time-varying interaction between the jet and bucket was analysed by modelling the bucket rotating past the jet. The outgoing water velocity from the bucket CFD analysis was used to estimate the path and envelope of the water sheets leaving the bucket.





Livestock transportation

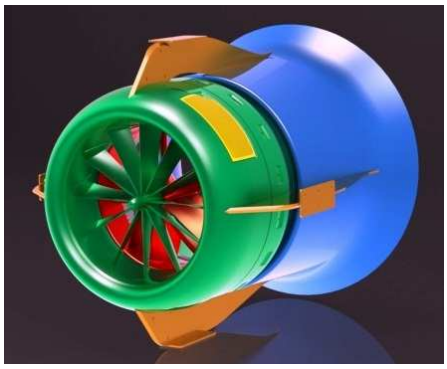
A study was conducted in order to optimise ventilation conditions on board a vessel designed to carry livestock. The simulation included such things as heat emanating from the animals, CO_2 production through their breathing and spacing of the animals along the length of the vessel.

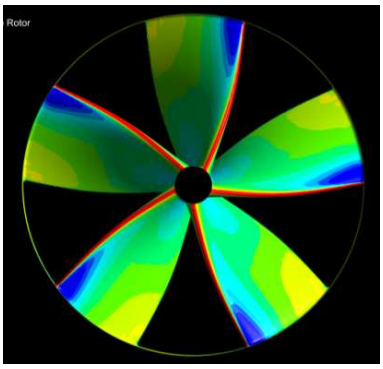
Modelling of water flow around a marine vessel using multiphase solvers

A CFD study was undertaken on a marine incident involving a recreational motor yacht. Results from the analysis describe the flow around the vessel's hull. This information was useful in determining the actions of the driver of the vessel. The flow behaviour e3k modelled was validated by police in experiments conducted with the vessel, providing correlation with the CFD data.

Determination of power output from a novel tidal energy generator

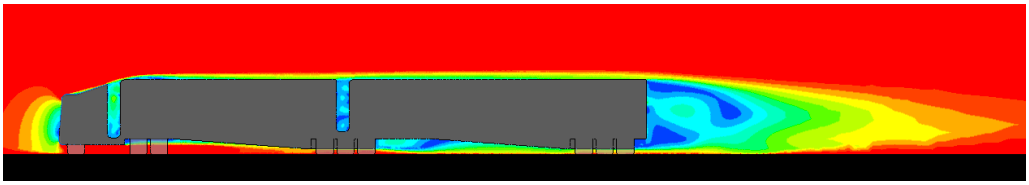
e3k was engaged to analyse and design a novel, 2kW tidal energy device. This project collaboration won a 2012 Engineers Australia National Engineering Excellence Award and was the overall winner of the 2012 Engineers Australia Engineering Excellence Awards, Newcastle Division. e3k used a combination of 2D and 3D analysis throughout this project, designing all components that interact with the tidal stream. Our analysis consisted of internal velocity augmentation and determination of output torque from the axial rotor blades.





Modelling air turbulence around a B-double prime mover

e3k have used CFD to analyse the air profile around a B-Double prime mover and trailer. Because air drag increases the required energy to power this type of vehicle by up to 50%, its reduction can mean large improvements in fuel efficiency. Our engineers are creating novel solutions to reduce this drag for road transport as Australia moves toward a low carbon future.



HVAC study of a railway train subway

e3k was engaged to undertake a computational fluid dynamics study of a new subway in Delhi, India. The modelling was used to optimise tunnel ventilation to ensure operational air-conditioning in the event of a train failure mid-tunnel. The simulation included heat transfer to the ground and through tunnel walls.

Optimisation of the outlet temperature of a commercial dishwasher boiler

To reduce temperature fluctuations from the spray arms of a commercial dishwasher, e3k used CFD to optimise the dynamic heat dispersion within the boiler.