Towards a Next-Generation Multimechanics Simulation Environment

A FET–Open Project

David Le Touzé*
Adam Marsh
Matthieu Kerhuel
Daniel Barcarolo
Nicolas Grenier
***Ecole Centrale Nantes**, France*

Andrea Colagrossi
Salvatore Marrone
Matteo Antuono
***CNR–INSEAN**, Italy*

Erwan Jacquin
Guillaume Oger
Matthieu De Leffe
Pierre-Michel Guilcher
***HydrOcean,** France*

Francis Leboeuf
Joelle Caro
Andrea Amicarelli
Fang Le
***École Centrale Lyon**, France*

Nathan Quinlan
Libor Lobovský
Mihai Basa
National University of Ireland Galway

Jean-Christophe Marongiu
Etienne Parkinson
***Andritz Hydro**, Switzerland*

John Biddiscombe
Jérôme Soumagne
***CSCS**, Switzerland*

*Co-ordinator (david.letouze@ec-nantes.fr)

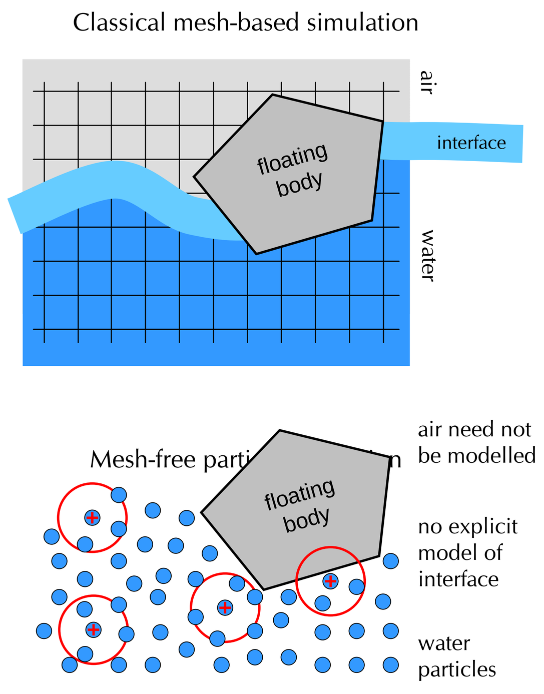
A new approach to mechanics modelling

In the **NextMuSE** team, visualisation and HPC specialists are working with engineers from diverse application areas who require more advanced and interactive simulation tools. The objective is to initiate a paradigm shift in the technology of Computational Fluid Dynamics (CFD) and Computational Multi-Mechanics (CMM) simulation software which is used to model physical processes in research, development and design across a range of industries. NextMuSE relies on a **mesh-free** method, Smoothed Particle Hydrodynamics (**SPH**), which is fundamentally different from conventional finite element or volume techniques. SPH offers the possibility of a novel, immersive, adaptive framework for user interaction, and has the potential for integrated **multi-mechanics modelling** in applications where traditional methods fail. The goal of NextMuSE is to create a simulation technology that is **robust and accurate** enough to deal with the most challenging physical phenomena in industry (e.g. simultaneous fluid and solid mechanics in a ship under extreme wave loading). Input and output will be **immersive and interactive**, and the simulation highly automated, so that the technology itself will be nearly invisible to the engineers who use it to develop new technology for **transport, energy, healthcare** and other sectors.

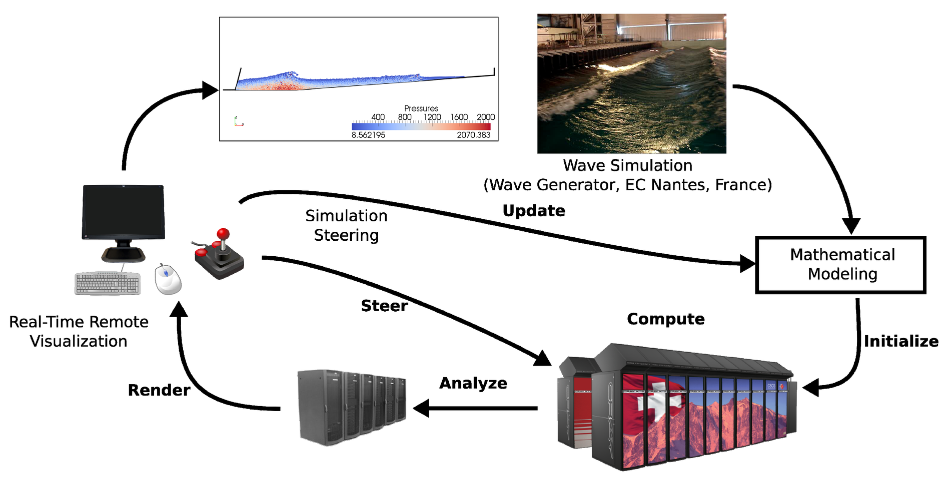
Computational Engine

Mesh-free particle simulation methods like SPH have emerged as an alternative to classical mesh-based methods. Instead of modelling the material with a fixed spatial mesh, the fluid or solid is modelled with moving particles. Particles have no explicit connectivity, but interact with neighbours within a specified range. This approach brings advantages in the treatment of complex physics with interfaces and moving bodies.

However, SPH is a relatively new technique. Within this project we are making progress on accuracy, speed, mathematical properties, and modelling of physical phenomena such as viscosity, multiphase flow and turbulence. One of the aims of this project is to develop SPH-based simulation that is reliable enough to be an engineering design tool.



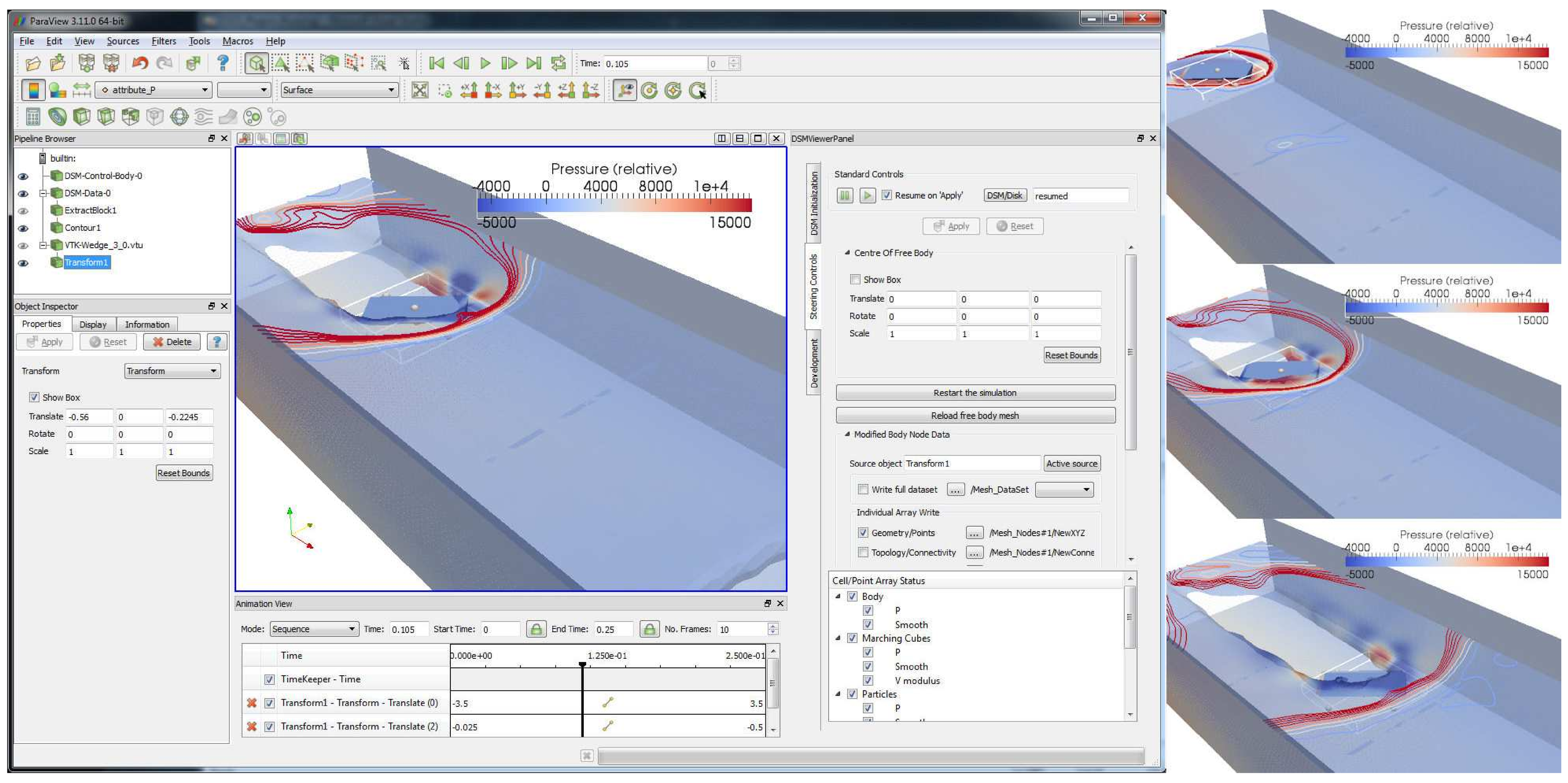
User Interface



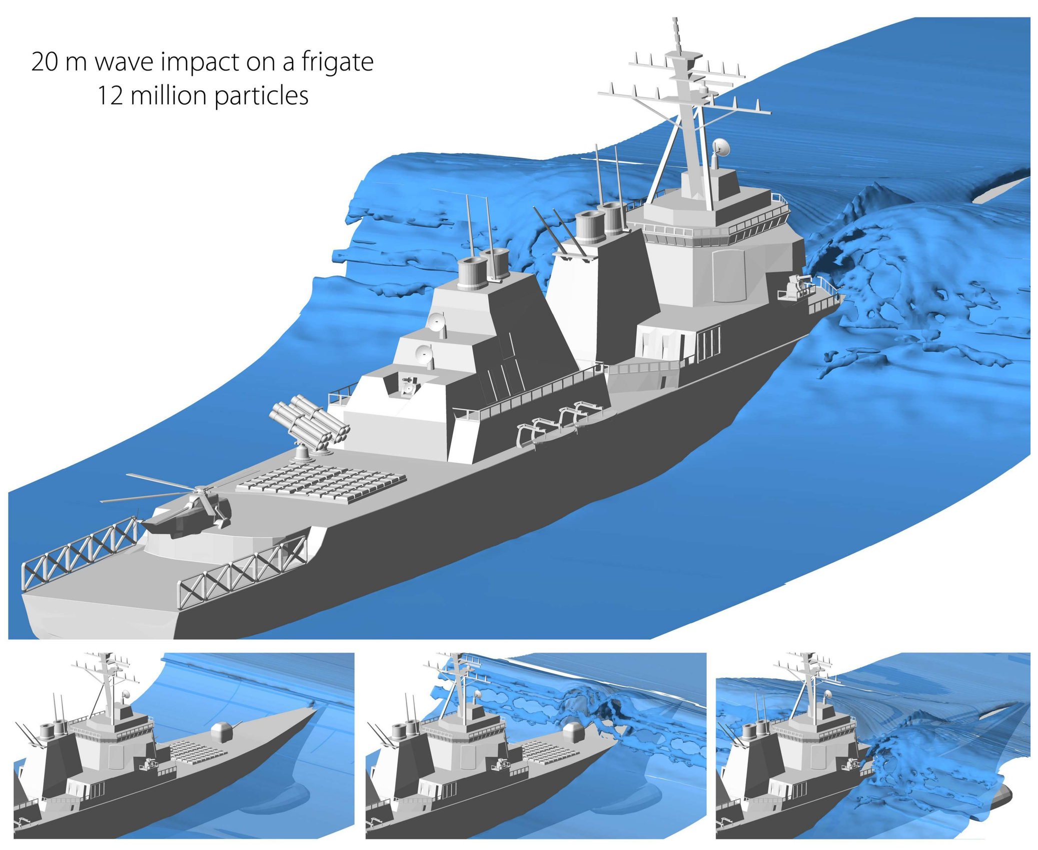
The **ICARUS** (Initialize Compute Analyze Render Update Steer) interface is a hardware/software-based computational visualization and steering environment which will act as a coordinating centre for all the packages within the project. ICARUS is implemented as a **ParaView** plug-in.

ICARUS makes use of **H5FDdsm,** is a Virtual File Driver for HDF5 which uses parallel communication to transfer data between applications using the HDF5 IO API and a distributed shared memory (DSM) buffer. HDFDdsm enables an ICARUS user to interact dynamically with a simulation running **millions of particles** on a remote parallel system. A user may perform geometric changes or deformations, or adjust simulation parameters, while the simulation executes. Source code is available on **hpcforge.org**.

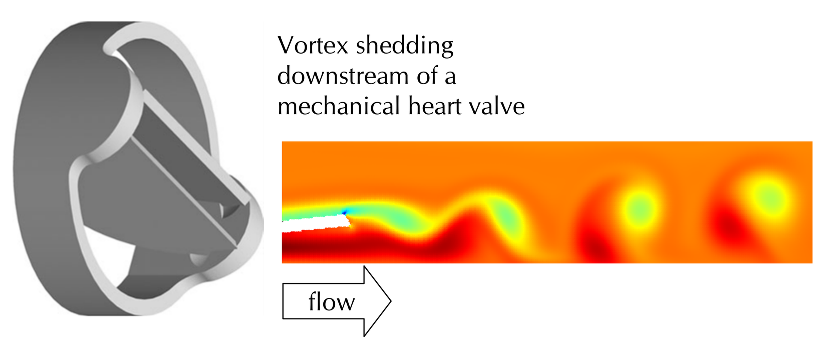
A user manipulates an object in a fluid during a live simulation, while output is analysed and rendered in the form of pressure contours.



Applications



Survival of **ships in extreme sea conditions** represents a difficult topic for numerical simulations. It is currently treated using simple models based on hydrostatic considerations, and with the help of empirical models. SPH methods have shown the capability to resolve the motion of the water surface, even in extreme events, and coupling with the motion of a ship.



Biomedical systems present unique modelling challenges, since they operate with very large structural motions and deformations (e.g. of a heart valve or artery wall). We are using the ICARUS-SPH paradigm to model a **mechanical heart valve** (MHV). To date, the difficulty of computational modelling has been an impediment to progress in MHV design. Preliminary results show that SPH can predict the major flow features observed experimentally.

The **Pelton turbine** is a hydraulic machine consisting of "buckets" on a rotating runner driven by high-velocity impinging water jets, used to produce energy from high head waterfalls. Numerical simulation can help in improving the runner design and the turbine efficiency, but traditional techniques are unable to accurately model the water sheets released from the buckets, because of excessive numerical diffusion. Using SPH it is possible to study the interactions of water sheets issued from one jet on the other, and the resulting possible perturbations. The on-line steering of such a computation by ICARUS includes variation of the upstream conditions (head, discharge), in order to simulate conditions over the turbine in one simulation run.

Centrale Nantes   INSEAN   ANDRITZ Hydro   HydrOcean   CSCS Swiss National Supercomputing Centre   ÉCOLE CENTRALE LYON   NUI Galway OÉ Gaillimh   SEVENTH FRAMEWORK PROGRAMME

Co-ordinator: David Le Touzé, Ecole Centrale Nantes, France. david.letouze@ec-nantes.fr

Funded by the European Community's Seventh Framework Programme (FP7/2007-2013) under grant agreement 225967 "NextMuSE".