IMDC

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.

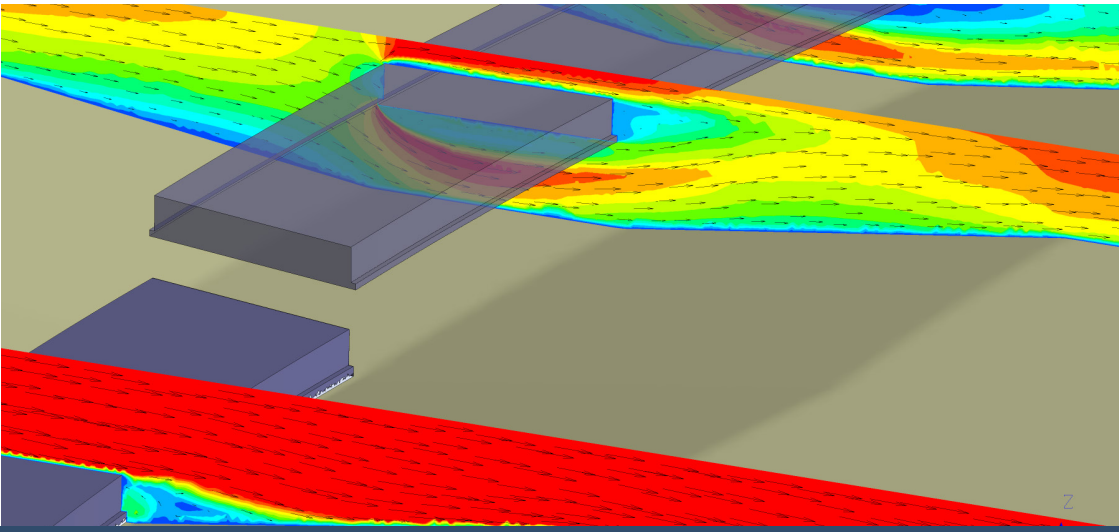
One of IMDC's core activities is presented in this booklet:
Computational Fluid Dynamics for Hydraulic Engineering .

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

Computational Fluid Dynamics for Hydraulic Engineering

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.



Applications and Clients

The model results are applied in the fields of:

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

Processes & Software

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

- 3D complex flows of a c**ontinuous 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.

www.imdc.be

Products

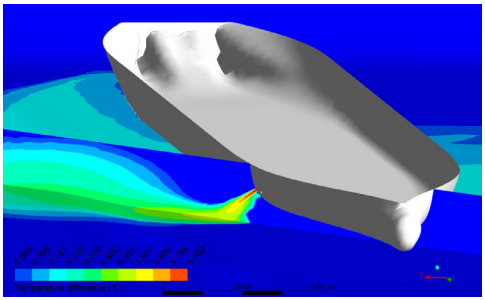
In general, CFD simulations at IMDC can be applied to execute, among others, following tasks:

- **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

Recirculation FSRU

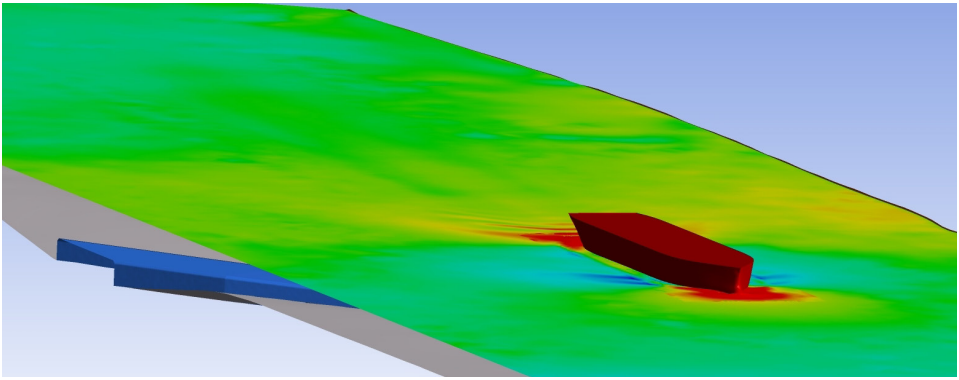
Floating Storage and Regasification Units (FSRU), are stationary vessels used to regasify liquefied gas, for example after offloading from gas tankers. In this process, heat is extracted from the sea water. Sea water is extracted through an intake, and after flowing through the regasification system it is released at a lower temperature through an outfall. Recirculation of outfall water, i.e. released water flowing back to the suction pipe of the intake, can affect the efficiency of the regasification system. CFD simulations have been executed to optimise the system of multiple intakes and outfalls.



CFD simulation of a thermal plume resulting from FSRU outfall.

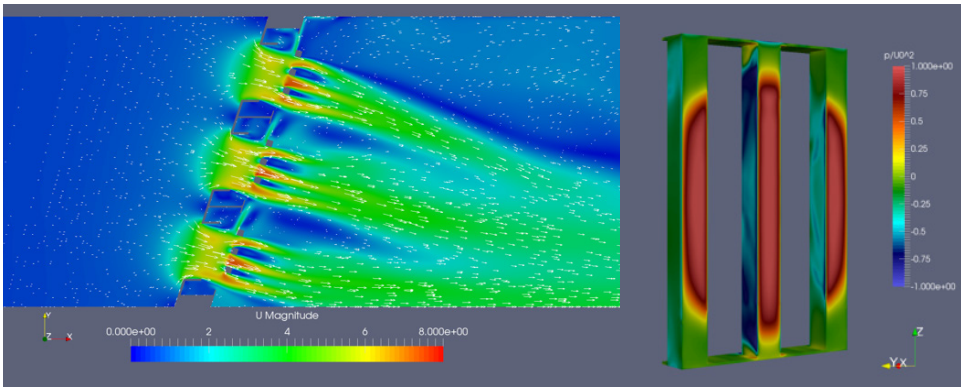
Passing ship effect

During marine construction works, or for moored vessels, the loads mooring loads induced by passing vessel might be difficult to quantify. Often expensive physical model tests are seen as the only option. IMDC provides numerical tests to simulate the passing ship effect. A moving vessel is simulated including the bow wave and induced velocity field. As a result, the pressure field on a submerged body can be calculated and hence the force and moment exerted.



Lock gate exchange flows

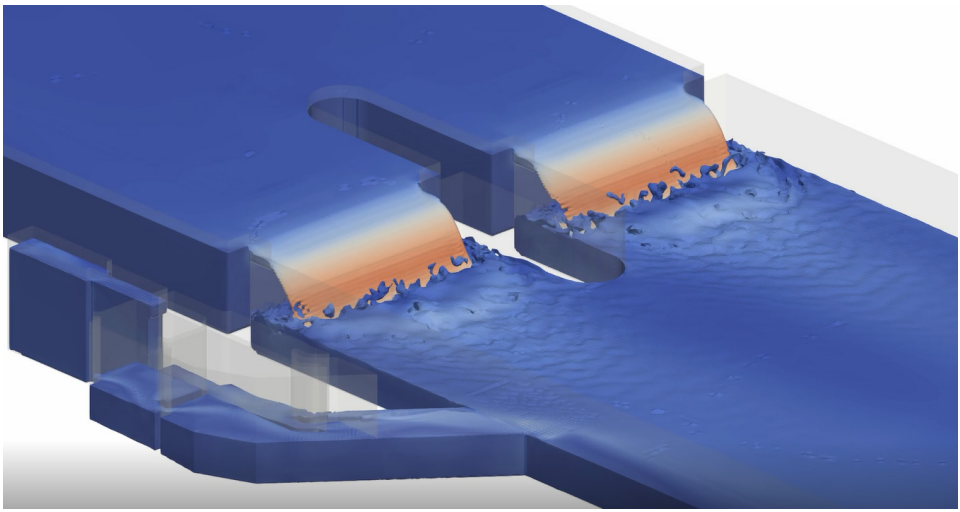
CFD simulations have been performed to study the high-velocity jets occurring during filling and emptying of lock chambers through gates in the lock doors. Several geometrical alternatives are investigated to optimise the breaking of jets in order to minimise the jet impact on vessels moored in the chamber during locking.



CFD simulation of the flow through lock gate spills fitted with jet-breaking bars. Detail of dynamic pressure distribution on the breaking bars (right) and large-scale flow pattern through three gates fitted with three breaking bars each (left).

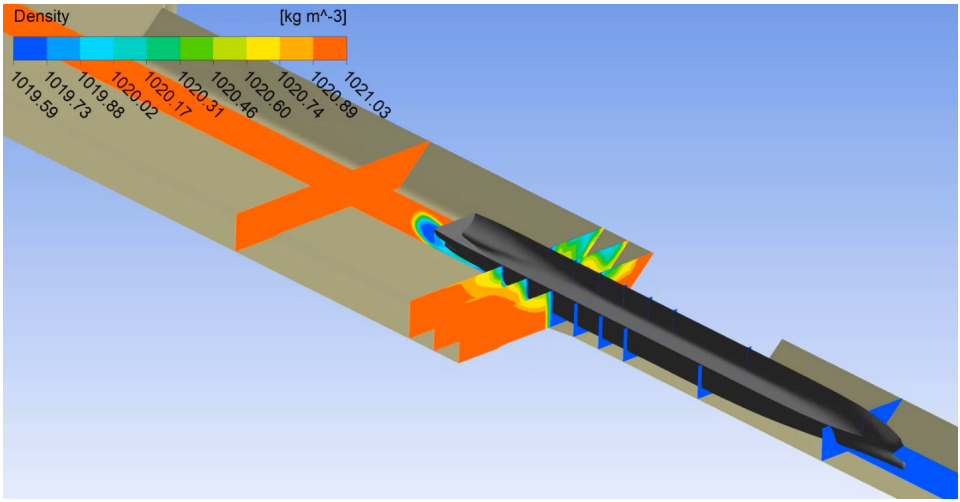
Fish ladder design

To aid the optimisation of conditions for optimal fish ladder functioning, CFD models can be applied. For example it can be verified how fish can be attracted and how flow velocity above sprint speed can can be avoided.



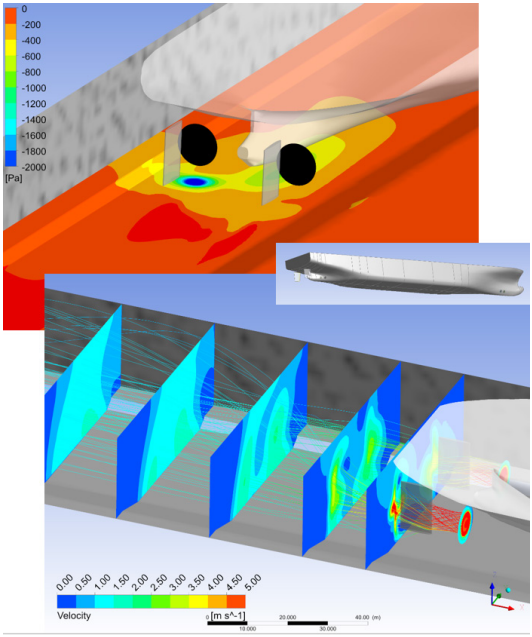
Lock exchange flows

Our CFD models can also be applied to study the salt intrusion at locks and sluice gates. A vessel entering the lock can be simulated. The CFD models are used to estimate the salt exchange or to design salt barriers, but also to estimate piston effects and water surface gradient during levelling.

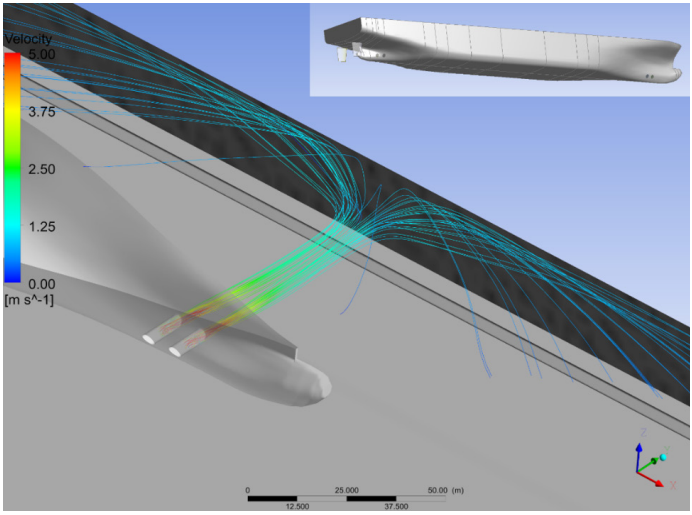


Propeller jets

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.



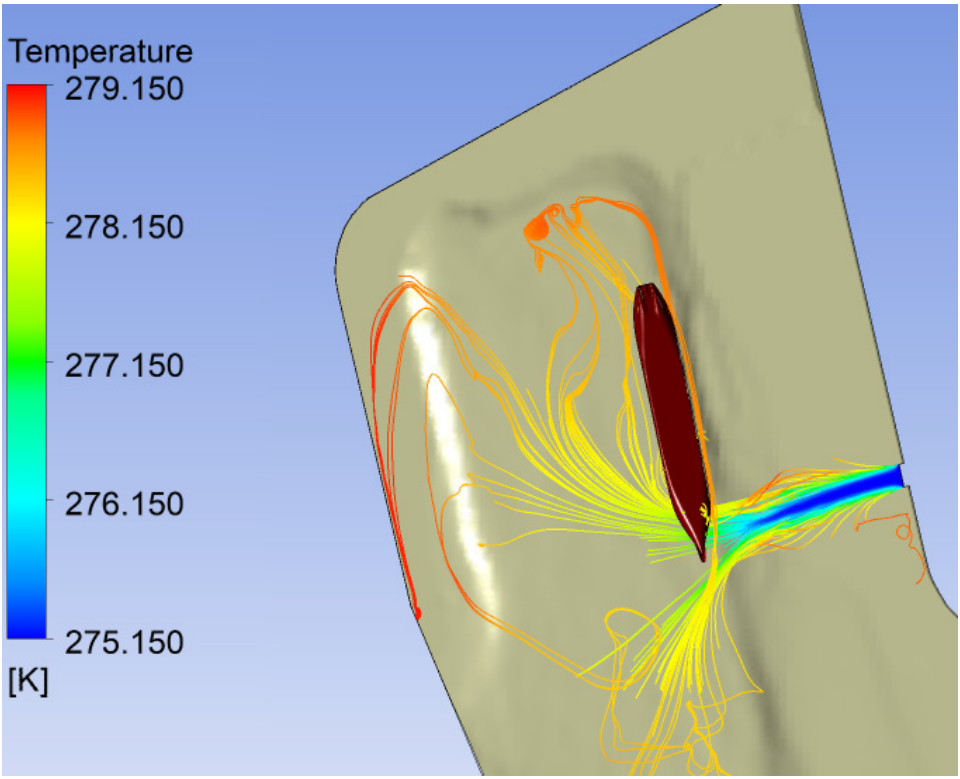
Flow field induced by the main propellers of a large container carrier starting to manoeuvre away from a quay wall. Dynamic pressure distribution over berthing area (top) and flow velocity pattern along vertical planes and streamlines (bottom).



Streamlines of the flow velocity pattern formed by two bow thrusters blowing at the quay wall.

Thermal plumes

Near power plants or LNG terminals, outfalls often release a thermal load. The assessment of dispersion of the thermal plumes is usually required for environmental impact or recirculation studies. Our CFD models can simulate the temperature field in combination with for example tidal flows in a detailed way.



MORE INFO