Novelties on DualSPHysics v6.0 BETA

G. Fourtakas & the DualSPHysics team

Overview of DualSPHysics

COLLABORATIVE PROJECT

OPEN-SOURCE CODE

LGPL (Lesser General Public License) can be **used in commercial applications**. Software can be incorporated into both free software and proprietary software.

DEVELOPERS:

Universidade de Vigo, Spain
The University of Manchester, UK
Instituto Superior Tecnico, Lisbon, Portugal
Università degli studi di Parma, Italy
Universitat Politècnica de Catalunya, Spain
New Jersey Institute of Technology, USA

COLLABORATORS:

Flanders Hydraulics Research , Belgium
Universidad Politécnica de Madrid, Spain
TECNALIA. Inspiring Business, Spain
Imperial College London, UK
Universiteit Gent, Belgium
University of Salerno, Italy
Universidad de Guanajuato, Mexico
Uppsala University, Sweden
… many more

Overview of DualSPHysics

DUALSPHYSICS TEAM

Project Leaders:
- José M. Domínguez. Universidade de Vigo, Spain
- Georgios Fourtakas. The University of Manchester, UK
- Alejandro J.C. Crespo. Universidade de Vigo, Spain
- Benedict D. Rogers. The University of Manchester, UK
- Renato Vacondio. Università degli studi di Parma, Italy
- Corrado Altomare. Universitat Politècnica de Catalunya, Spain
- Angelo Tafuni. New Jersey Institute of Technology, US

Project Coordinators:
- Peter Stansby. The University of Manchester, UK
- Moncho Gómez Gesteira. Universidade de Vigo, Spain

Developers:
- Orlando García Feal. Universidade de Vigo, Spain
- Joseph O’Connor. The University of Edinburgh, UK
- Aaron English. Università degli studi di Parma, Italy
- Iván Martínez Estévez. Universidade de Vigo, Spain
- Francesco Ricci. Università degli studi di Parma, Italy

Wiki Coordinator:
- Bonaventura Tagliafierro. Uppsala Universitet, Sweden

Universidade Vigo

MANCHESTER 1824 The University of Manchester

UNIVERSITÀ DI PARMA

UNIVERSITAT POLITÈCNICA DE CATALUNYA BARCELONATECH

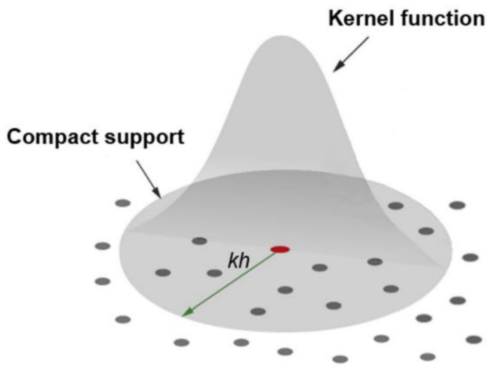
NJIT New Jersey Institute of Technology

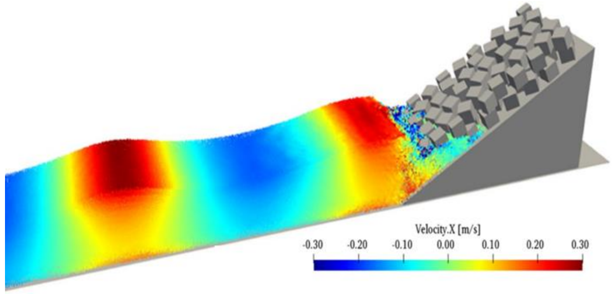
THE UNIVERSITY of EDINBURGH

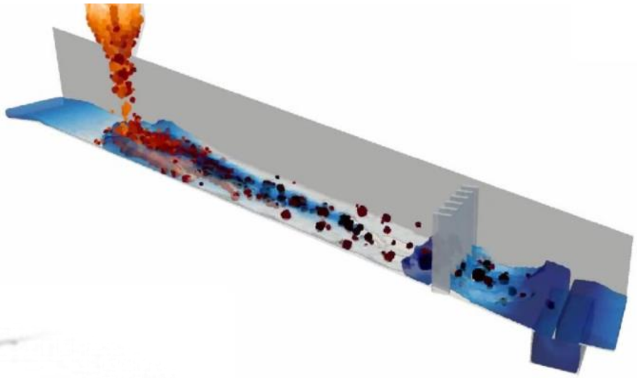
UPPSALA UNIVERSITET

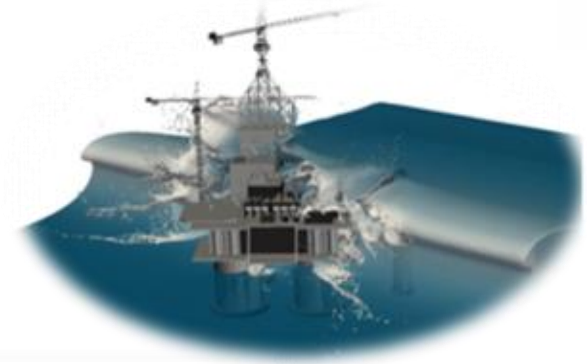
Overview of DualSPHysics

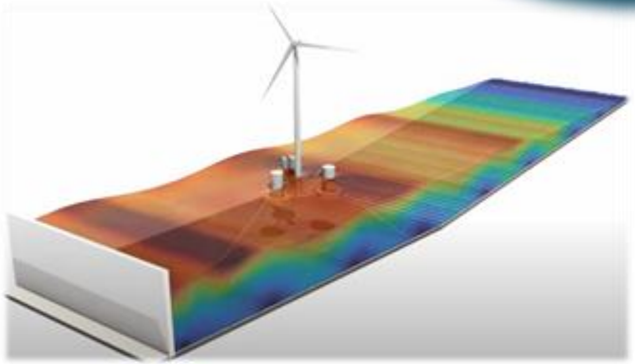
Implementation of **Smoothed Particle Hydrodynamics** method for **complex fluid dynamics** using HPC

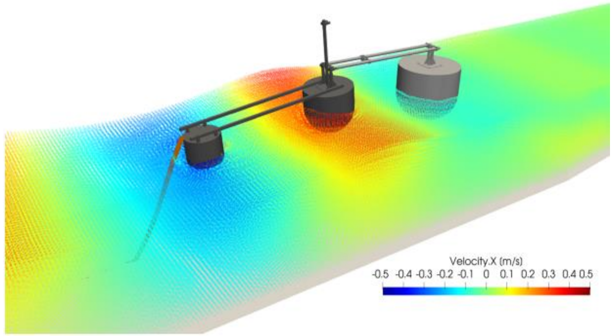












Overview of DualSPHysics

SPH HIGHLY PARALLELISED

GPU

NVIDIA CUDA

CPU

GPU CPU

x100

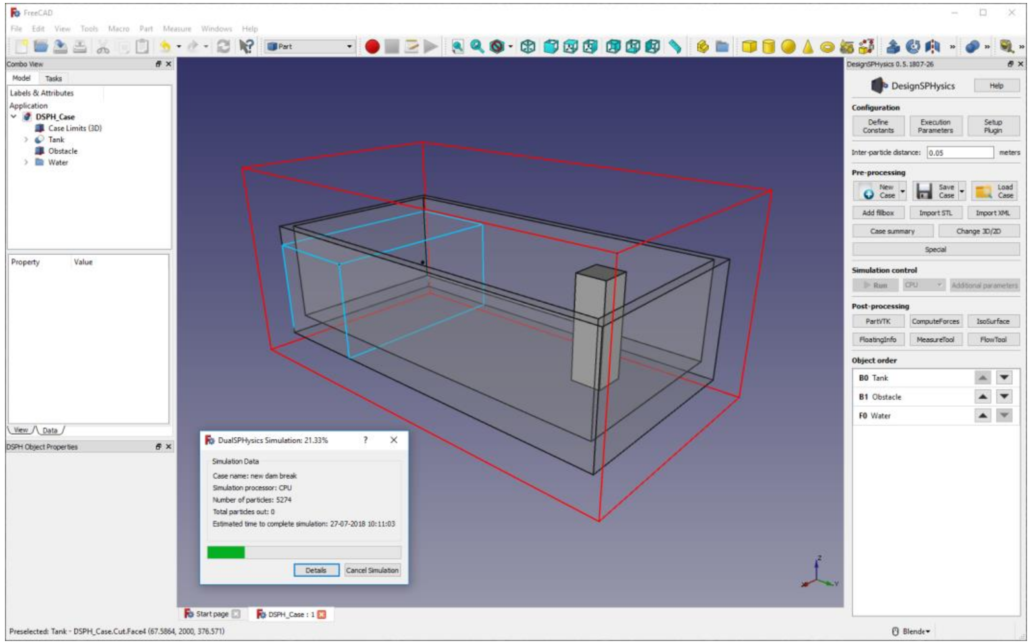
It includes **two implementations**:
- **CPU**: C++ and OpenMP.
- **GPU**: CUDA.

Both options optimized for the best performance of each architecture.

Overview of DualSPHysics

DesignSPHysics

(Graphical User Interface)



COMPLETE TOOLKIT

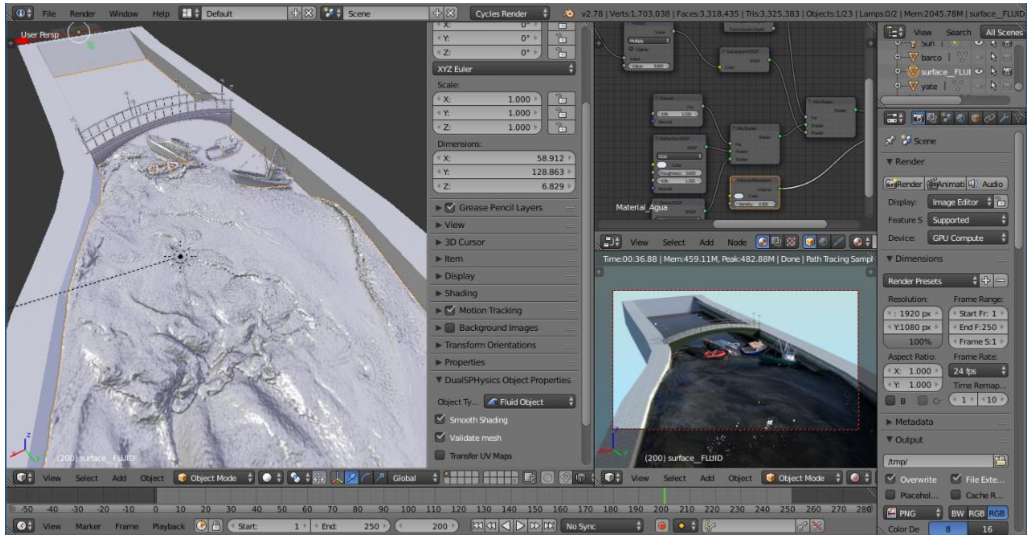
- SPH solver
- Pre-processing tools
- Post-processing tools

but also…

- Graphical User Interface
- Advanced visualisation

VisualSPHysics

(Advanced visualisation tool)



DualSPHysics downloads

DualSPHysics V1.2 (2011)
Downloads: 701 (65% Windows)

DualSPHysics V2.0 (2012)
Downloads: 6,472 (71% Windows)

DualSPHysics V3.0 (2013-2015)
Downloads: 14,426 (73% Windows)

DualSPHysics V4.0 (2016)
Downloads: 13,804 (72% Windows)

DualSPHysics V4.2 (May 2018)
Downloads: 8,429

DualSPHysics V4.4 (April 2019)
Downloads: 22,520

DualSPHysics V5.0 (July 2020)
Downloads: 63,735

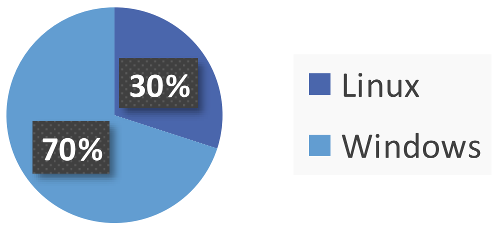
DualSPHysics V5.2 (May 2023)
Downloads: 20,163

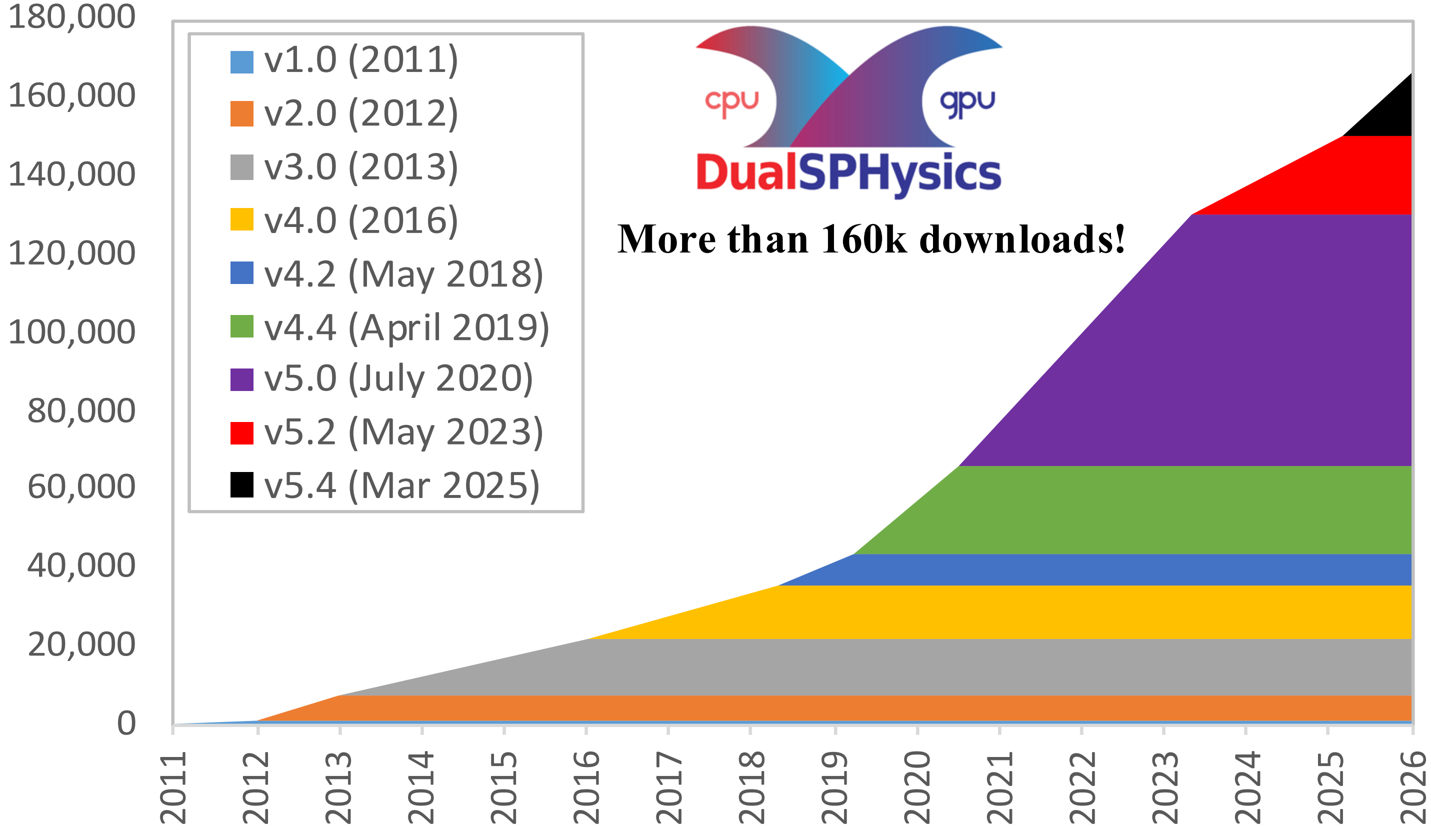
DualSPHysics V5.4 (Mar 2025)
Downloads: 15,996

DualSPHysics - ALL VERSIONS

Downloads: 166,246 (70% Windows)

https://dual.sphysics.org/downloads/





New Features in v6.0 beta

SPH solver novelties:

Solvers

- New **multi-GPU SPH solver** with dynamic load balancing
- New **single-phase geomechanics solver** based on DualSPHysics v5.2 (GEOMECH)

Methodologies

- New **coupling with FEA** module based on Project Chrono v9.
- A **velocity divergence cleaning (incompressibility control)** formulation for filtering acoustic waves
- New **improved mDBC mode** (TanVel0) that impose v=0 to boundary particles velocity with penetration correction
- **Staggered time-stepping** scheme within the **TLSPH** (FlexStruct)

Features

- New **restart** option for DBC, mDBC, floating bodies and wave generation with AWAS.
- The mDBC kernel correction and **penetration control** method is also applied to floating body particles.
- Combination of multiple **BC configurations to MK** (Mixed BC).
- **MoorDynPlus v2.2** with one-way fluid-mooring interaction.
- **Project Chrono v9** with significant performance improvements.
- Wave generation **code** (precompiled JWaveGenLib library dependency removed).

New Features in v6.0 beta

Pre-processing novelties (*GenCase*):

- Fluid properties initialisation based on previous 2D or 3D simulations.
- Free drawing mode with points defined in external files.
- General improvements in performance and memory management.

Documentation:

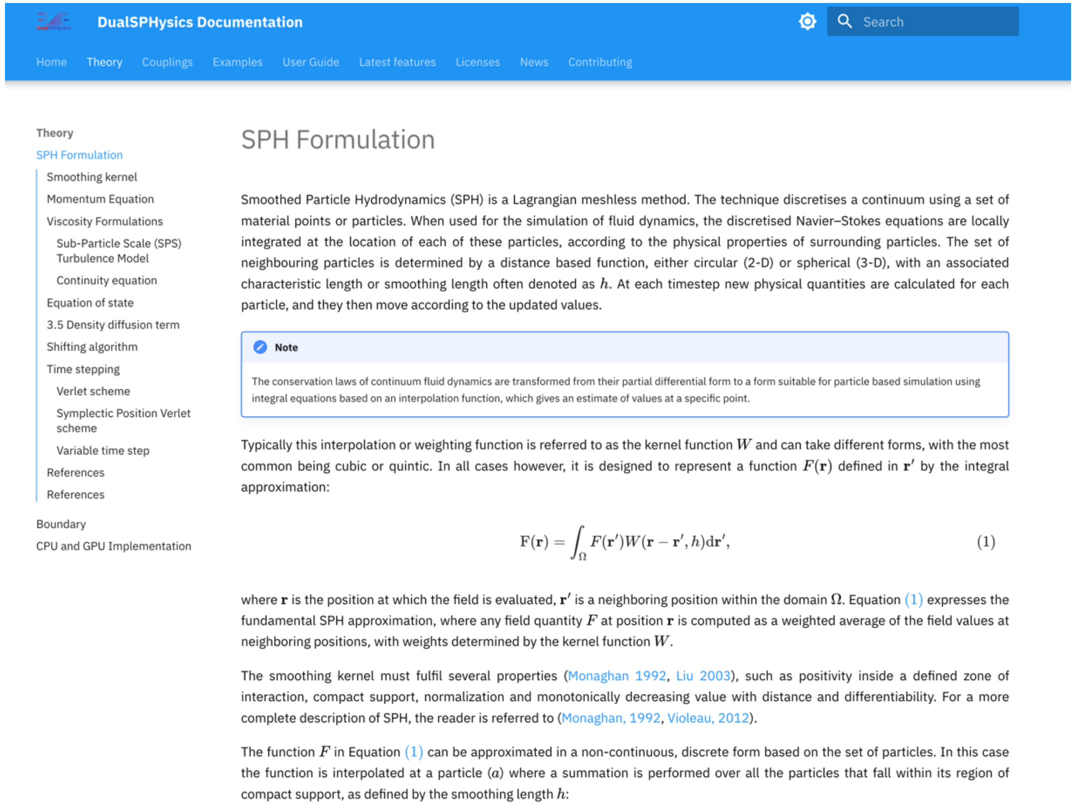
- New online Wiki with the formulation.

Post-processing novelties:

- New option -elevationrel to compute fluid elevation from bottom line definition (MeasureTool).
- New option -savevtkmesh to generate VTK files as triangle meshes instead of points with fluid properties like velocity or pressure (MeasureTool).
- Important performance improvement in moments calculation (ComputeForces).

…and *many other minor improvements*, bug fixes, code optimisation, etc.

Wiki documentation for DualSPHysics: work in progress



DualSPHysics Documentation

Search

Home Theory Couplings Examples User Guide Latest features Licenses News Contributing

Theory
SPH Formulation
Smoothing kernel
Momentum Equation
Viscosity Formulations
Sub-Particle Scale (SPS) Turbulence Model
Continuity equation
Equation of state
3.5 Density diffusion term
Shifting algorithm
Time stepping
Verlet scheme
Symplectic Position Verlet scheme
Variable time step
References
References
Boundary
CPU and GPU Implementation

SPH Formulation

Smoothed Particle Hydrodynamics (SPH) is a Lagrangian meshless method. The technique discretises a continuum using a set of material points or particles. When used for the simulation of fluid dynamics, the discretised Navier–Stokes equations are locally integrated at the location of each of these particles, according to the physical properties of surrounding particles. The set of neighbouring particles is determined by a distance based function, either circular (2-D) or spherical (3-D), with an associated characteristic length or smoothing length often denoted as h. At each timestep new physical quantities are calculated for each particle, and they then move according to the updated values.

Note

The conservation laws of continuum fluid dynamics are transformed from their partial differential form to a form suitable for particle based simulation using integral equations based on an interpolation function, which gives an estimate of values at a specific point.

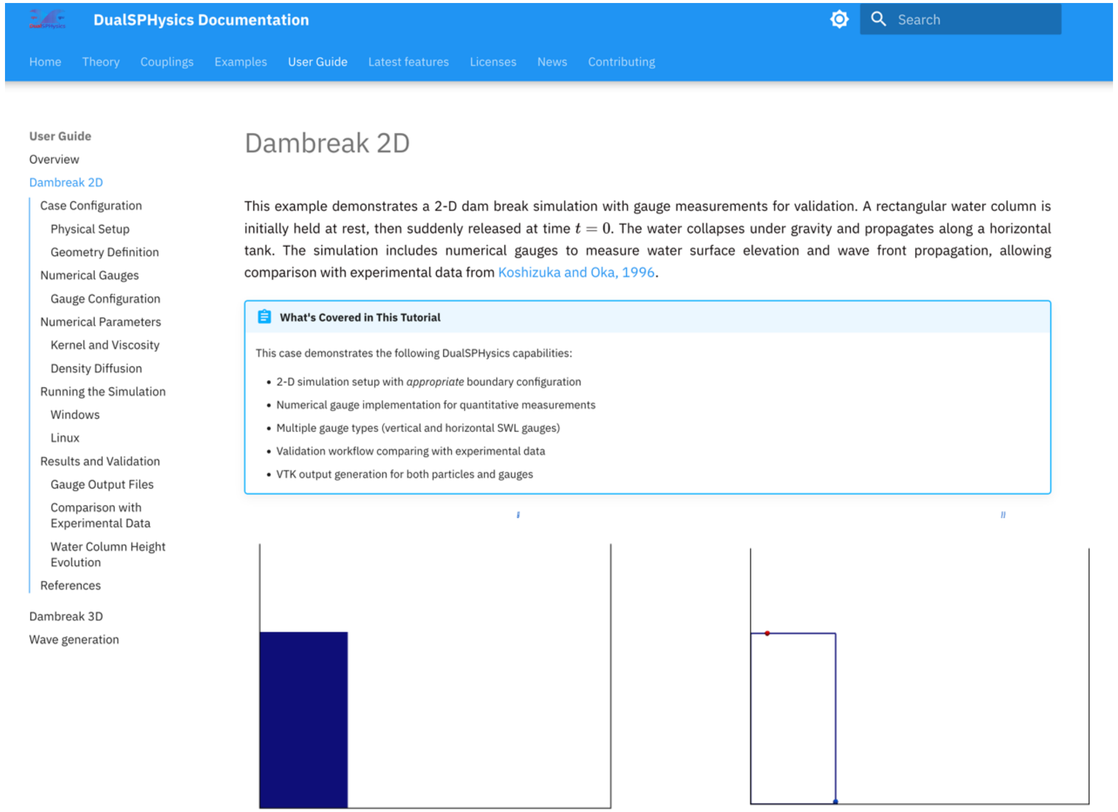
Typically this interpolation or weighting function is referred to as the kernel function W and can take different forms, with the most common being cubic or quintic. In all cases however, it is designed to represent a function $F(\mathbf{r})$ defined in $\mathbf{r}'$ by the integral approximation:

$$F(\mathbf{r}) = \int_{\Omega} F(\mathbf{r}')W(\mathbf{r}-\mathbf{r}', h)d\mathbf{r}', \quad (1)$$

where $\mathbf{r}$ is the position at which the field is evaluated, $\mathbf{r}'$ is a neighboring position within the domain Ω. Equation (1) expresses the fundamental SPH approximation, where any field quantity F at position $\mathbf{r}$ is computed as a weighted average of the field values at neighboring positions, with weights determined by the kernel function W.

The smoothing kernel must fulfil several properties (Monaghan 1992, Liu 2003), such as positivity inside a defined zone of interaction, compact support, normalization and monotonically decreasing value with distance and differentiability. For a more complete description of SPH, the reader is referred to (Monaghan, 1992, Violeau, 2012).

The function F in Equation (1) can be approximated in a non-continuous, discrete form based on the set of particles. In this case the function is interpolated at a particle (a) where a summation is performed over all the particles that fall within its region of compact support, as defined by the smoothing length h:



DualSPHysics Documentation

Search

Home Theory Couplings Examples User Guide Latest features Licenses News Contributing

User Guide
Overview
Dambreak 2D
Case Configuration
Physical Setup
Geometry Definition
Numerical Gauges
Gauge Configuration
Numerical Parameters
Kernel and Viscosity
Density Diffusion
Running the Simulation
Windows
Linux
Results and Validation
Gauge Output Files
Comparison with Experimental Data
Water Column Height Evolution
References
Dambreak 3D
Wave generation

Dambreak 2D

This example demonstrates a 2-D dam break simulation with gauge measurements for validation. A rectangular water column is initially held at rest, then suddenly released at time $t = 0$. The water collapses under gravity and propagates along a horizontal tank. The simulation includes numerical gauges to measure water surface elevation and wave front propagation, allowing comparison with experimental data from Koshizuka and Oka, 1996.

What's Covered in This Tutorial

This case demonstrates the following DualSPHysics capabilities:

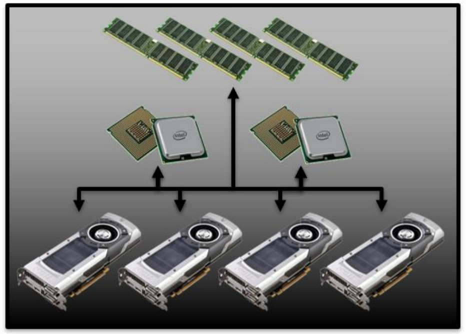
- 2-D simulation setup with *appropriate* boundary configuration
- Numerical gauge implementation for quantitative measurements
- Multiple gauge types (vertical and horizontal SWL gauges)
- Validation workflow comparing with experimental data
- VTK output generation for both particles and gauges

- Latest formulation for SPH theory and coupling methodologies
- Integrates user guide from xml to theory
- Updated example cases

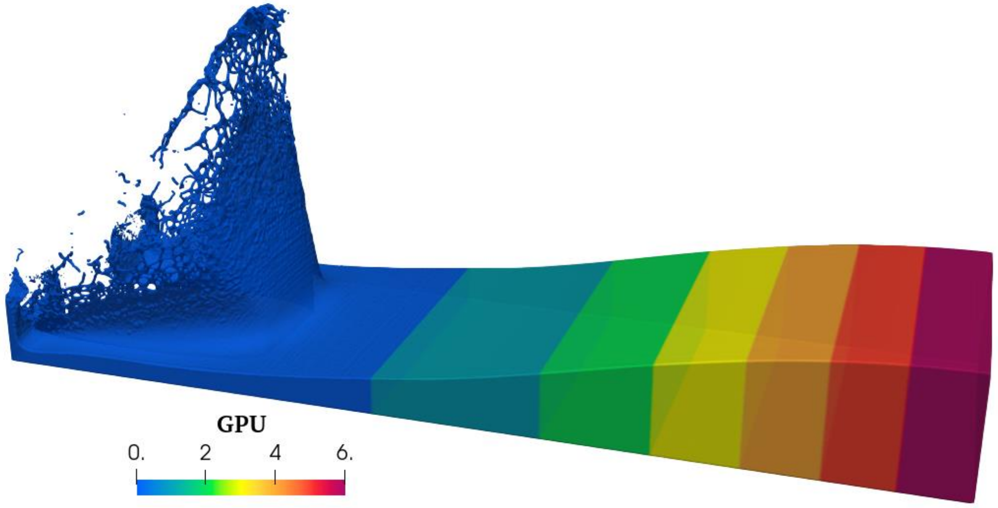
New Multi-GPU approach for single-node

Implementation based on C++ threads and CUDA streams (not MPI)

- **Efficient multi-GPU to run on workstations** or computing nodes with 4-10 GPUs
- Multi-GPU **useful for researchers** using DualSPHysics (not computer engineers)
- More **portable and easy to use** in Linux & Windows
- **The target is full support** of all current DualSPHysics functionalities
- Enables **100-200M** particle simulations **without extra user effort**
- Not special pre-processing and post-processing tools required
- BETA version **available in the DualSPHysics package v6.0** (only for workshop attendees)

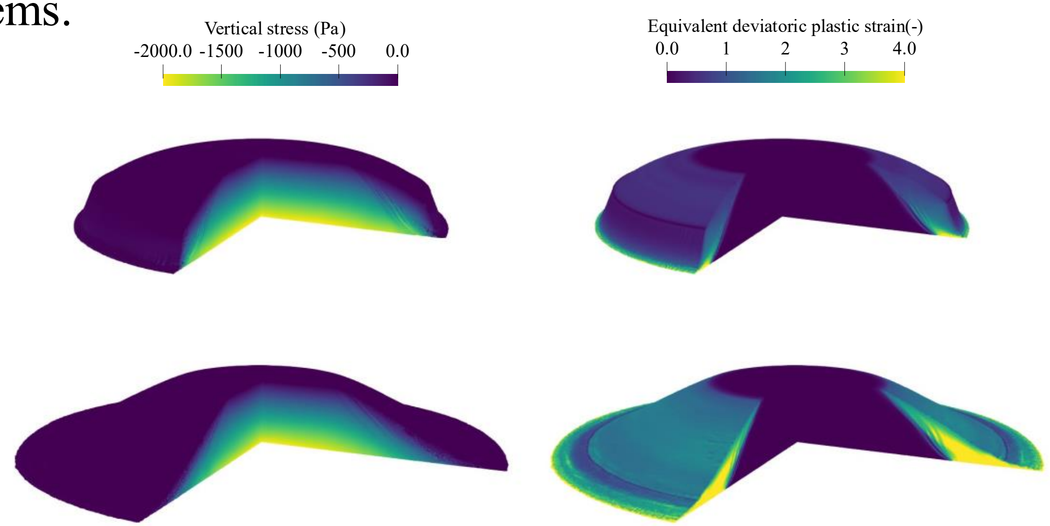


multi-GPU machine



GEOMECH model in DualSPHysics

- SPH formulations based on the total stress tensor.
- Implements **elastic-plastic constitutive model** with **strain-softening** for **cohesionless** and **cohesive** granular materials.
- Introduces a new cDBC and extends mDBC for stress tensor boundary treatment.
  - cDBC based on the first-order local interpolation (no normals)
  - mDBC based on the first-order extrapolation (more accurate)
- Enables **coupling with Chrono** for the interaction with multi-body systems.



Computer Physics Communications 320 (2026) 109965

Contents lists available at ScienceDirect

Computer Physics Communications

journal homepage: www.elsevier.com/locate/cpc

ELSEVIER

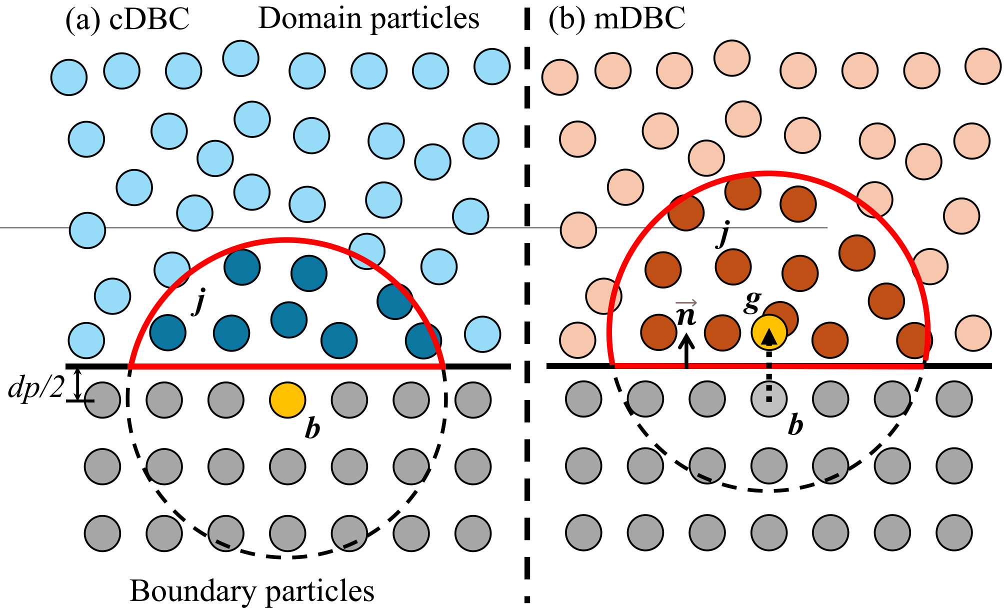
Computer Programs in Physics

GeoDualSPHysics: a high-performance SPH solver for large deformation modelling of geomaterials with two-way coupling to multi-body systems

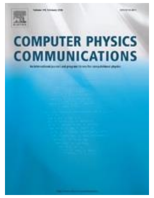
Ruofeng Feng [a,*], Jidong Zhao [a,*], Georgios Fourtakas [b], Benedict D Rogers [b]

[a] Department of Civil and Environmental Engineering, Hong Kong University of Science and Technology, China
[b] School of Engineering, The University of Manchester, UK

Feng, R., Zhao, J., Fourtakas, G., & Rogers, B.D. 2026. **GeoDualSPHysics: a high-performance SPH solver for large deformation modelling of geomaterials with two-way coupling to multi-body systems.** *Computer Physics Communications*, 320, 109965. doi:10.1016/j.cpc.2025.109965

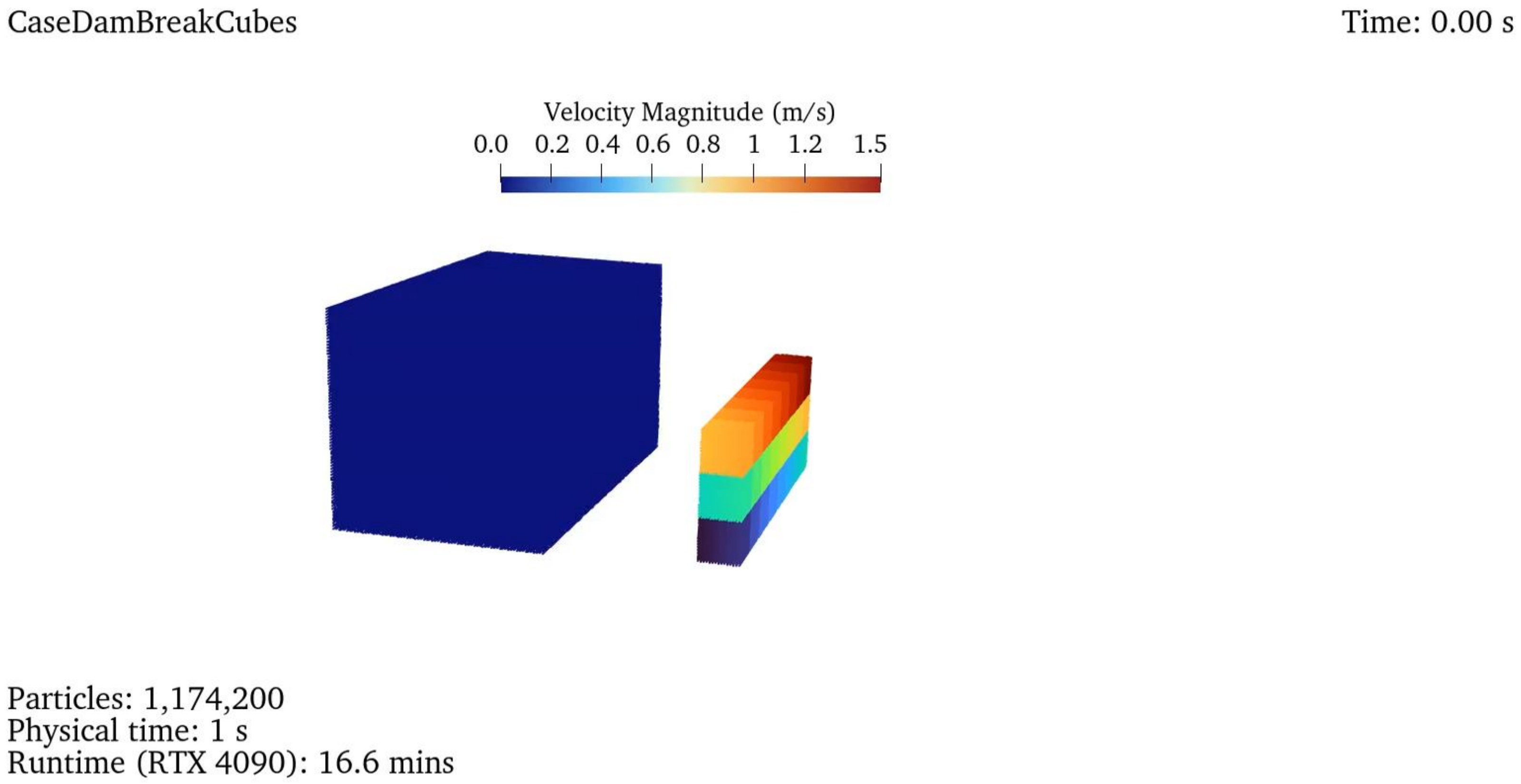


GEOMECH model in DualSPHysics



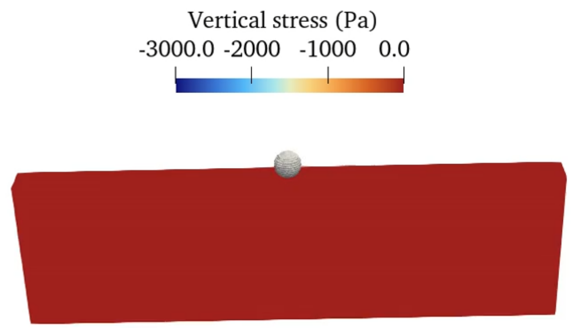
Feng, R., Zhao, J., Fourtakas, G., & Rogers, B.D. 2026. **GeoDualSPHysics: a high-performance SPH solver for large deformation modelling of geomaterials with two-way coupling to multi-body systems.** *Computer Physics Communications*, 320, 109965. doi:10.1016/j.cpc.2025.109965

- Allows the simulation of **large deformation of geomaterials** following elastic-plastic theory.
- Enables **coupling with Chrono** for geo–multibody interactions.
- Force gauge is reformulated for impact force evaluations.
- Expands solver scope to geomechanics, terramechanics, and granular & powder handling processes.



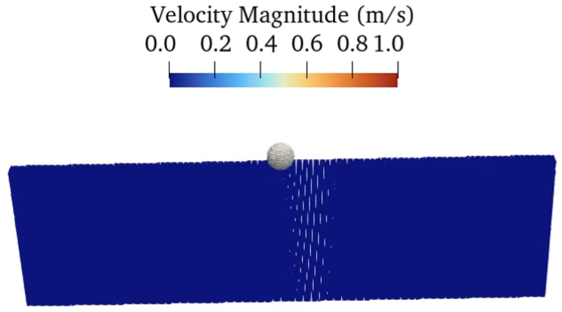
Time: 0.0 s

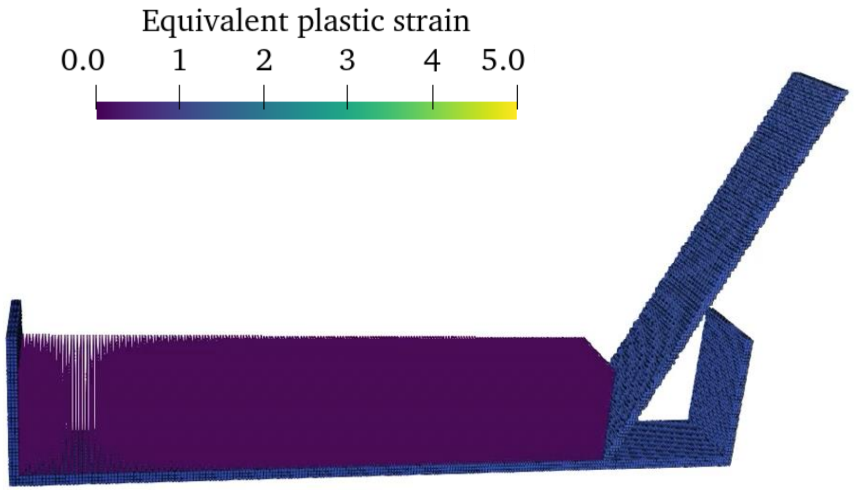
CaseBallDrop3D



Particles: 444,893
Physical time: 0.2 s
Runtime (RTX 4090): 69 s

Time: 0.000 s





Coupling DualSPHysics-FEA

Software components

DualSPHysics ⇄ DSPHChronoLib ⇄ PROJECT CHRONO

- Solves the fluid
- Solves the fluid-solid interaction

- Controls the communication process
- Handles the information transfer

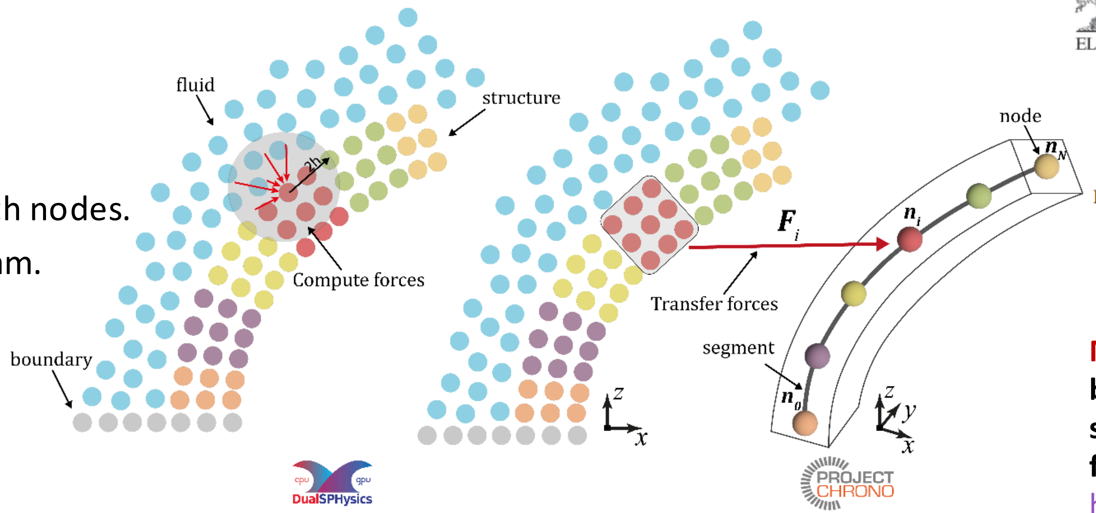
- Solves mechanical constraints
- Solves collisions
- Solves flexible structure


Computer Physics Communications 283 (2023) 108581
Contents lists available at ScienceDirect
Computer Physics Communications
journal homepage: www.elsevier.com/locate/cpc
Coupling of an SPH-based solver with a multiphysics library
I. Martínez-Estévez [a,*], J.M. Domínguez [a], B. Tagliafierro [b], R.B. Canelas [c], O. García-Feal [a], A.J.C. Crespo [a], M. Gómez-Gesteira [a]
[a] Environmental Physics Laboratory, CIM-UVIGO, Universidade de Vigo, Spain
[b] Department of Civil Engineering, University of Salerno, Italy
[c] Bentley Systems, Lisbon, Portugal

Martínez-Estévez et al., (2023)a. Coupling of an SPH-based solver with a multiphysics library. https://doi.org/10.1016/j.cpc.2022.108581

SPH-FEA formulation

- **SPH** method for fluid.
- Finite Element Analysis (**FEA**) for structure.
- Structure is a set of segments connected with nodes.
  - Each segment is a 3-D Euler-Bernoulli beam.
  - Corotational formulation.
  - Large rotations and displacements.
- Each mode is a Finite Element with 6 DoF.



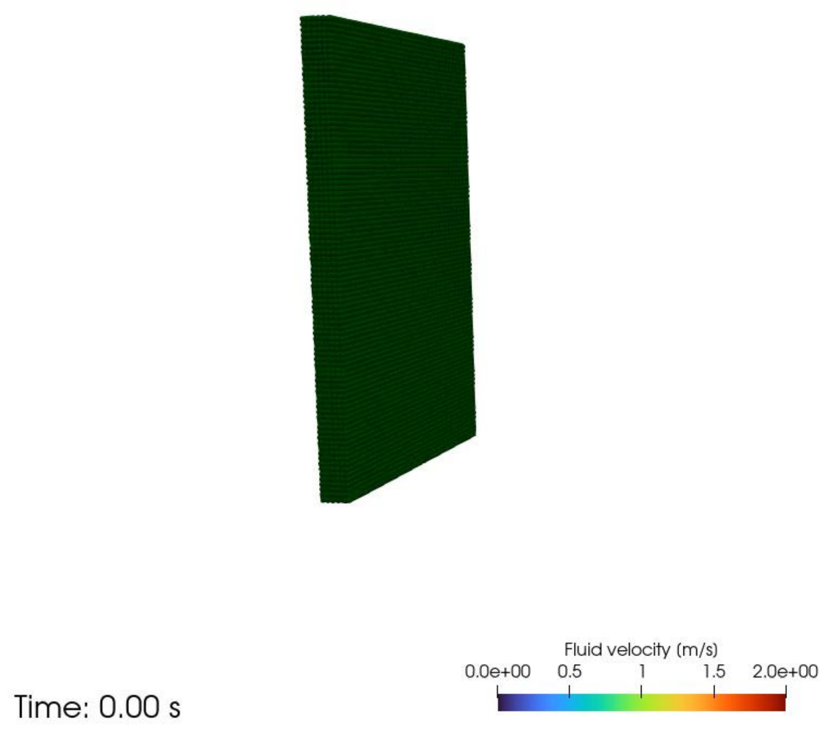

Available online at www.sciencedirect.com
ScienceDirect
Comput. Methods Appl. Mech. Engrg. 410 (2023) 115989
Computer methods in applied mechanics and engineering
www.elsevier.com/locate/cma
Coupling an SPH-based solver with an FEA structural solver to simulate free surface flows interacting with flexible structures
I. Martínez-Estévez [a,*], B. Tagliafierro [b], J. El Rahi [c], J.M. Domínguez [a], A.J.C. Crespo [a], P. Troch [c], M. Gómez-Gesteira [a]
[a] Environmental Physics Laboratory, CIM-UVIGO, Universidade de Vigo, Spain
[b] Laboratori d'Enginyeria Marítima, Universitat Politècnica de Catalunya - BarcelonaTech (UPC) Barcelona, Spain
[c] Department of Civil Engineering, Ghent University, Belgium
Received 9 December 2022; received in revised form 23 February 2023; accepted 3 March 2023
Available online 27 March 2023

Martínez-Estévez et al., (2023)b. Coupling an SPH-based solver with an FEA structural solver to simulate free surface flows interacting with flexible structures. https://doi.org/10.1016/j.cma.2023.115989

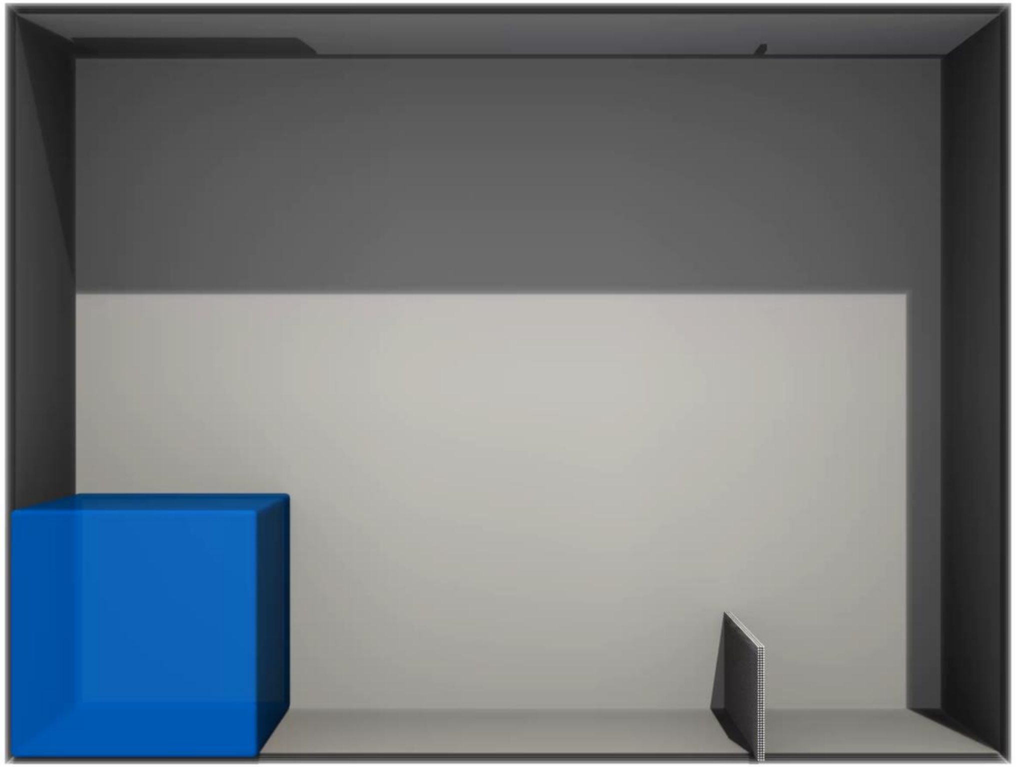
Coupling DualSPHysics-FEA

Examples

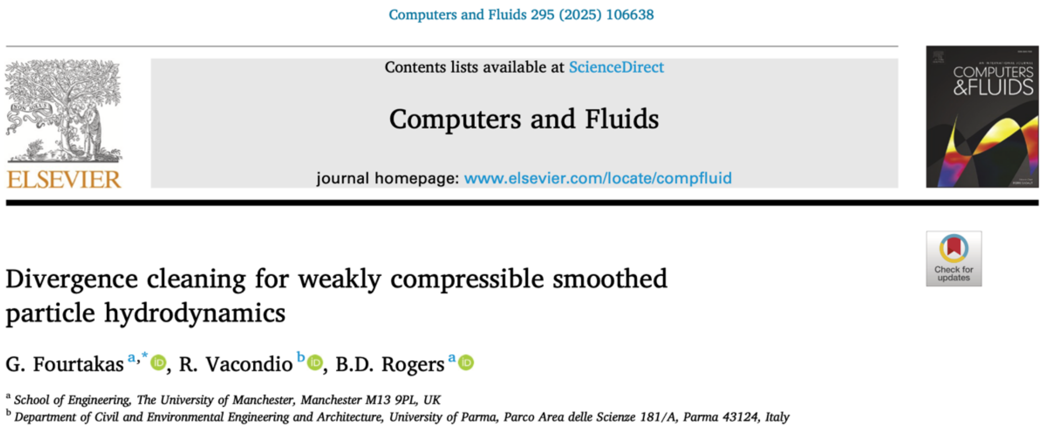
Flexible pendulum



Dam break with rubber plate



Velocity divergence cleaning


Computers and Fluids 295 (2025) 106638

Contents lists available at ScienceDirect

Computers and Fluids

journal homepage: www.elsevier.com/locate/compfluid

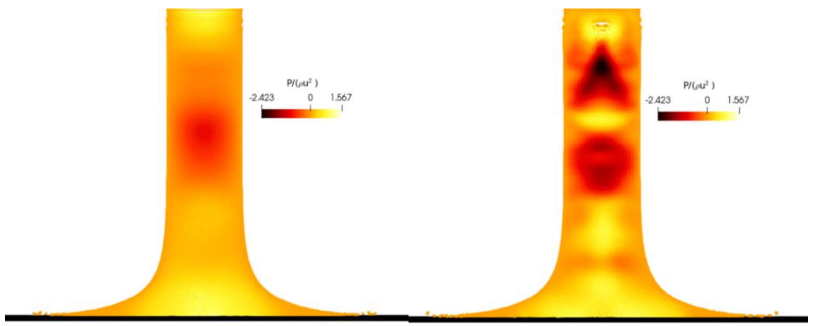
Divergence cleaning for weakly compressible smoothed particle hydrodynamics

G. Fourtakas [a,*], R. Vacondio [b], B.D. Rogers [a]

[a] School of Engineering, The University of Manchester, Manchester M13 9PL, UK
[b] Department of Civil and Environmental Engineering and Architecture, University of Parma, Parco Area delle Scienze 181/A, Parma 43124, Italy

- Hyperbolic/parabolic **divergence cleaning**
- Dissipation of acoustic waves with a speed of sound proportional to the local Mach number.
- **Eliminate non-physical acoustic waves** in the pressure field,

<u>Highly desirable in impact flows or where pressure waves are present</u>



New slip modes and mixed boundaries

Multiple solid boundary options now in DualSPHysics

Bound mode:

1. Dynamic boundary condition (DBC)
   - Vel = 0 for boundary particles

2. Modified DBC (mDBC)
   - Vel = 0 (Slipmode=1)

3. mDBC with improved pressure extrapolation
   - No-slip (Slipmode=2)
   - Free slip (Slipmode=3)
   - Vel = 0 (Slipmode=11)
   - No-penetration

4. Mixed Boundaries
   - Use any combination of the above together in the same simulation
   - Select boundary method by mkbound

Open Access ARTICLE

Boundary Conditions Generated by Dynamic Particles in SPH Methods

A. J. C. Crespo[1], M. Gómez-Gesteira[1], R. A. Dalrymple[2]

1 Grupo de Fқsica de la Atmósfera y del Océano, Facultad de Ciencias, Universidad de Vigo, Spain.

2 Department of Civil Engineering, Johns Hopkins University, Baltimore, USA.

Computers, Materials & Continua **2007**, *5*(3), 173-184. https://doi.org/10.3970/cmc.2007.005.173

Home > Computational Particle Mechanics > Article

Modified dynamic boundary conditions (mDBC) for general-purpose smoothed particle hydrodynamics (SPH): application to tank sloshing, dam break and fish pass problems

Computational Particle Mechanics

Aims and scope →

Open access | Published: 12 April 2021

Volume 9, pages 1–15, (2022) Cite this article

You have full access to this open access article

Download PDF ↓ Save article

A. English, J. M. Domínguez, R. Vacondio, A. J. C. Crespo, P. K. Stansby, S. J. Lind, L. Chiapponi & M. Gómez-Gesteira

Sections | Figures | References

Abstract

ELSEVIER

Computers & Fluids

Volume 303, 15 December 2025, 106870

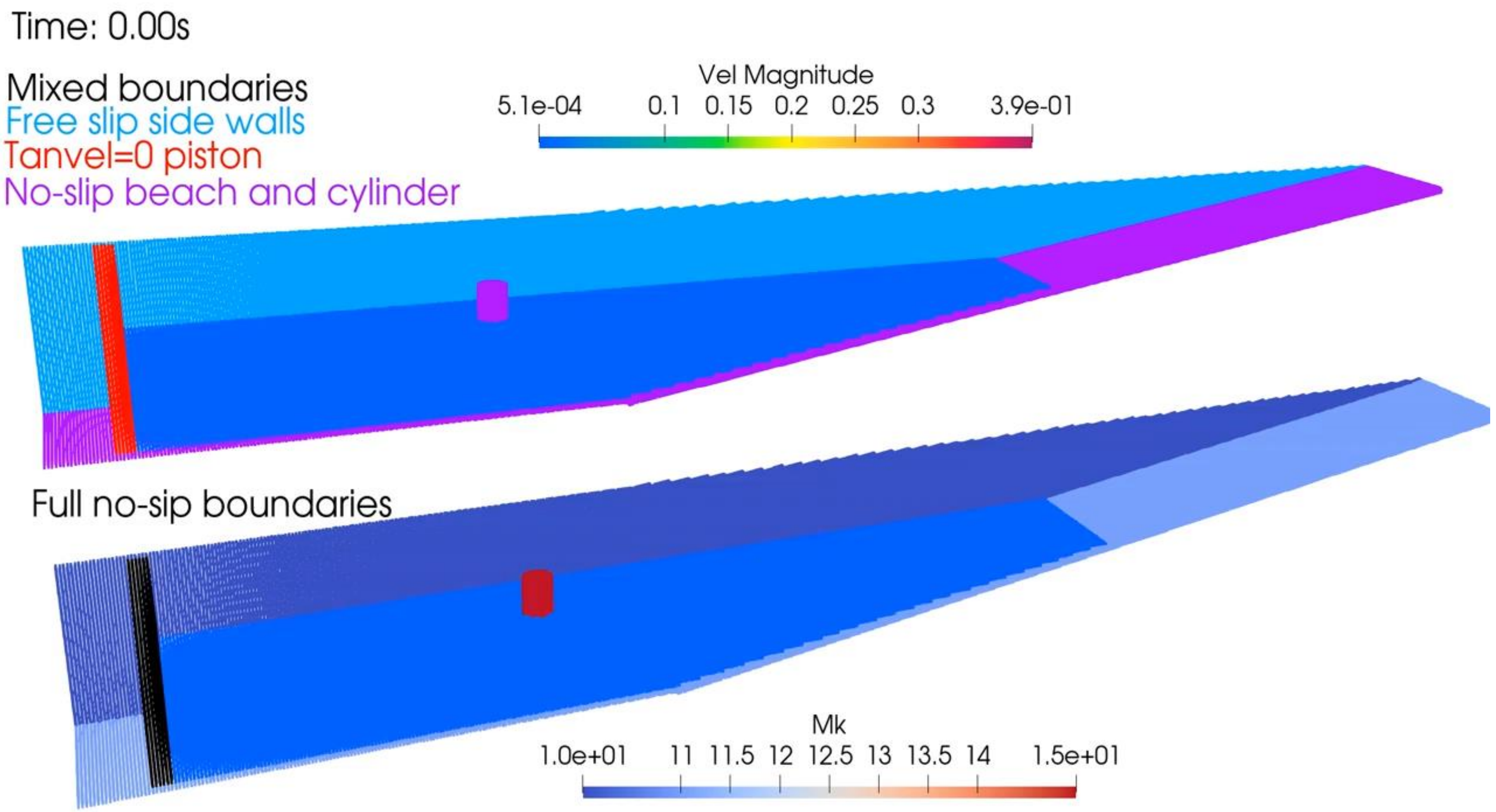
Smoothed particle hydrodynamics modelling of river flows past bridges

Aaron English [a], Renato Vacondio [a], Susanna Dazzi [a], José M. Domínguez [b]

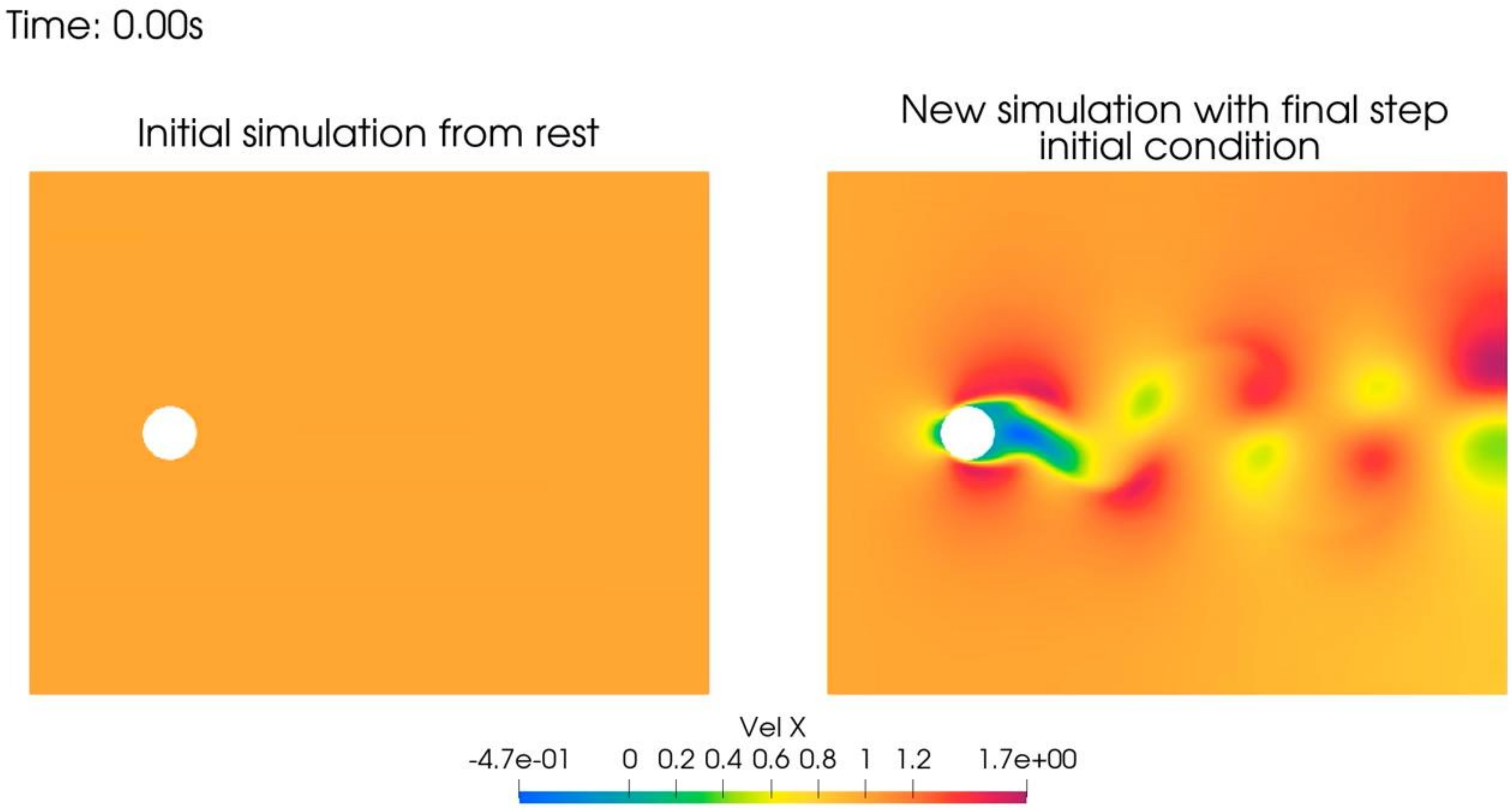
Show more

+ Add to Mendeley Share Cite

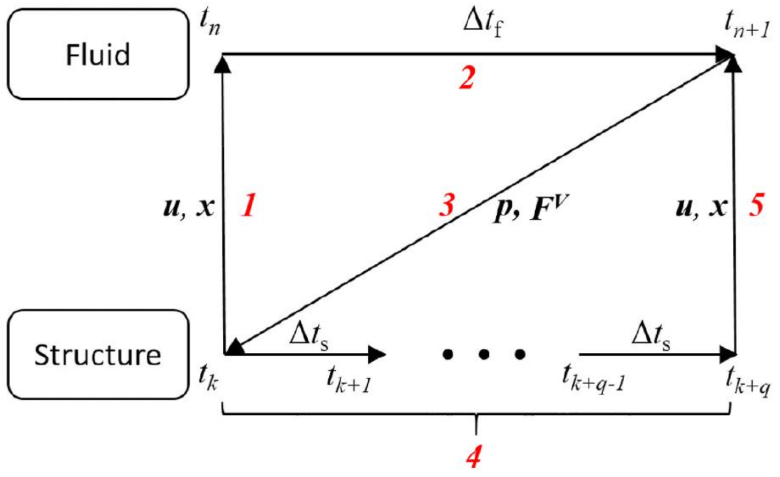
Mixed boundary conditions



Defining initial conditions from previous simulations



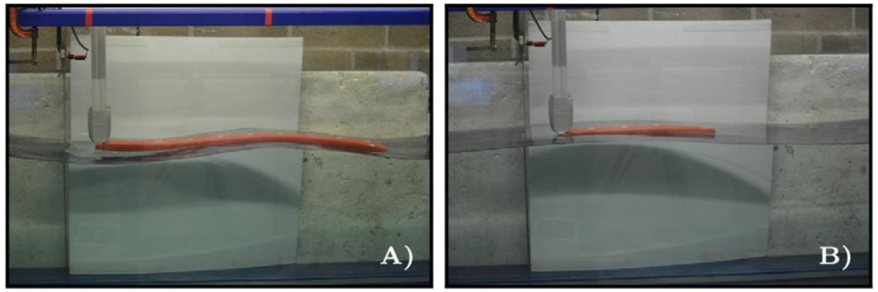
FSI Staggered Timestep - TLSPH



1. compute $q = \Delta t_f / \Delta t_s$
2. fluid solvers advances one time step Δt_f
3. structure provides boundary condition
4. structure solver advances q time step Δt_s

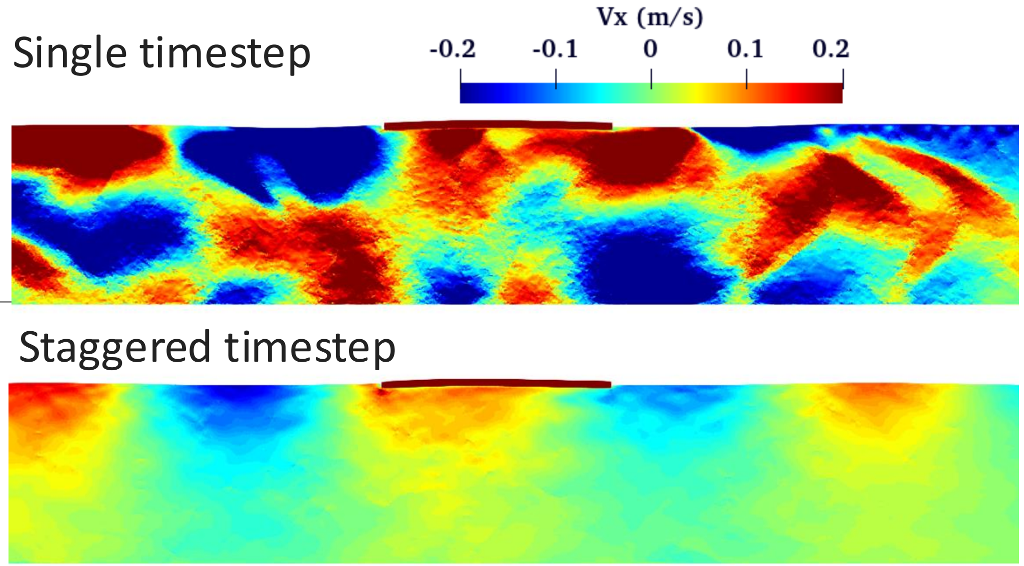
   (fluid provides hydro pressure and viscous force)

Floating Flexible Structure



ρ = 412 kg/m^3; E = 0.565 Mpa; ν = 0.49

C_f = 27 m/s; C_s = 153 m/s → $dt_f \approx$ 5e-5; $dt_s \approx$ 6e-5



[1] Sun et al. "An accurate FSI-SPH modeling of challenging fluid-structure interaction problems in two and three dimensions." *OE* (2021)
[2] Brown et al. "Investigation of wave-driven hydroelastic interactions using numerical and physical modelling approaches." *APOR* (2022)

How to download DualSPHysics v6.0 beta

cpu gpu

DualSPHysics

Version 6.0 beta

New version only available to workshop attendees!!

The full package v6.0 includes:

- Several SPH approaches
- New pre- and post-processing tools
- Improved documentation guides
- More than 130 examples (including new features)

Download the full package (*DualSPHysics_v6.0_BETA.zip*) from:
https://dual.sphysics.org/sphcourse/DualSPHysics_v6.0_BETA/

User: **XXX**
Password: **XXX**

J.M. Domínguez, G. Fourtakas, C. Altomare, R.B. Canelas, A. Tafuni, O. García-Feal, I. Martínez-Estévez, A. Mokos, R. Vacondio, A.J.C. Crespo, B.D. Rogers, P.K. Stansby, M. Gómez-Gesteira. 2022. **DualSPHysics: from fluid dynamics to multiphysics problems.** Computational Particle Mechanics. 9(5): 867-895. doi:10.1007/s40571-021-00404-2