# Youtube channel
click [here](#) to watch

CFD simulation of the NDS-60 OSV

Full scale CFD simulations are carried out in order to predict the hull performance for several speeds and drafts. The friction and pressure resistance of the hull is calculated and streamlines around the hull provided to our client Naval Dynamics (www.navaldynamics.com). The obtained data and streamlines can be used to optimize the hull shape for the purpose of the vessel.
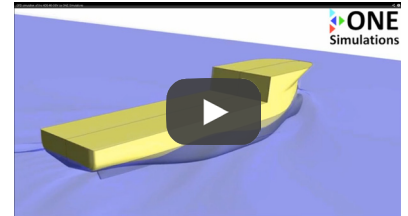

Pitch and Heave Motion Response

ORCA Offshore carried out lift dynamic motion analysis using the software tool MOSES from Ultramarine. One major challenge for this project was the limited bottom clearance of only 0.6 m. Furthermore, the seabed is formed from very soft mud which will have an unknown effect on the behaviour of the vessel. The standard diffraction calculation as used in MOSES cannot take this into account.
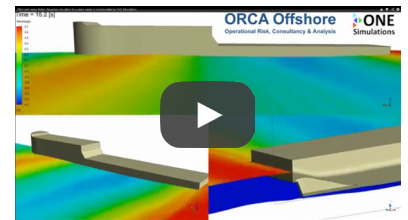
To verify the effect of the small keel clearance and the soft mud layer, several CFD (Computational Fluid Dynamics) simulations have been carried out by ONE Simulations. With a CFD simulation it is possible to calculate the motions of a rigid body in a dynamic environment composed of air, water and mud. The CFD simulations have been executed for several regular wave periods and calculate the Heave and Pitch Motion Response of the vessel, as well as the pressure on the skeg. A CFD simulation has also been used to calculate the Pitch Free Decay Period of the vessel in this specific environment.


Lowering gravity base

One Simulations performed a simulation using Computational Fluid Dynamics for the lowering process of the gravety base. The simulation shows that the free falling base de-accellerates starting at about 10 m from the seabed before a smooth landing occurs.

More information: http://www.monobasewind.com/Engineering
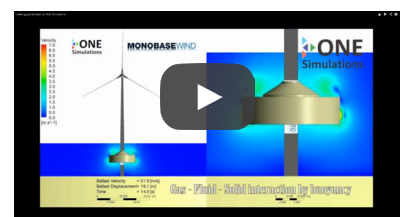
Smoke exhaust system car park

This movie shows a sequence of results with 1 minute interval. The ventilation system is designed for smoke removal, furthermore the smoke spread is limited by fire doors.

ONE Simulations

▶ Fluid dynamics
▶ Fire safety
▶ Environment
◀ Energy
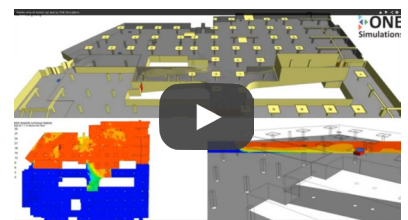
Optimization buffer tank

Optimization study for unloading and loading a buffer tank for the cooling system of a datacenter by means of CFD in collaboration with Cofely. The function of the tank is to guarantee the availability of chilled water for the equipment for a minimum time interval. So the servers will not be exposed to higher temperatures during a power failure. The performance of the buffer tank is depending on the temperature stratification. In collaboration with Cofely, several designs are verified in order to find the most suitable solution.
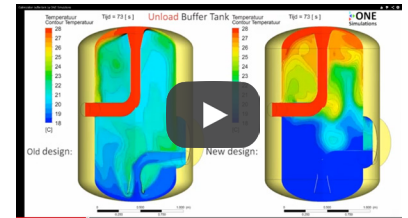

Pump impeller optimization

This study has been performed in collaboration with Holland Special Pumps in order to optimize the impeller. By the study the performance of the pump is analysed based on pressure losses, cavitation, flow patterns etc.
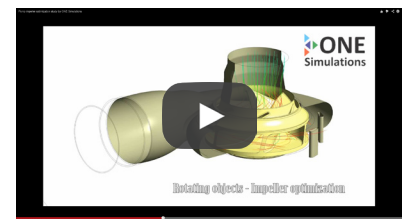

Fire simulation hospital atrium

Simulation of a fire in a top level room to determine the thermal load on the steel structure. The atrium roof is constructed of a glass structure supported by steel beams. The complete study included flash over analyses in the room, vertical fire spread by breaking windows and evacuation possiblities of the hospital rooms.
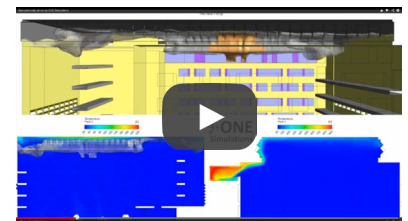
Evacuation of a road tunnel

The evacuation process is showed for the Al Salam road tunnel including the obstruction of cars, trucks and busses.
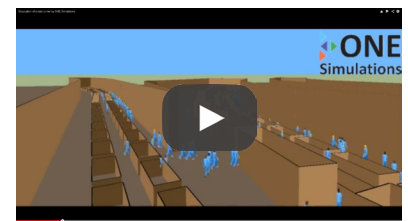
Evacuation large entertainment centre

This study shows the evacuation of a large entertainment centre (Heineken Music Hall). The simulation considers 6.364 persons divided over 4 building floors. In this movie the ground floor is shown. The model includes stairs, doors and stands. With use of simulations the effectiveness of the evacuation protocol has been analyzed for a large group of visitors.