International Marine and Dredging Consultants (IMDC) is an engineering and consultancy company specialised in a vast range of water related projects. Our highly-qualified staff offers advice based on recent research results of leading universities and research institutes and hands-on experience acquired throughout the years.

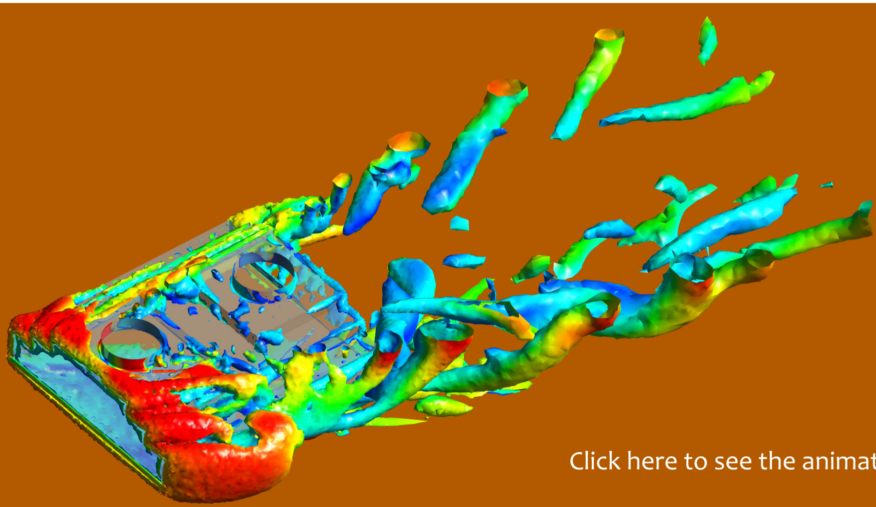
One of IMDC's core activities is presented in this booklet:
Computational Fluid Dynamics for Marine and Coastal applications.

More information can be found on our website: **www.imdc.be**

Computational Fluid Dynamics for Marine and Coastal applications

Computational Fluid Dynamics (CFD) models provide insight in **complex flow patterns** of internal and external flow phenomena. The flow can consist of a continuous phase (e.g. water) and one or more dispersed phases such as sediment particles, tracers and air bubbles.

Highly **detailed geometry** can be implemented in the computational domain. Several numerical domains sliding or rotating relatively to each other can be implemented, e.g. for rotating tidal energy turbines or other moving objects.



Click here to see the animation.

Applications and Clients

The model results are applied in the fields of:

- **Offshore renewables** (tidal, wave, marine scour protection)
- **Dredging & reclamation** (plume dispersion, dredge pumps, pipe flow)
- **Ports & locks** (scour protection, sedimentation, density currents)
- **Intakes, outfalls, recirculation** (hydraulic design optimisation, diffusors)
- **Hydropower** (spillway, penstock, expansion chamber, surge tanks)
- **Ship hydrodynamics**

Processes & Software

Depending on the specific case, equations can be solved for:

- 3D complex flows of a **continuous phase** (water, air), stationary or transient
- **Turbulent motions** (Large-Eddy Simulation technique)
- Motions of **dispersed phases** carried by the continuous phase (sediment, rock, bubbles)
- Flow through **porous media** (e.g. filter layers, sand key)
- Interactions of the above

IMDC applies Ansys Fluent® software for fast delivery and design optimisations, using the powerful parameterisation tools provided. For in-depth studies of customised applications, the open-source code OpenFOAM is used.

Products

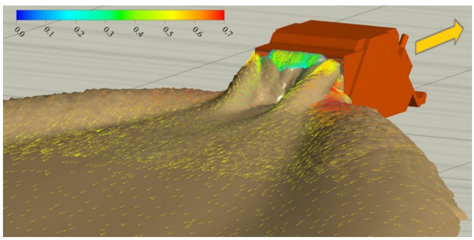
In general, CFD simulations at IMDC can be applied to execute, among others, following tasks:

- **Design optimisation** through for example streamlining;
- Determination of **design conditions**;
- Problem-solving through **flow analysis** (e.g. reduced efficiency, vibrations, vortices …);
- **Multi-phase** problems, e.g. water + air bubbles, granular flow, sediment flow.

Examples of Models

Deep sea mining carriage design and sediment suspension

Mining and dredging activities are increasingly moving to deeper seafloors as compared to the past. Due to the greater water depth, carriages are being designed to dredge or mine at the seabed. In order to design such carriages, and to assess environmental impact, CFD simulations can provide valuable information.
Below, an example is shown of a carriage designed to keep rejected sediments as closely as possible to the seabed. In this way, environmental impacts of suspended sediments can be minimised. Further, the shape of the carriage can be optimised and streamlined using CFD in order to reduce power consumption.

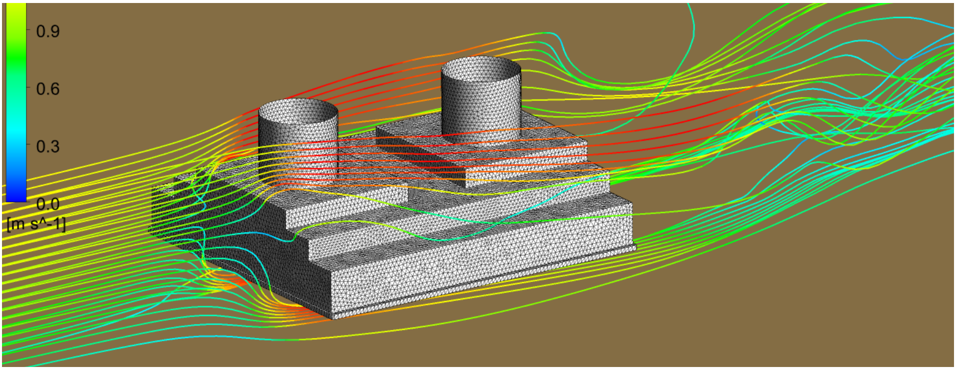


Streamlines of flow around a deep sea mining collector, and visualisation of sediment suspension in the wake of the carriage (in orange).

Line forces during floating caisson installation

Installation of offshore infrastructure is often executed by ballasting floating elements such as caissons. During installation, the large-volume elements are under the influence of sea currents, waves and wind. Vortex-Induced Motions (VIM) might be an additional challenge. Mooring lines need to be designed in order to safely keep the elements in place.

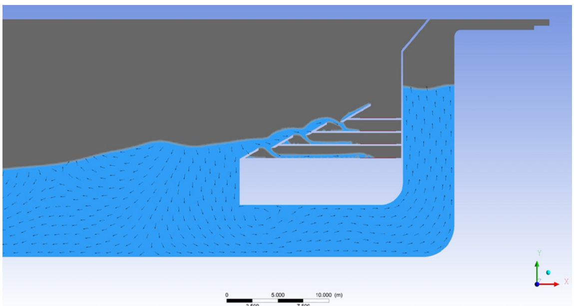
During the design of the mooring lines, CFD simulations are of great value to understand the forces and moments acting on the element. Coupled with a static or dynamic mooring analysis, mooring lines can be designed directly based on the simulations. Below an example is shown of a 3D flow simulation around a large bridge foundation caisson during installation.



Streamlines of strong marine currents around a floating caisson, basis for calculation of forces and moments acting on the element.

Efficiency of wave energy convertors (WEC)

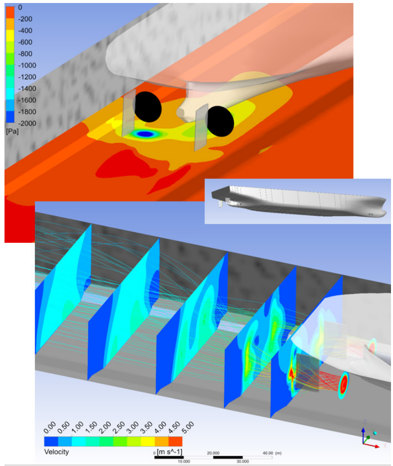
Future developments in the blue energy sector can be executed at lower cost when applying CFD as a virtual laboratory. CFD can reduce the number of expensive physical model tests during for example the design process of WEC's. In the example given, a CFD simulation is shown of waves arriving at a WEC integrated in a breakwater. The efficiency is determined by simulating the internal flows and discharges moving through turbines.



CFD simulation for efficiency optimisation of a hybrid Wave Energy Convertor (WEC) integrated in a breakwater.

Propeller jets

Design conditions for the scour protection at berthing areas of a new port terminal have been computed. Near-bed flow velocity and dynamic pressure are determined to find the requirements for the scour protection design. The full geometry of a large container vessel was implemented in the model domain, three-dimensional flows have been simulated including the effect of twin propellers, rudders and bow thrusters.



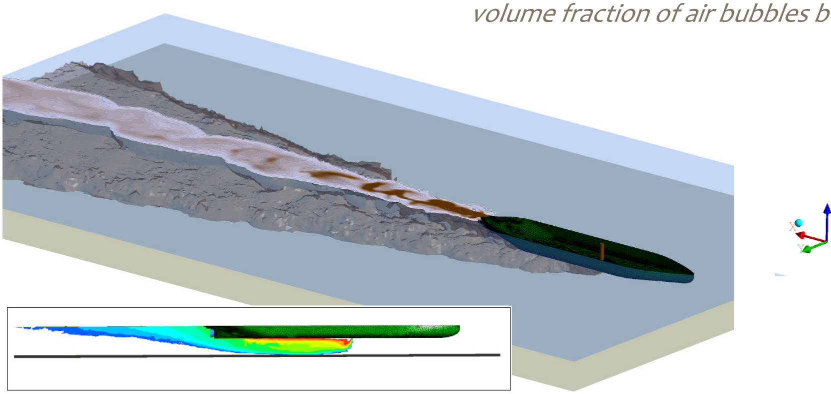
Fow field induced by the main propellers of a large container carrier starting to manoeuvre away from a quay wall. Dynamic pressure distribution over berthing area (top) and flow velocity pattern along vertical planes and streamlines (bottom).

Sediment plumes dredging works

During dredging with Trailing Suction Hopper Dredgers, excess water is removed from the hopper through an overflow shaft. The sediments contained by the released waters form a sediment plume with potential environmental effects. Plume prediction and adaptation of dredging works planning (if needed) can be achieved through accurate sediment plume dispersion modelling.

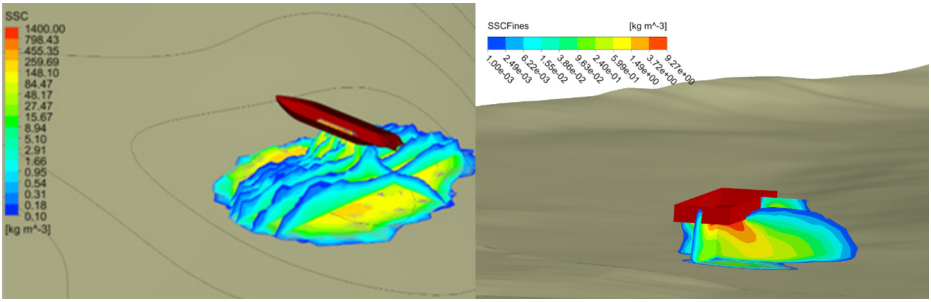
The near-field dispersion of these plumes has been modelled in CFD by IMDC. The effects of sailing speed, density differences, air bubbles and swirling propeller jets on the plume dispersion have been included in the simulations. A highly detailed distribution of sediments near the vessel is obtained.

Multi-phase CFD simulation of the water-sediment-air mixture released by a dredging vessel's overflow shaft. Top: surface sediment plume and subsurface sediment density current. Inset: distribution of the volume fraction of air bubbles below the ship.



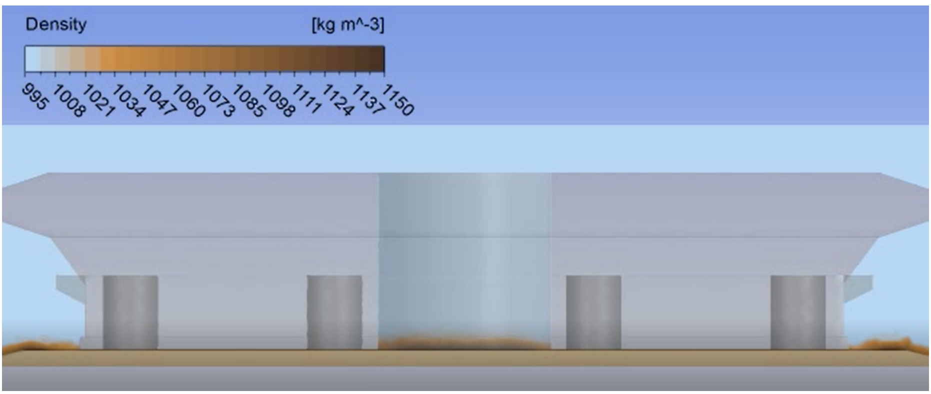
Dredge spoil disposal

During disposal of dredged sand or clay, it is often relevant how the material spreads on the seabed or whether fine sediments wash out to form turbid plumes. To study such phenomena in detail, IMDC developed models that can simulate disposal of sand, clay balls, fine sediments or mixtures of these.
The models can determine impact speed, seafloor spreading and current-induced losses.



Installation of structures on porous beds and mud layers

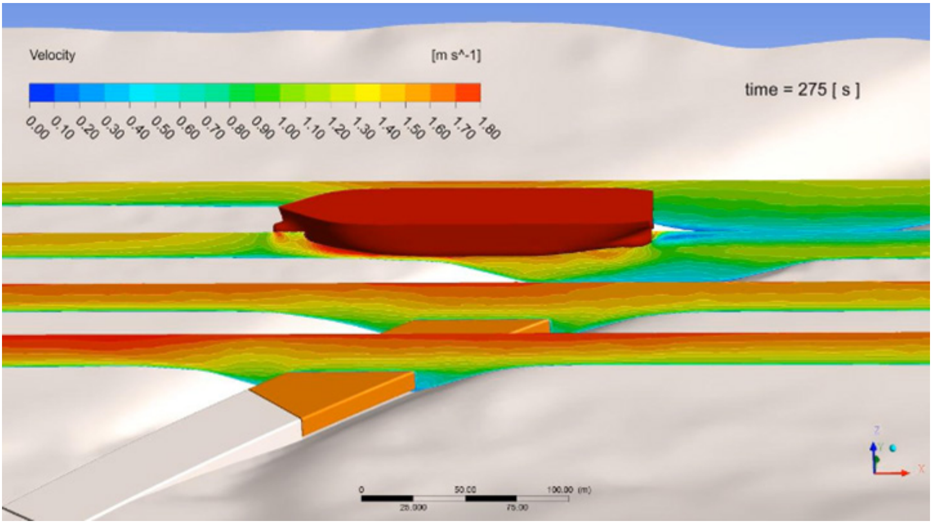
During marine construction works involving installation of structures over porous beds, special care must be given to the existence of fluid mud between the bed and the structure. Fluid mud can present severe limitations especially with respect to delays in installation process and proper placement of those structures. The thickness of the mud layer and the rheological characteristics (density, plasticity, viscosity etc.) play a crucial role for the installation process.
To study this process, IMDC has developed a numerical model that simulates the last stage of installation of structures over a porous bed, taking into account the presence of the mud layer. The model can determine the hydrodynamic forces exerted by the flow conditions on the structure along with the motion of the mud layer during the installation procedure.



View of the mud layer density after installation of an intake structure (grey) onto a porous bed (light brown).

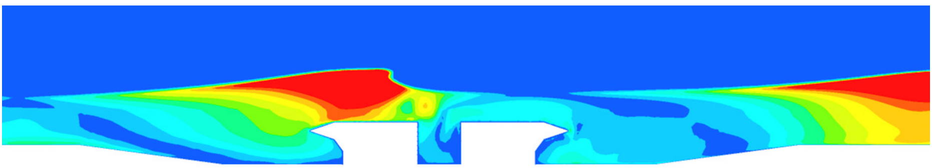
Ship passing over tunnel elements

The passing of a vessel over a tunnel element can exert significant forces on a tunnel element during installation. In addition, it can temporally modify the flow field at a significant extend.
For that purpose, IMDC has developed a two-phase (air-water) model that simulates the motion of a vessel over a tunnel element and is capable of calculating the pressure distribution on the tunnel element and the temporal variation of flow induced forces and moments on the tunnel element.



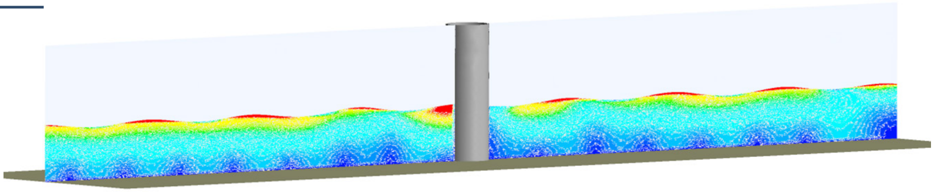
Extreme wave loads over submerged structures

Design wave loads are often difficult to asses in the presence of complex structures since standard formulas published in guidelines are not valid.
Wave forces on a structure, or stresses acting on scour protection around the structure can be assessed in high degree of detail using CFD.

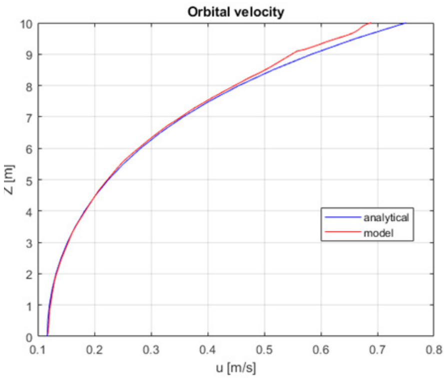


Wave loading on marine structures

IMDC has prepared a three dimensional CFD template for estimating wave-induced forces on offshore structures such as cables, pipelines, wind turbine, bridge foundations, caissons and tunnel elements.



Velocity field along the middle of the numerical domain.



Comparison between numerical model and analytical solution for orbital velocity.

In addition, a two dimensional template has been developed for estimating wave-induced forces and overtopping at coastal walls and breakwaters. The model results have been compared with analytical solutions from literature regarding pressure distributions on vertical walls.

Comparison between numerical model and analytical solutions for dynamic pressure distribution along a vertical wall.