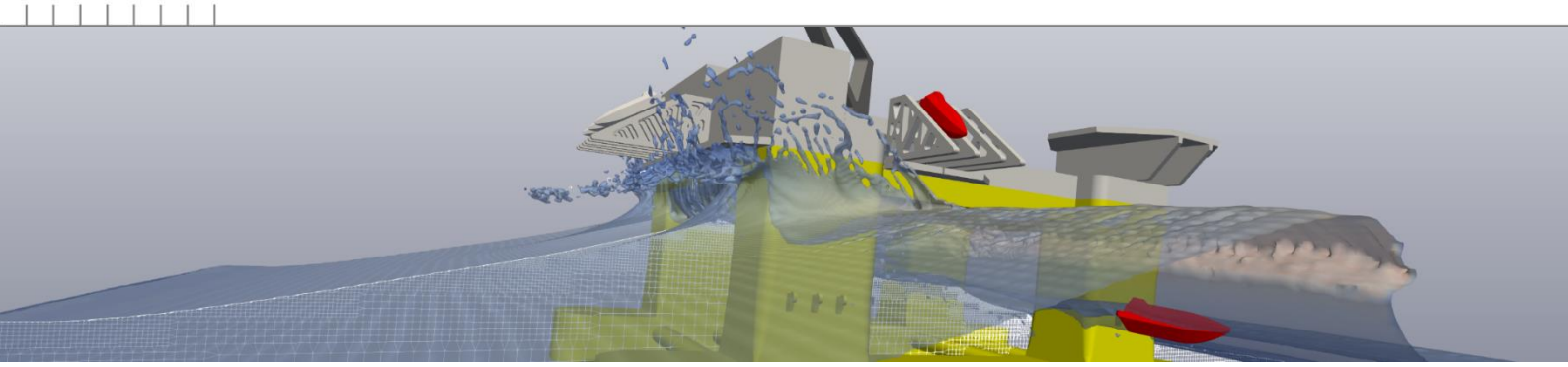


ComFLOW User Group

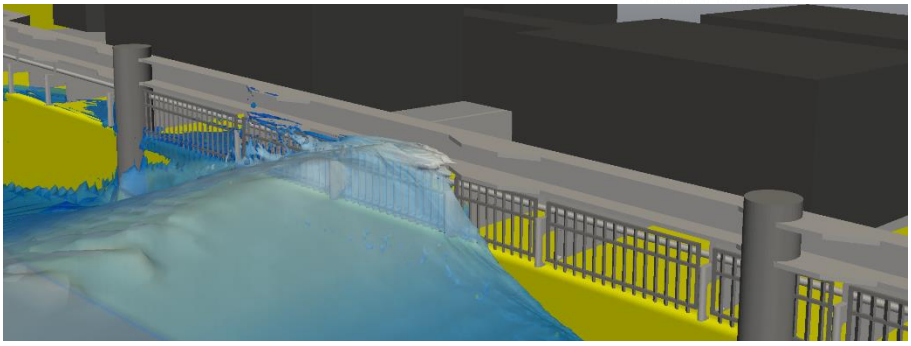
Provides a dedicated free-surface CFD tool and a platform for sharing knowledge and user experience

ComFLOW is a CFD simulation method specifically designed for the fast and efficient simulation of violent free-surface flows. More than a decade of developments within a series of JIPs resulted in a unique combination of numerical algorithms, which sets ComFLOW apart from many other CFD codes and makes it particularly effective for solving complex free-surface flow problems both in terms of CPU time and level of accuracy. With the introduction of a dedicated user group, ComFLOW is now available to all interested industrial partners for use within their daily design and engineering practice.

- ✓ **Unlimited use of executables:**
  - Windows and Linux
  - No restrictions on # nodes, # users and # simulations
- ✓ **Maintenance and support**
- ✓ **Half-yearly meetings to share knowledge and experience**
- ✓ **Additional developments when agreed upon by users:**
  - New functionality
  - Case studies
  - User coding through open modules

ComFLOW Methodology

ComFLOW is a Cartesian-grid based Volume-of-Fluid CFD code, dedicated to violent free-surface flows. Initially developed to study sloshing of liquid fuel in spacecraft, ComFLOW deploys an accurate, sharp (PLIC-VoF) description of the free surface. ComFLOW is based on a staggered finite-volume discretization of the Navier-Stokes equations, for both single- and multi-phase incompressible and compressible flows. The flow domain is covered by a locally refined grid, based on both static and adaptive (*i.e.* automatic) refinement criteria. At the in- and outflow of the domain, generating- and absorbing boundaries are applied to minimize wave reflections. A cut-cell approach for the representation of the geometry is used. ComFLOW incorporates strongly-coupled multi-body 6-DOF fluid-structure interaction, with the possibility for coupling to other numerical simulation tools (*e.g.* dynamic mooring modules or FEM software). A new generation of turbulence models is available, following the 'minimum dissipation' principle. The code is



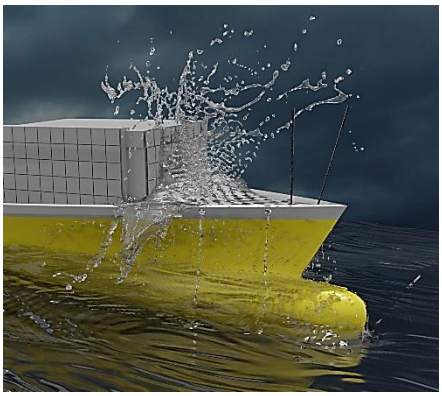
Top: Breaking wave impact on a soft-moored semi-submersible while performing two free-fall lifeboat drops (courtesy of ComMotion JIP). Bottom: Evaluating the performance of a side protection design to mitigate on-deck green water flow alongside a turret-moored FPSO (courtesy of Bluewater Energy Services).

The ComFLOW user group is open to all interested parties at the following costs:

Annual fee[1] :
- 7.5 kEuro/year

Additional entry costs[2]:
Former ComFLOW JIP members:
- 7.5 kEuro/year in first year

New participants:
- 7.5 kEuro/year in first two years



Simulation of green water over the deck of a container vessel. 'Life-like' render of ComFLOW data using dedicated post-processing (courtesy of CRS).

Minimum system requirements:
- Windows: 7 or higher
- Linux: all conventional distros (contact MARIN for details)
- Workstation or cluster with (multi-core) x64 processor(s)
- Approx. 6GB RAM per Mcells

No gridding tool required

VTK output + ParaView plugins

parallelized by means of a hybrid OpenMP-MPI strategy and runs on both Windows and Linux workstations, clusters and supercomputers.

Applicability

ComFLOW has been successfully applied and validated for a wide range of free-surface applications for fixed and moving structures, both moored and at forward speed:

- sloshing (*e.g.* in LNG containment systems and swimming pools on yachts);
- wave run-up and deck impacts (*e.g.* semi-submersibles, TLPs, monopiles);
- air entrapment during breaking wave impacts ("cushioning");
- green water (*e.g.* at the bow and alongside FPSOs and ocean-going vessels);
- non-linear wave propagation, deterministic breaking and crest distributions;
- global response analysis;
- design optimization (*e.g.* bulwarks, green water protection, riser balconies, local deflectors for on-deck structures such as lifeboats);
- bow and stern slamming (*e.g.* cruise ships).

User group

Over the course of 4 consecutive JIPs and with financial support of the Dutch government, the offshore industry, MARIN and the Universities of Groningen and Delft have worked together on the development of ComFLOW. During the latest JIP they jointly decided to follow-up these developments by introducing a dedicated user group. This will enable all current and previous ComFLOW users as well as newly interested parties to make use of the most recent ComFLOW developments, and to benefit from each other's knowledge and experience.

For more information contact MARIN:

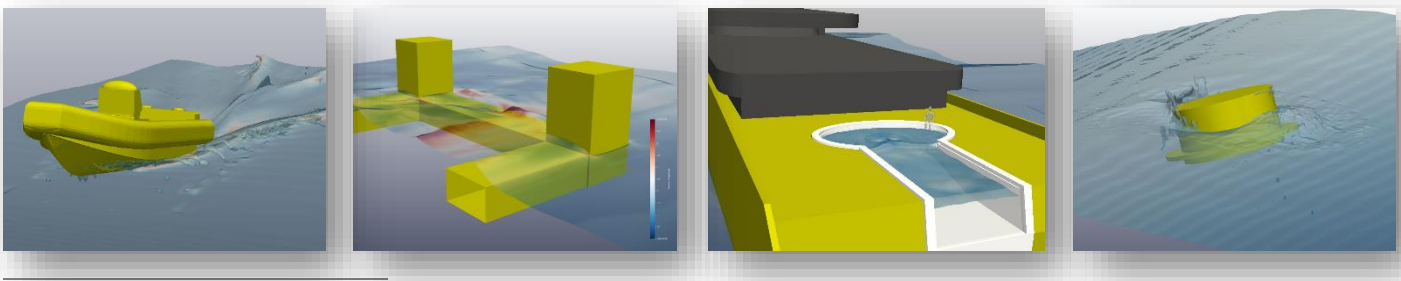
Joop Helder
T + 31 317 49 35 11
E j.helder@marin.nl

Peter van der Plas
T + 31 317 49 35 34
E p.v.d.plas@marin.nl

Additional information on ComFLOW can be found in the following papers and many references therein:

Veldman, A.E.P. et al. "*Turbulence Modelling, Local Grid Refinement and Absorbing Boundary Conditions for Free-Surface Flow Simulations in Offshore Applications*.". OMAE2014-24427.

Veldman, A.E.P. et al. *"Computational Methods for Moving and Deforming Objects in Extreme Waves"*. OMAE2019-96321.



[1] University of Groningen (RUG) and Delft University of Technology (TUD) are non-paying user-group members and consultants to the user group.
[2] No entry costs apply to ComMotion JIP members: Damen, Deltares, DNV-GL, Force, GustoMSC, HHI and MARIN. Former ComFLOW JIP members are: ABS, Aker Solutions, Astano/Navantia, Ausenco Sandwell, Bluewater, BP, BV, Chevron/Texaco, ConocoPhillips, DSME, FMC Sofec, Hess Corp., Health and Safety Executive (UK), IHC, Kawasaki HI, NPD, Petrobras, SHI, Shell, SBM, Statoil/Equinor and WS Atkins.