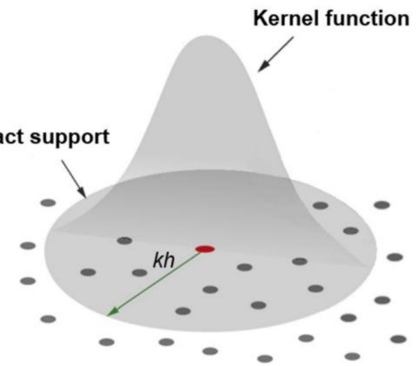
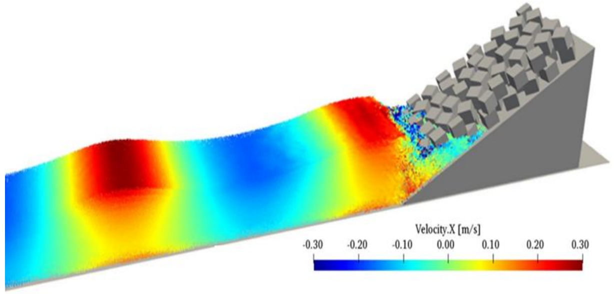
Novelties on DualSPHysics v5.4

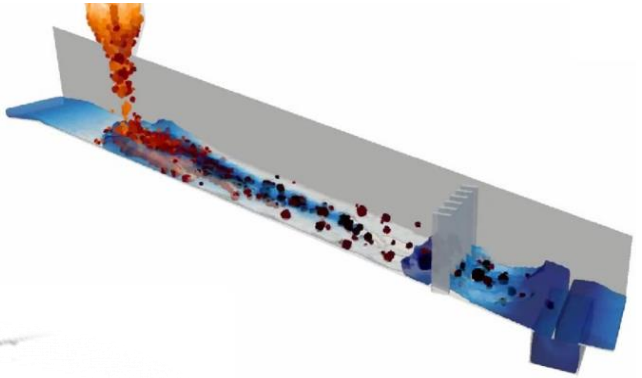
J.M. DOMÍNGUEZ

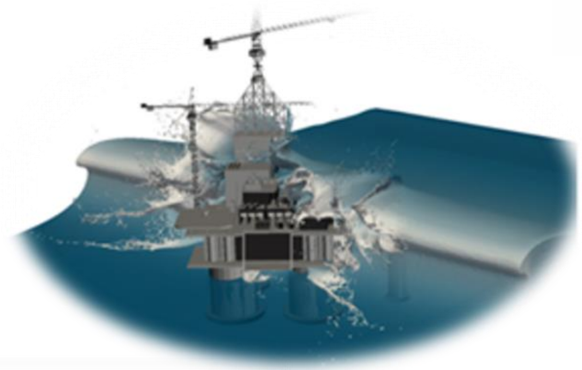
Overview of DualSPHysics

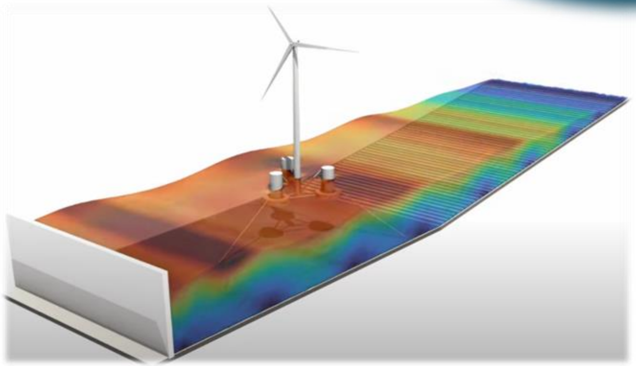
Implementation of **Smoothed Particle Hydronynamics** method for **complex fluid dynamics** using HPC techniques

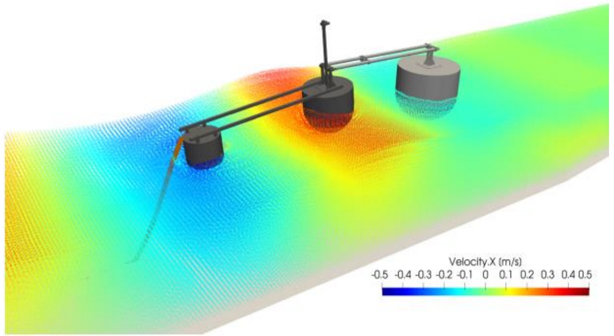












Overview of DualSPHysics

cpu gpu
DualSPHysics

It includes **two implementations**:

- **CPU**: C++ and OpenMP.
- **GPU**: CUDA.

Both options optimized for the best performance of each architecture.

SPH HIGHLY PARALLELISED

GPU	CPU
	
NVIDIA CUDA	C++

GPU ⟶ CPU
x100

Overview of DualSPHysics

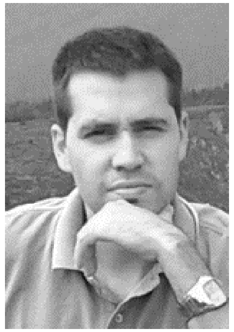
DUALSPHYSICS TEAM

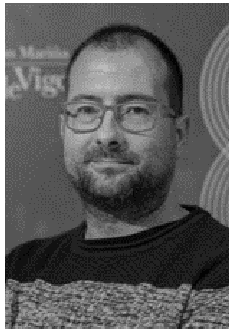
Project Leaders:
- Dr José M. Domínguez. Universidade de Vigo, Spain
- Dr Georgios Fourtakas. The University of Manchester, UK
- Prof. Alejandro J.C. Crespo. Universidade de Vigo, Spain
- Prof. Benedict D. Rogers. The University of Manchester, UK
- Dr Renato Vacondio. Università degli studi di Parma,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

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

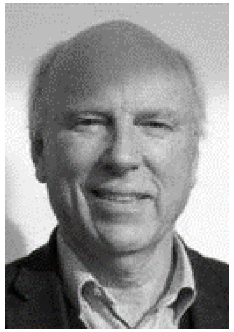
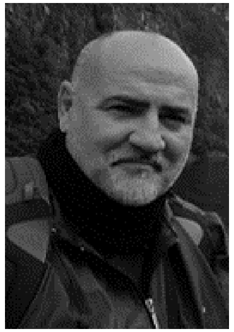
Wiki Coordinator:
- Dr Bonaventura Tagliafierro. Universitat Politècnica de Catalunya – BarcelonaTech, Spain

Project Coordinators:
- Dr Orlando García Feal. Universidade de Vigo, Spain
- Dr Joseph O'Connor. The University of Edinburgh, UK
- Dr Aaron English. The University of Manchester, UK
- Iván Martínez Estévez. Universidade de Vigo, Spain
- Dr Francesco Ricci. New Jersey Institute of Technology, US







Overview of DualSPHysics

DUALSPHYSICS TEAM

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

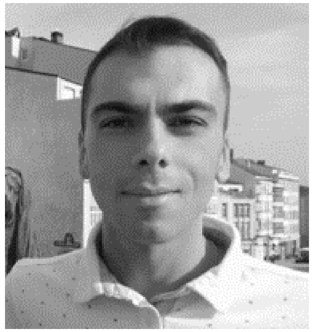
Project Coordinators:
•Prof. Peter Stansby. The University of Manchester, UK
•Prof. Moncho Gómez Gesteira. Universidade de Vigo, Spain

Wiki Coordinator:
•Dr Bonaventura Tagliafierro. Universitat Politècnica de Catalunya – BarcelonaTech, Spain

Project Coordinators:
•Dr Orlando García Feal. Universidade de Vigo, Spain
•Dr Joseph O'Connor. The University of Edinburgh, UK
•Dr Aaron English. The University of Manchester, UK
•Iván Martínez Estévez. Universidade de Vigo, Spain
•Dr Francesco Ricci. New Jersey Institute of Technology, US

Latest incorporations to DualSPHysics team


Bonaventura


Iván

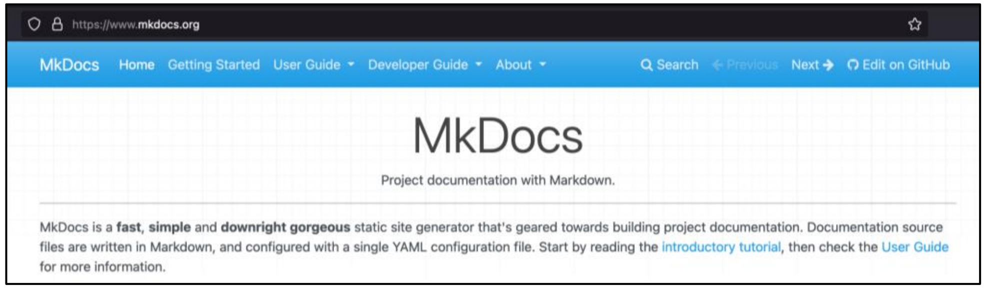

Aaron


Francesco

Overview of DualSPHysics

New Wiki under development

Wiki Coordinator: Dr Bonaventura Tagliafierro

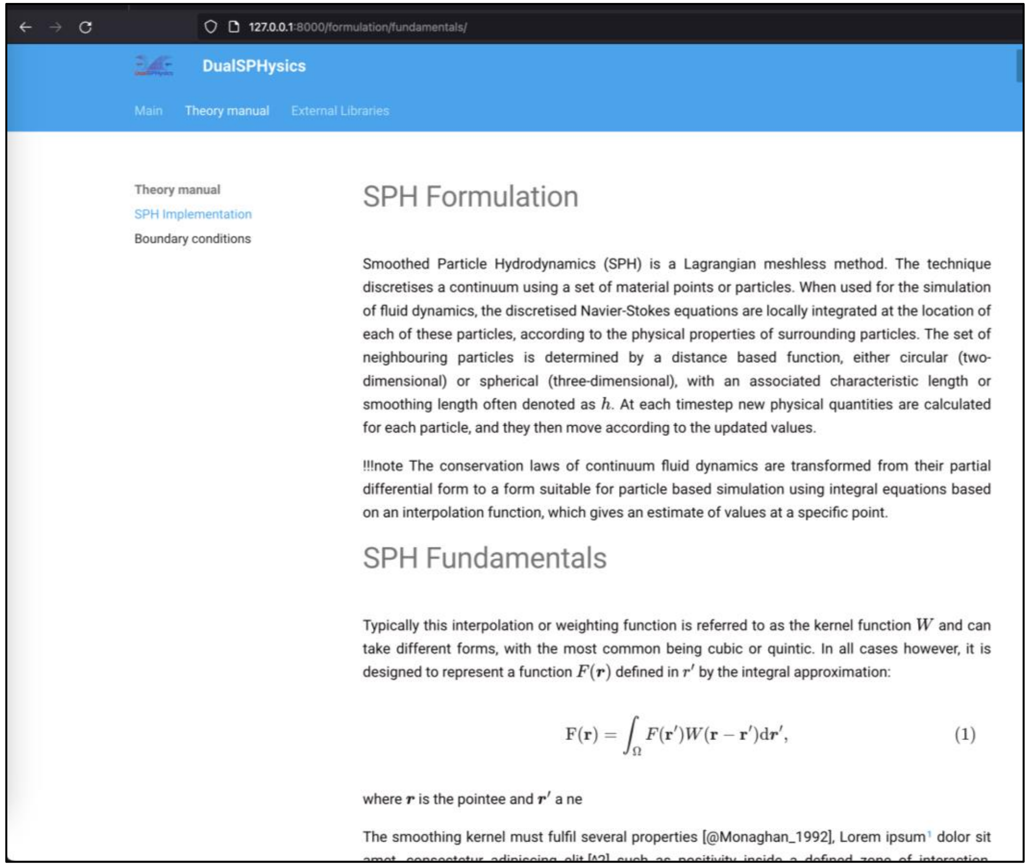


MkDocs License (BSD)

Copyright © 2014, Tom Christie. All rights reserved.

https://www.mkdocs.org/about/license

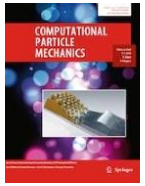
- Very easy to customize
- Easy to maintain
- Easy to port
- Markdown for input source files
- It can build either websites or doc files (limited for now)
- Support for html as well



The new wiki will contain

- Structured content
- Provide walkthrough for template cases
- Anyone can maintain their page (or section)

Overview of DualSPHysics



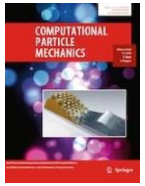
J.M. Domínguez, G. Fourtakas, C. Altomare, R.B. Canelas, A. Tafuni, O. García-Feal, I. Martínez-Estévez, A. Mokos, R. Vacondio, A.J.C. Crespo, B.D. Rogers, P.K. Stansby, M. Gómez-Gesteira. 2022. **DualSPHysics: from fluid dynamics to multiphysics problems.** Computational Particle Mechanics. 9(5): 867-895.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

Formulation and capabilities

- SPH approaches:
  - **Single phase free-surface flow solver**
  - Multi-phase: liquid and gas
  - Multi-phase: non-Newtonian flows
- Kernel functions:
  - Cubic Spline (Monaghan and Lattanzio, 1985)
  - Quintic Wendland (Wendland, 1995)
- Density diffusion Term:
  - Molteni (Molteni and Colagrossi, 2009)
  - Fourtakas (Fourtakas et al., 2019)
- Viscosity:
  - Artificial (Monaghan, 1992)
  - **Laminar (Lo & Shao, 2002)**
  - Laminar + SPS turbulence model (Dalrymple & Rogers, 2006)
- Weakly compressible approach using Tait's equation of state (Batchelor, 1974)
- Time integration scheme:
  - Verlet (Verlet, 1967)
  - Symplectic (Leimkhuler, 1996)
- Variable time step (Monaghan and Kos, 1999)
- Shifting algorithm (Lind et al., 2012)
- Boundary conditions:
  - Dynamic boundary conditions (Crespo et al., 2007)
  - Modified Dynamic boundary conditions (English et al., 2022)
  - **No-Slip Modified Dynamic boundary conditions**
- Periodic open boundaries (Gómez-Gesteira et al., 2012)
- Inflow-outflow boundary conditions (Tafuni et al., 2018)
- **MESH-IN: Meshed inlet offline coupling (Ruffini et al., 2023)**
- External forces (Longshaw and Rogers, 2015)

Overview of DualSPHysics

Formulation and capabilities



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

- Floating objects (Monaghan et al., 2003)
- Floating objects with mDBC support

- **Full-integrated lagrangian formulation for flexible fluid-structure interaction (O'Connor et al., 2021)**

- Coupling with Discrete Element Method (Canelas et al., 2016)
- Coupling with Project Chrono (Canelas et al., 2018; Martínez-Estévez et al., 2022)
- Coupling with MoorDyn (Domínguez et al., 2019)

- Piston- and flap-type long-crested second-order wave generation (Altomare et al., 2017)
- Solitary waves (Domínguez et al., 2019)
- Focused waves (Whittaker et al., 2017)

- Passive and Active Wave Absorption System (Altomare et al., 2017)

- Relaxation Zone method and coupling with wave propagation models (Altomare et al., 2018)

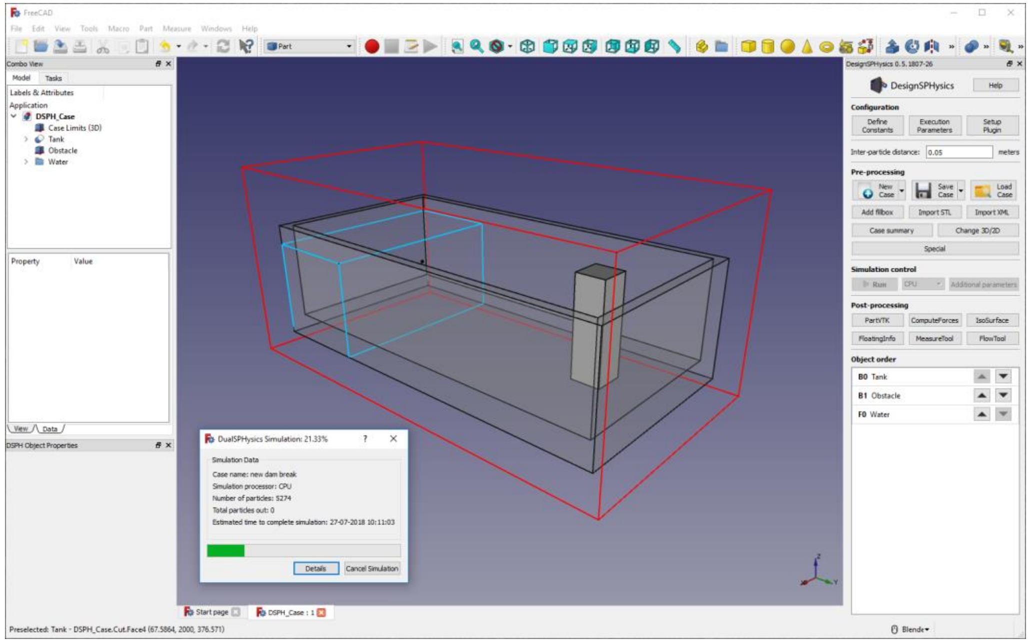
- Non-linear wave generation and absorption using open boundaries (Verbrugghe et al., 2019)

- **Variable resolution based on domain decomposition (Ricci et al., 2024)** ***BETA version***

Overview of DualSPHysics

DesignSPHysics
(Graphical User Interface)

FreeCAD
Open Source parametric 3D CAD modeler



COMPLETE TOOLKIT

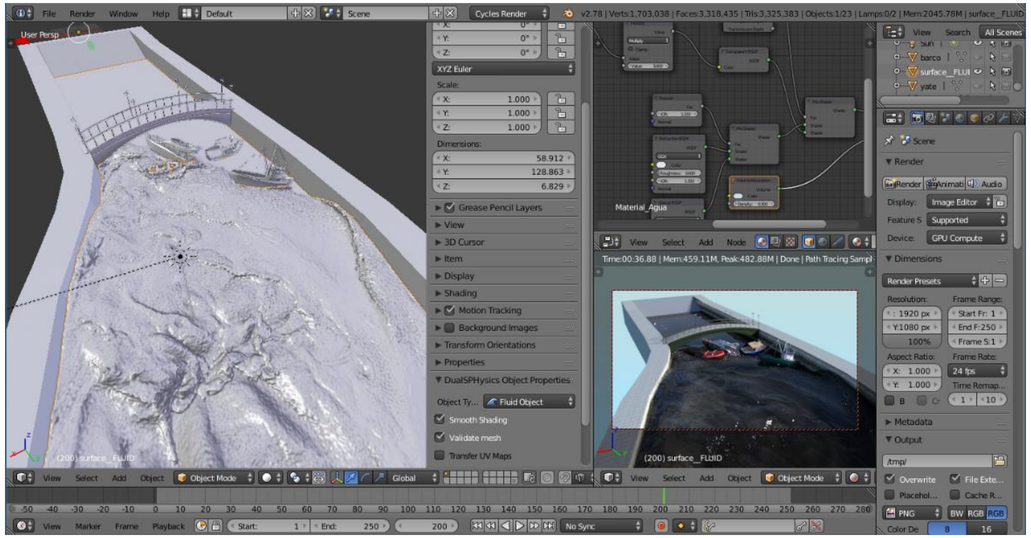
- SPH solver
- Pre-processing tools
- Post-processing tools

but also…

- Graphical User Interface
- Advanced visualisation

VisualSPHysics
(Advanced visualisation tool)

blender™



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: 14,210 (73% Windows)

DualSPHysics v4.0 (2016)
Downloads: 13,150 (72% Windows)

DualSPHysics v4.2 (May 2018)
Downloads: 8,076

DualSPHysics v4.4 (April 2019)
Downloads: 21,921

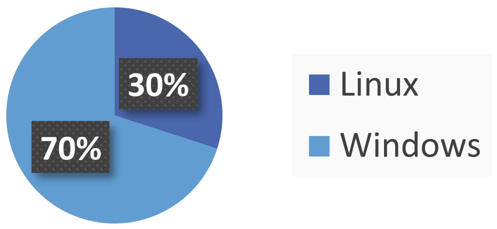
DualSPHysics v5.0 (July 2020)
Downloads: 61,240

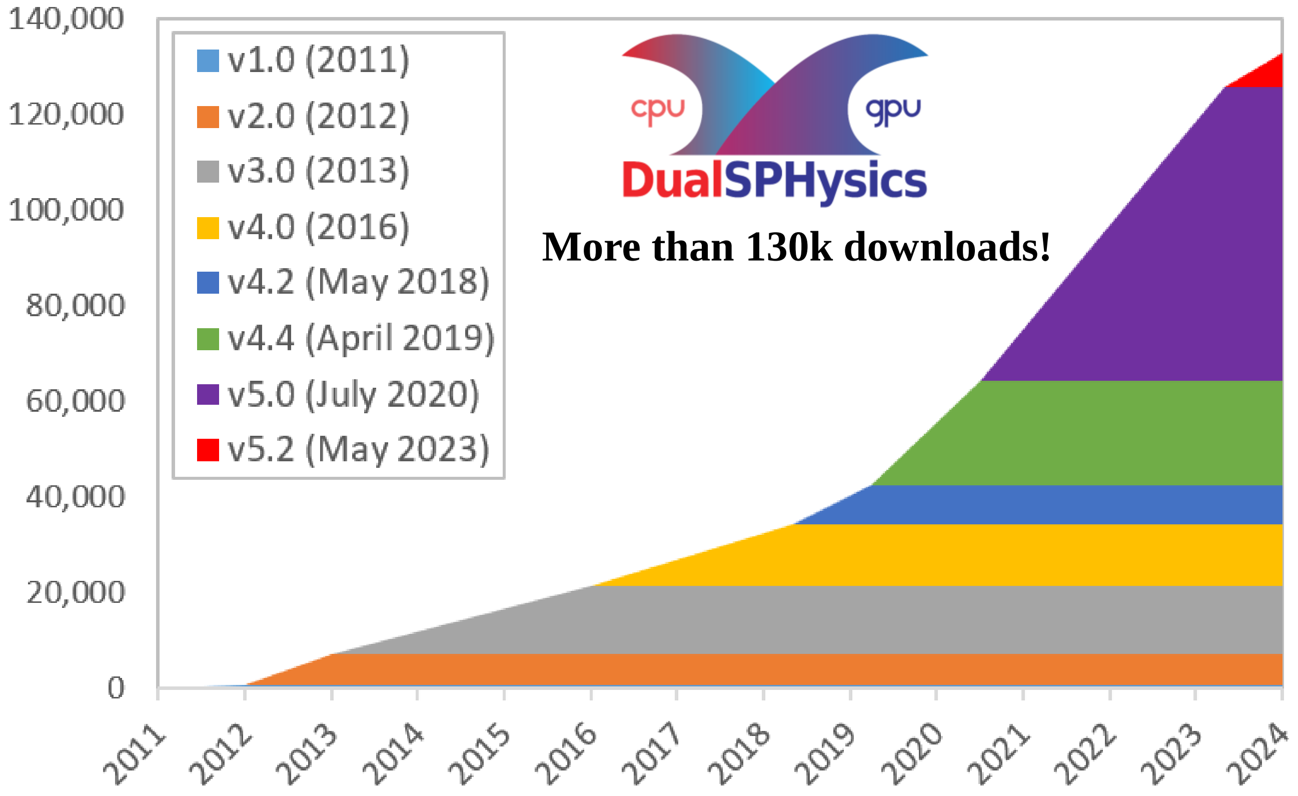
DualSPHysics v5.2 (May 2023)
Downloads: 8,163

DualSPHysics - ALL VERSIONS

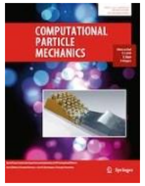
Downloads: 133,933 (70% Windows)

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





Main novelties in v5.4



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

SPH solver novelties:

- New **variable resolution** method with binaries and test cases
- New **mDBC no-slip** option
- Major improvement in inlet-outlet by using mesh data (**MESH-IN**)
- Full-integrated lagrangian formulation for flexible fluid-structure interaction.
- Laminar viscosity option without SPS turbulence model (better and faster results).
- Filter options for output particle data (space saving)
- Improved implementation for particle arrays on CPU and GPU
- New simpler and faster implementation for few floating bodies

Pre-processing novelties (*GenCase*):

- Supports new XML-VTK formats automatically (VTK,VTP, VTU)
- Supports new VTM file format with filters according to index or surface name
- External geometries with a huge number of triangles are supported
- Major performance and memory usage improvements on advanced drawing mode
- Significant performance improvements on automatic normal calculation
- Supports output files larger than 4 GB to generate hundreds of millions of particles

Post-processing novelties:

- Much faster motion processing without data loading (*FloatingInfo* & *BoundaryVtk*)
- More floating body information on fluid forces and external forces (*FloatingInfo*)
- Higher frequency output on moving and floating bodies (*BoundaryVtk* & *FloatingInfo*)
- Moment force calculation on intrinsic and extrinsic axis (*ComputeForces*)
- Improved area selection for flow calculation by XML file with many options (*FlowTool*).

Main novelties in v5.4

F. Ricci, R. Vacondio, A. Tafuni. 2024. **Multiscale Smoothed Particle Hydrodynamics based on a domain-decomposition strategy.** Comput. Methods Appl. Mech. Engrg. 418: 116500. doi:10.1016/j.cma.2023.116500

Variable resolution in DualSPHysics (Ricci et al., 2024)

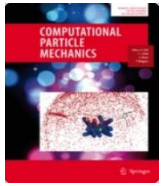
- New method based on domain decomposition.
- Allows you to define higher resolution areas within the simulation.
- Very useful to reduce the number of particles and execution time.
- Enables detailed simulations previously impossible without a supercomputer.
- Includes several interesting capabilities...
  See Ricci's talk

v5.4 includes binaries and test cases

3 resolution levels (from 4 to 1 mm) requires 1M particles and 3 hours **vs.** using **uniform resolution** (1 mm) requires 7M particles and 9 hours

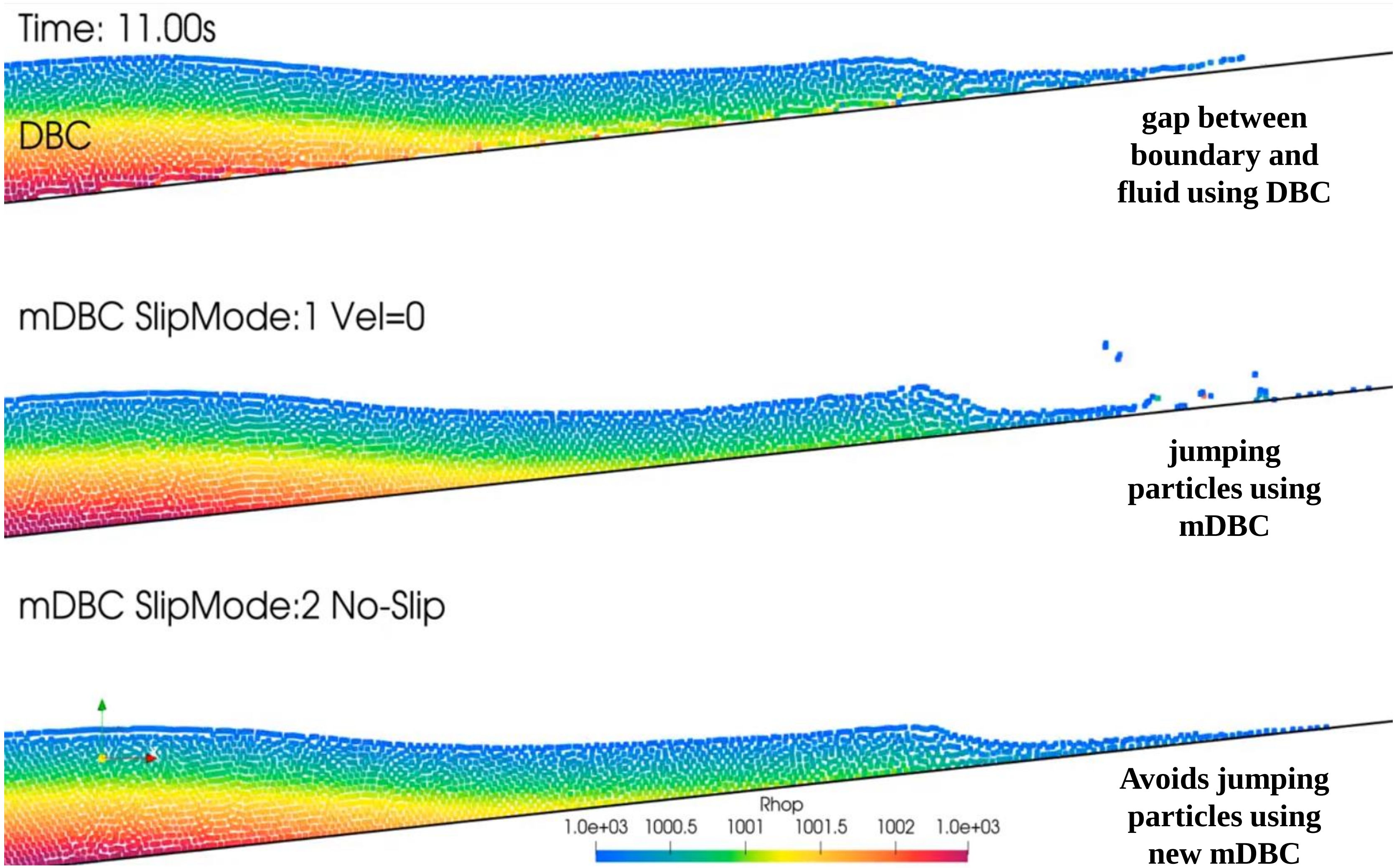
Main novelties in v5.4

New mDBC no-slip option



A. English, J.M. Domínguez, R. Vacondio, A.J.C. Crespo, P.K. Stansby, S.J. Lind, L. Chiapponi, M. Gómez-Gesteira. 2022. **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, 9(5): 911-925. doi:10.1007/s40571-021-00403-3

- mDBC provides better results with lower resolution.
- mDBC setup is more complicated than DBC and present some issues.
- New formulation avoids particle jumping when kernel support is not complete.
- The new no-slip option is essential for the accurate modelling of some applications.
- More details in **English's talk.**



gap between boundary and fluid using DBC

jumping particles using mDBC

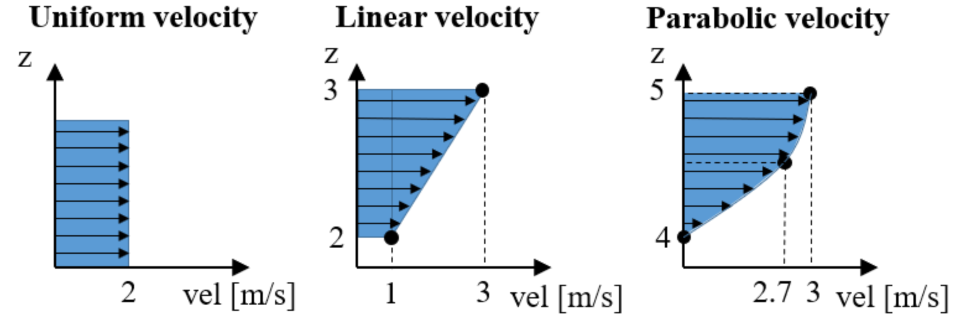
Avoids jumping particles using new mDBC

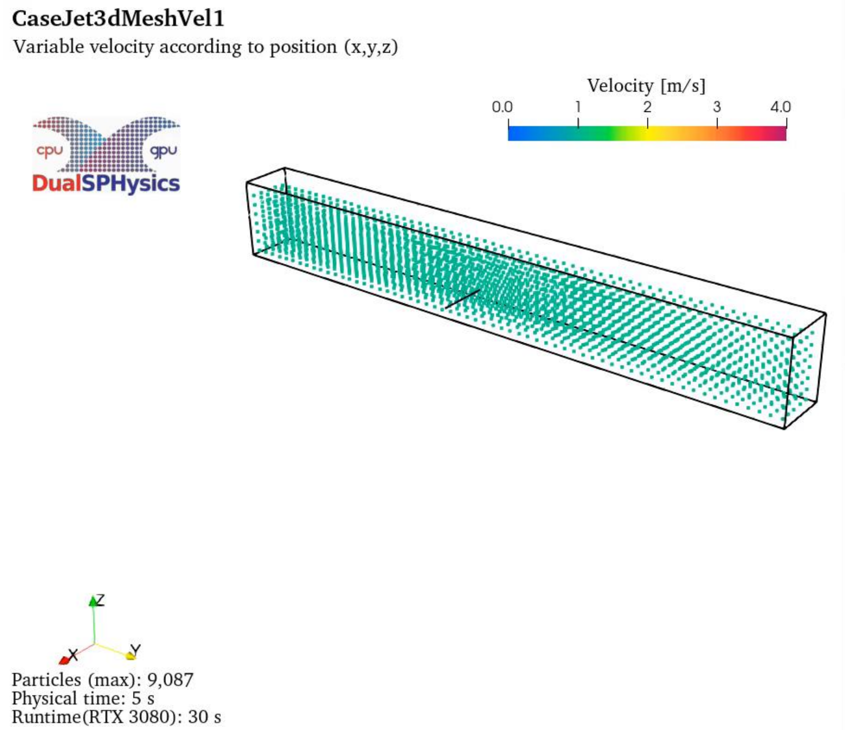
Main novelties in v5.4

Inlet condition by using mesh data (MESH-IN)

- Full control of velocity (magnitude and direction) throughout the inlet area.
- Full control of elevation throughout the inlet area.

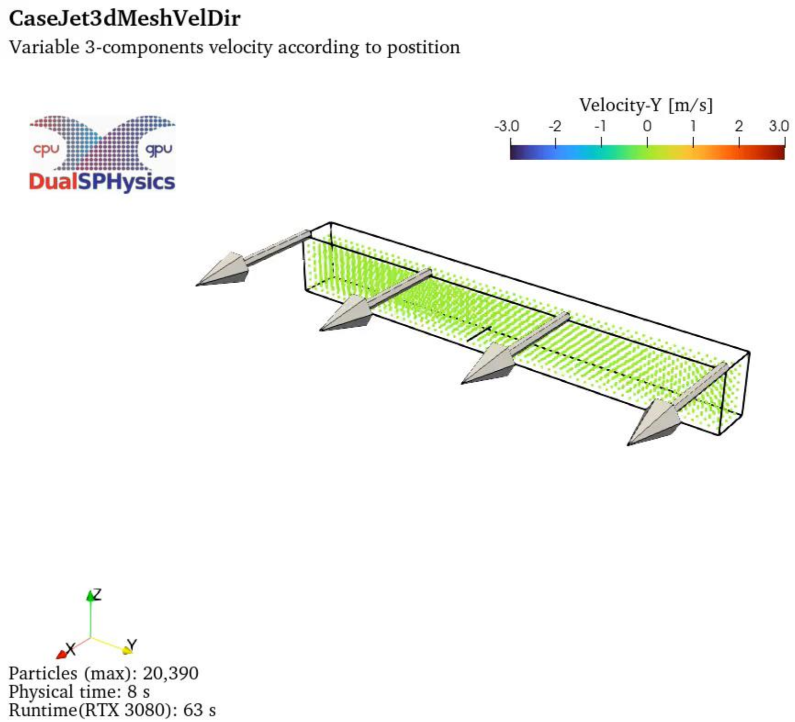
Inlet velocity profiles in **v5.2**





Inlet in **v5.4** enables:

- Full custom velocity profile
- Full custom velocity direction.
- Fixed or variable in time.



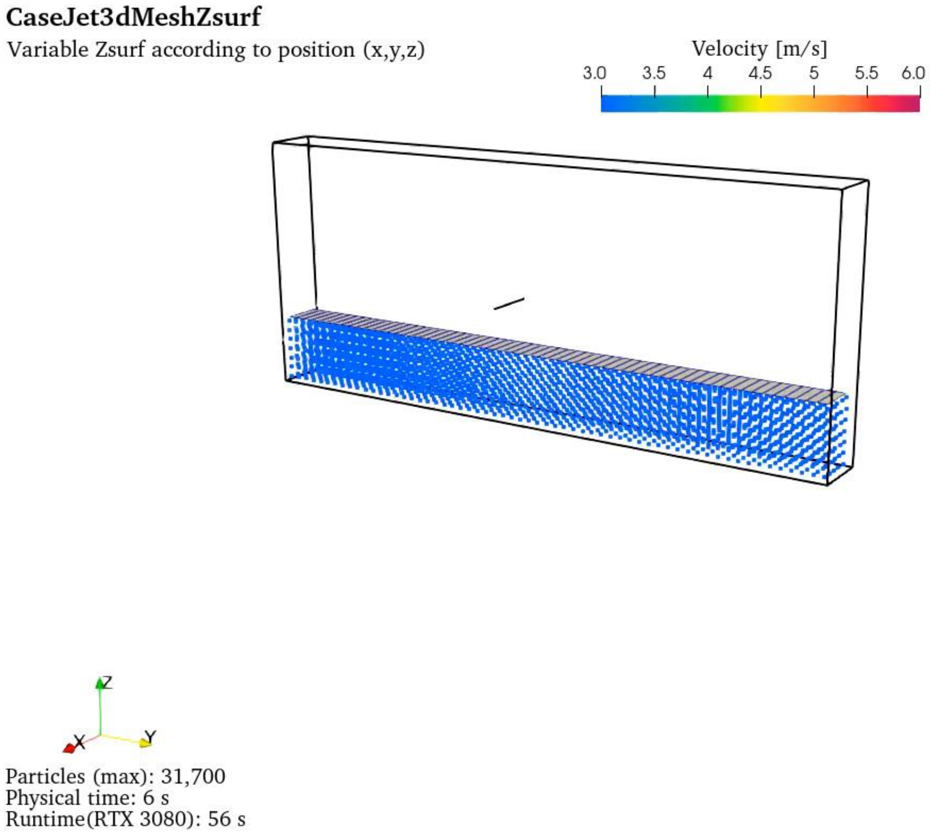
Examples available at
`examples\inletmesh\01_Basic`

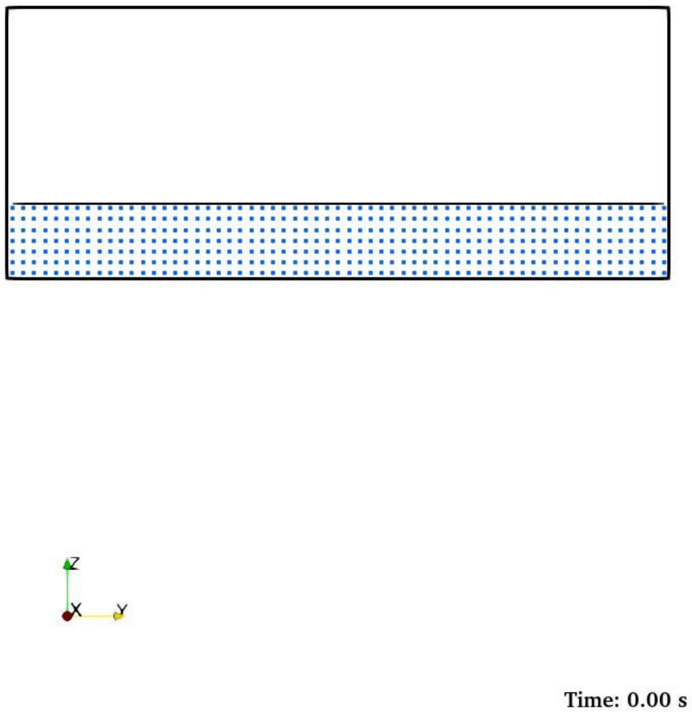
Main novelties in v5.4

Inlet condition by using mesh data (MESH-IN)

- Full control of velocity (magnitude and direction) throughout the inlet area.
- Full control of elevation throughout the inlet area.

Example available at
`examples\inletmesh\01_Basic`

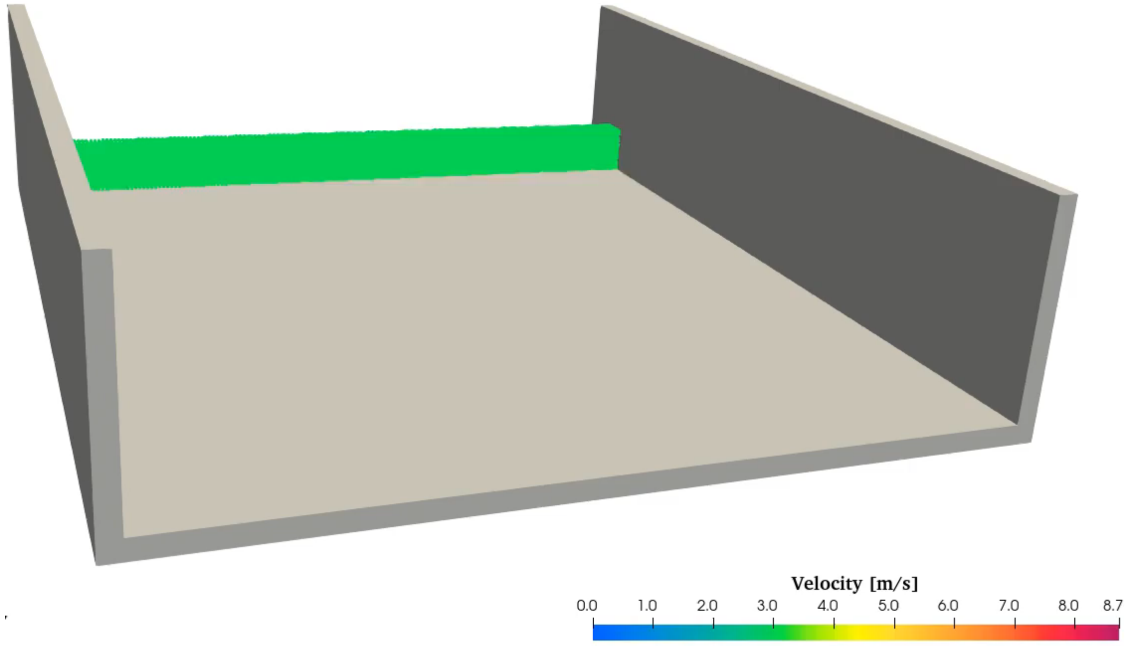




Main novelties in v5.4

Inlet condition by using mesh data (MESH-IN)

- Full control of velocity (magnitude and direction) throughout the inlet area.
- Full control of elevation throughout the inlet area.
- The velocity and elevation values are defined in a grid by using a simple CSV file.
- The interpolated value from the data grid is applied at each inlet position.



Complex inlet changing velocity and height according to mesh data in two CSV files generated by Python scripts in a simple way (available at `examples\inletmesh\02_Complex`).

	A	B	C	D	E	F	G	H	I	J	K	L	M	N
1	Format	DataName	DataUnits	DataType	Data12	Npt1	Npt2	Npt3	PtRef.x [m]	PtRef.y	PtRef.z	Vec1.x [m]	Vec1.y	Vec1.z
2	JMeshTData:	Vel	[m/s]	float3	true	4	1	53	0	0	0	0	1	0
3	time [s]	v(0:0).x	v(0:0).y	v(0:0).z	v(1:0).x	v(1:0).y	v(1:0).z	v(2:0).x	v(2:0).y	v(2:0).z	v(3:0).x	v(3:0).y	v(3:0).z	
4	0	4	0	0	4	0	0	4	0	0	4	0	0	
5	1	4	0	0	4	0	0	4	0	0	4	0	0	
6	1.5	2.828	2.828	0	2.828	2.828	0	2.828	2.828	0	2.828	2.828	0	
7	2	2.309	2.309	2.309	2.309	2.309	2.309	2.309	2.309	2.309	2.309	2.309	2.309	
8	3	2.309	2.309	2.309	2.309	2.309	2.309	2.309	2.309	2.309	2.309	2.309	2.309	
9	3.5	2.309	-2.309	2.309	2.309	-2.309	2.309	2.309	-2.309	2.309	2.309	-2.309	2.309	
10	4	2.309	2.309	2.309	2.309	2.309	2.309	2.309	2.309	2.309	2.309	2.309	2.309	
11	4.5	2.309	-2.309	2.309	2.309	-2.309	2.309	2.309	-2.309	2.309	2.309	-2.309	2.309	
12	5	4	0	0	4	0	0	4	0	0	4	0	0	
13	5.5	4	0	0	4	0	0	4	0	0	4	0	0	
14	6	2.309	-2.309	2.309	2.309	-2.309	2.309	2.309	2.309	2.309	2.309	2.309	2.309	
15														

CSV file with 3-component velocity in 12 grid points (available at `examples\inletmesh\01_Basic`)

Main novelties in v5.4

Inlet condition by using mesh data (MESH-IN)

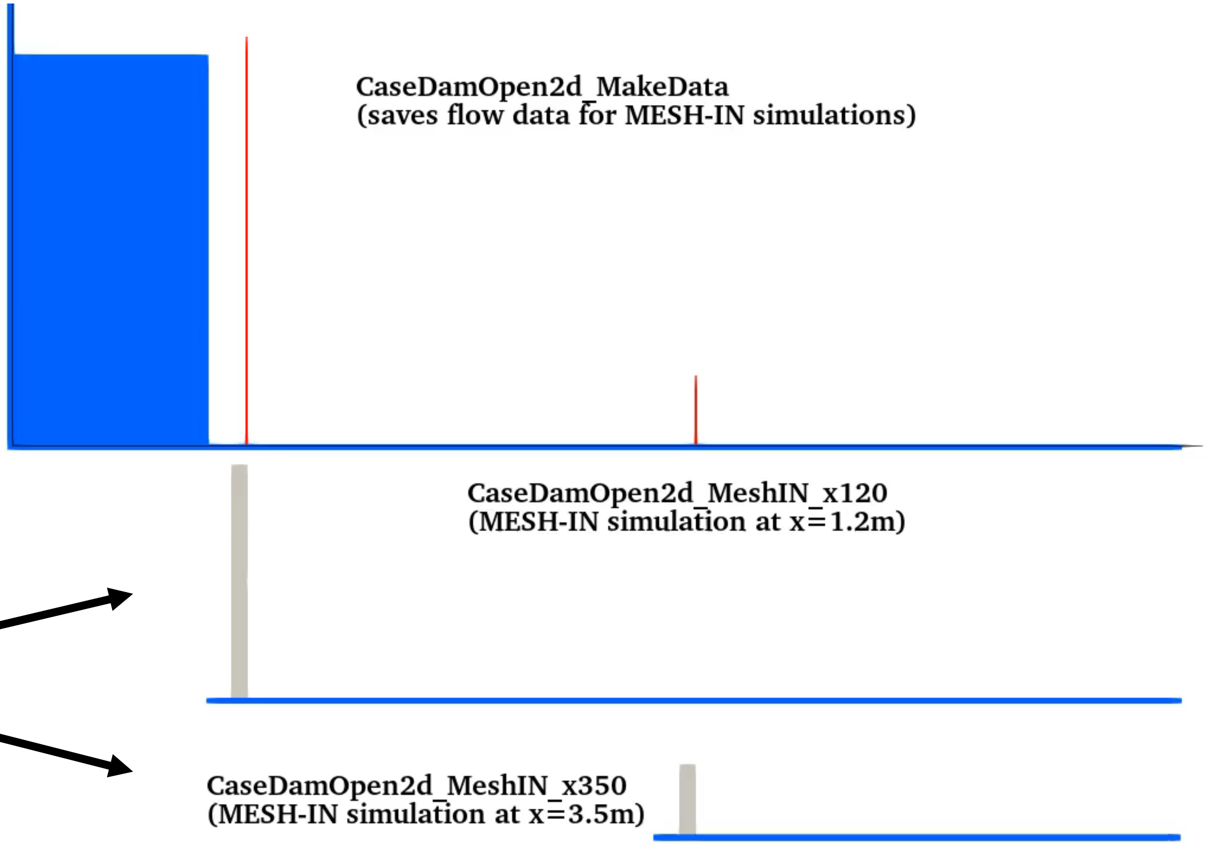
G. Ruffini, J.M. Domínguez, R. Briganti, C. Altomare, J. Stolle, A.J.C. Crespo, B. Ghiassi, S. Capasso, P. Girolamo. 2023. **MESH-IN: A MESHed INlet offline coupling method for 3-D extreme hydrodynamic events in DualSPHysics.** Ocean Engineering, 268: 113400. doi:10.1016/j.oceaneng.2022.113400

- Inlet input data can be generated manually or obtained from a simulation.
- The gauge system in DualSPHysiscs allows to record the velocity field and the fluid elevation in an area.
- Mesh data is stored in CSV or binary format (.mbi4).

Example available at
`examples\inletmesh\03_Dambreak`

Full simulation field velocity and elevation (at red lines) generating a mesh data file (.mbi4)

Reduced inlet simulation uses mesh data file to reproduce flow properties.

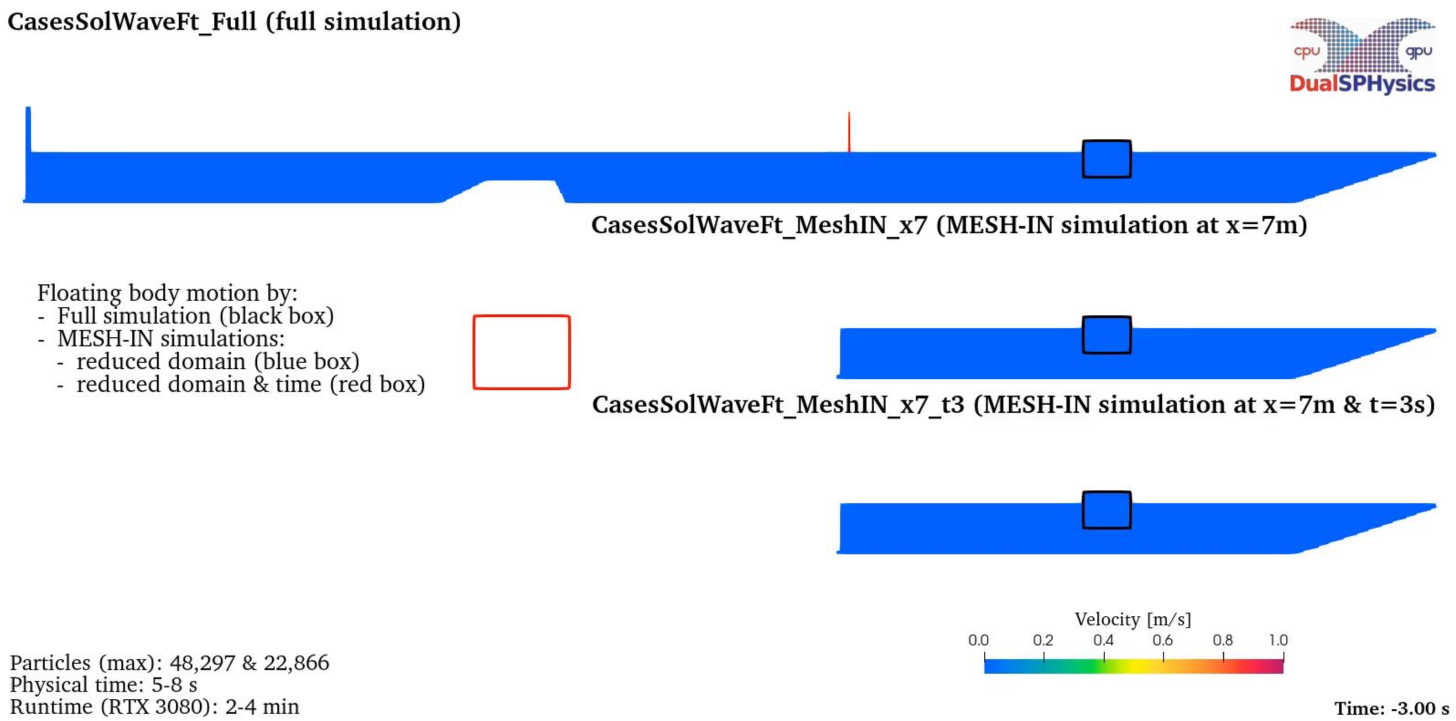


Main novelties in v5.4

G. Ruffini, J.M. Domínguez, R. Briganti, C. Altomare, J. Stolle, A.J.C. Crespo, B. Ghiassi, S. Capasso, P. Girolamo. 2023. **MESH-IN: A MESHed INlet offline coupling method for 3-D extreme hydrodynamic events in DualSPHysics.** Ocean Engineering, 268: 113400. doi:10.1016/j.oceaneng.2022.113400

Inlet condition by using mesh data (MESH-IN)

- 2-D or 3-D flow data can be obtained from a simplified simulation or from another source.
- Flow data can be used in:
  - New simulations with different setups.
  - The duration of the simulation can be reduced to the interval of interest.
  - The simulation domain can be reduced to the area of interest.
  - The resolution can be changed and 2-D data can be used for 3-D simulations.



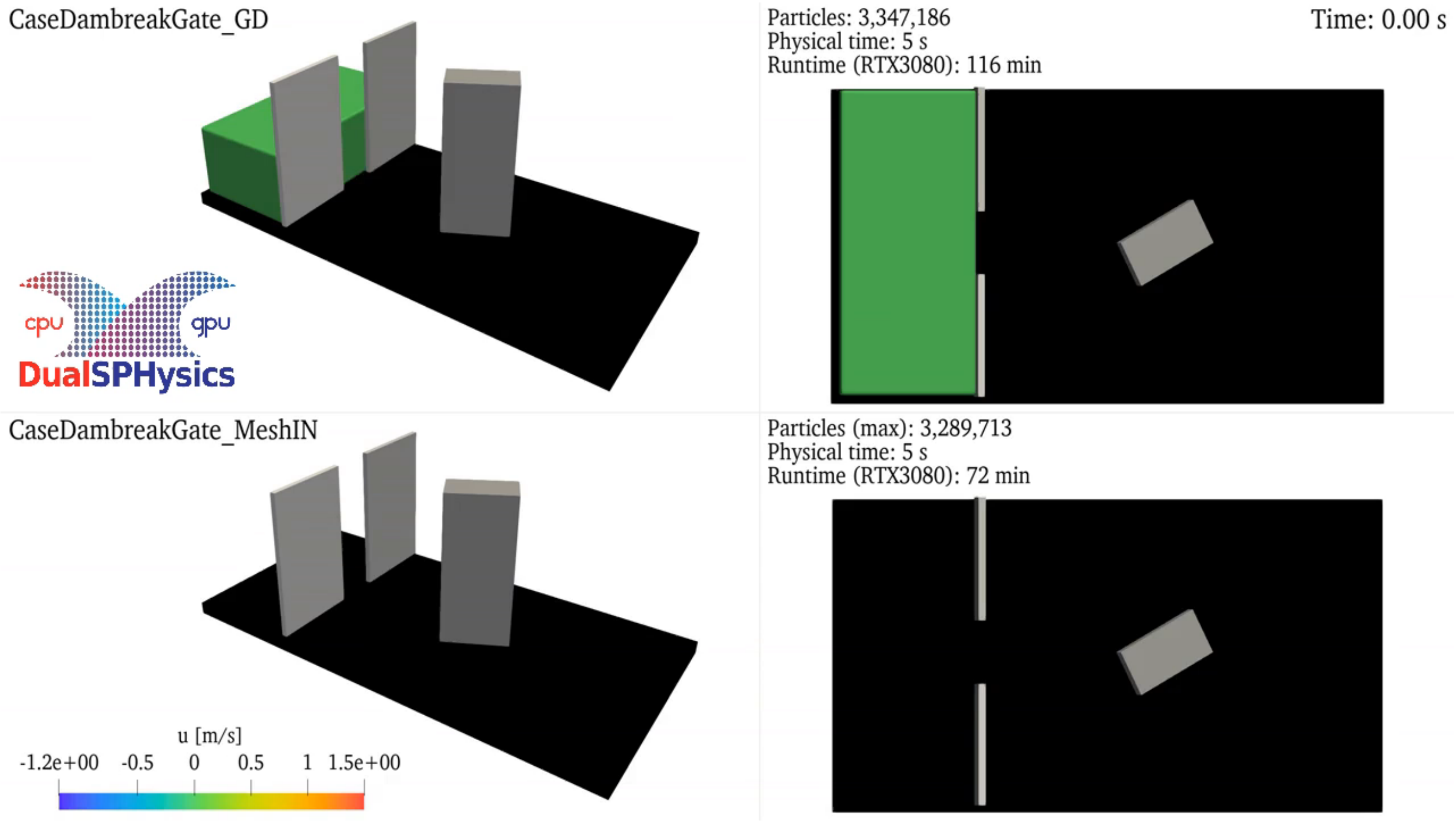
Solitary wave and floating body example available at
`examples\inletmesh\04_SolitaryWaveFt`

Main novelties in v5.4

G. Ruffini, J.M. Domínguez, R. Briganti, C. Altomare, J. Stolle, A.J.C. Crespo, B. Ghiassi, S. Capasso, P. Girolamo. 2023. **MESH-IN: A MESHed INlet offline coupling method for 3-D extreme hydrodynamic events in DualSPHysics.** Ocean Engineering, 268: 113400. doi:10.1016/j.oceaneng.2022.113400

Inlet condition by using mesh data (MESH-IN)

- 2-D or 3-D flow data can be obtained from a simplified simulation or from another source.
- Flow data can be used in:
  - New simulations with different setups.
  - The duration of the simulation can be reduced to the interval of interest.
  - The simulation domain can be reduced to the area of interest.
  - The resolution can be changed and 2-D data can be used for 3-D simulations.



3-D validation of MESH-IN (Ruffini et al., 2023)

3-D validation of MESH-IN available at
`examples\inletmesh\05_DambreakGate`

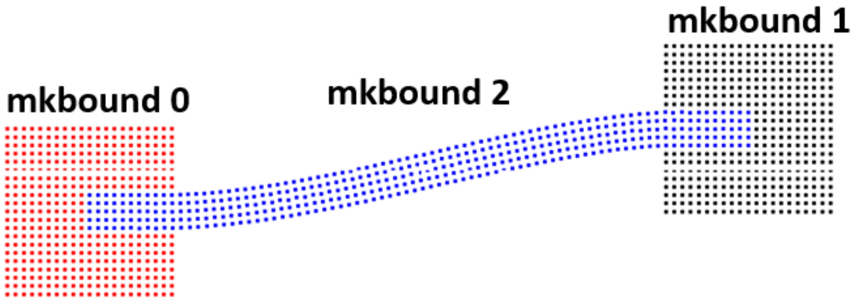
Main novelties in v5.4

J. O'Connor, B.D. Rogers. 2021. **A fluid-structure interaction model for free-surface flows and flexible structures using smoothed particle hydrodynamics on a GPU.** Journal of Fluids and Structures, 104. doi:10.1016/j.jfluidstructs.2021.103312

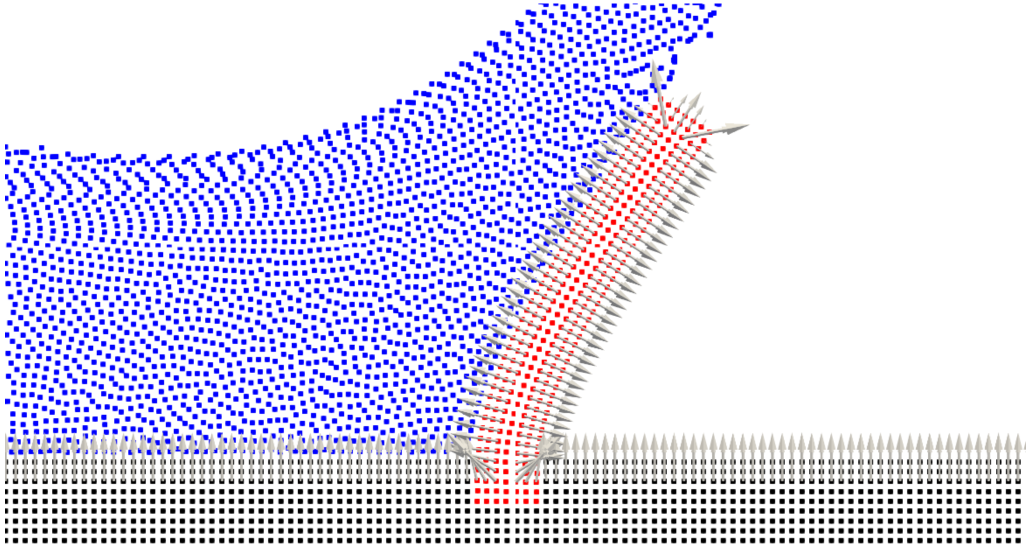
Improvements to Flexible FSI (FlexStruc)

- Full-integrated FlexStruc code in DualSPHysics v5.4.
- Improved error checking when specifying Poisson ratio for different constitutive models.
- Improved specification of zero, single or multiple mkclamp objects.

```
<flexstrucbody mkbound="2" mkclamp="0,1">
```



- Initial implementation with mDBC.
    - Currently disabled by default as still in development
    - Can enable by uncommenting **DISABLE_FLEXSTRUC_MDBC** in *DualSphDef.h*
    - Use at own risk! (issues with particle penetration)

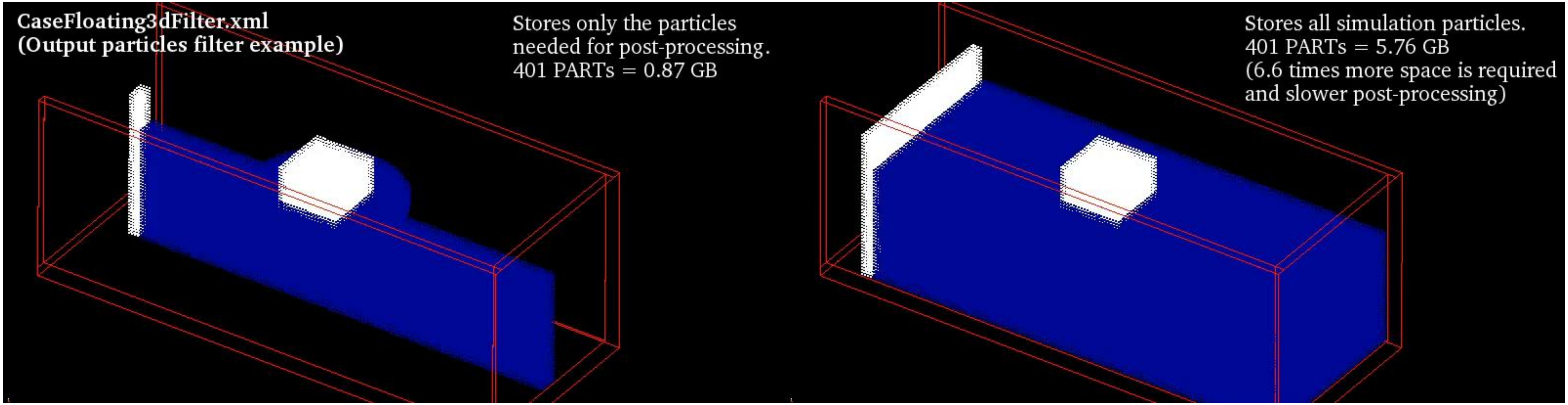


Main novelties in v5.4

Filter options for output particle data (space saving)

- Long and/or high resolution simulations generate an huge volume of output data (890 MB per 10M particles).
- It creates memory space problems and high post-processing run times.
- New option allows you to select the particles you need for post-processing and ignore the rest.
  - Several options for particle filtering (position, type, mk...)
  - Several options to combine different filters.
  - Filtering particles around moving or floating objects is also possible.

Full output data requires 5.8 GB and filtered output data 0.9 GB (**6.6 times less required space**)



Main novelties in v5.4

Other SPH solver improvements:

- Laminar viscosity option without SPS turbulence model is enabled.
    - Laminar + SPS is too dissipative, so it is recommended only for high resolution and turbulent flows (as shown in Tagliafierro's talk).
    - Laminar without SPH is faster (~30%).

- Significant improvements in the code (only for developers).
    - New implementation for particle arrays on CPU and GPU. Some tasks are now automatic and it is easier to add new properties.
    - New simpler and faster implementation for floating bodies among CPU and GPU.
    - Significant simplification in different parts (gauges, inlet/outlet, some formulations…).

Main novelties in v5.4 for pre-processing

New formats are supported

- Supports new XML-VTP format (PolyData datasheet, ASCII/binary/appended, base64/raw).
- Supports new XML-VTU format (UnstructuredGrid datasheet, ASCII/binary/appended, base64/raw) generated by clipping operations in ParaView.
- Different variants of VTK, VTU and VTP files are supported by the <DRAWFILEVTK> operation transparently to the user.

```
<drawfilevtk file="File.vtk"/>
<drawfilevtk file="File.vtp"/>
<drawfilevtk file="File.vtu"/>
```

- Supports new VTM format that groups multiple named surfaces into VTP and VTU files.
- VTM files are supported by the new XML operation <DRAWFILEVTM> for regular and advanced drawing mode, and includes multiple filtering options (by surface name or id) and assignment of different MK values.

```
<drawfilevtm file="Surfaces.vtm">
  <onlysurfaces index="0-3"/>
  <onlysurfaces index="4,8,6,9" mkbound="40-42,49"/>
  <onlysurfaces name="Solid1,Solid2"/>
  <onlysurfaces name="Solid3" mkbound="3"/>
  <onlysurfaces name="Solid44" mkbound="44"/>
</drawfilevtm>
```

Main novelties in v5.4 for pre-processing

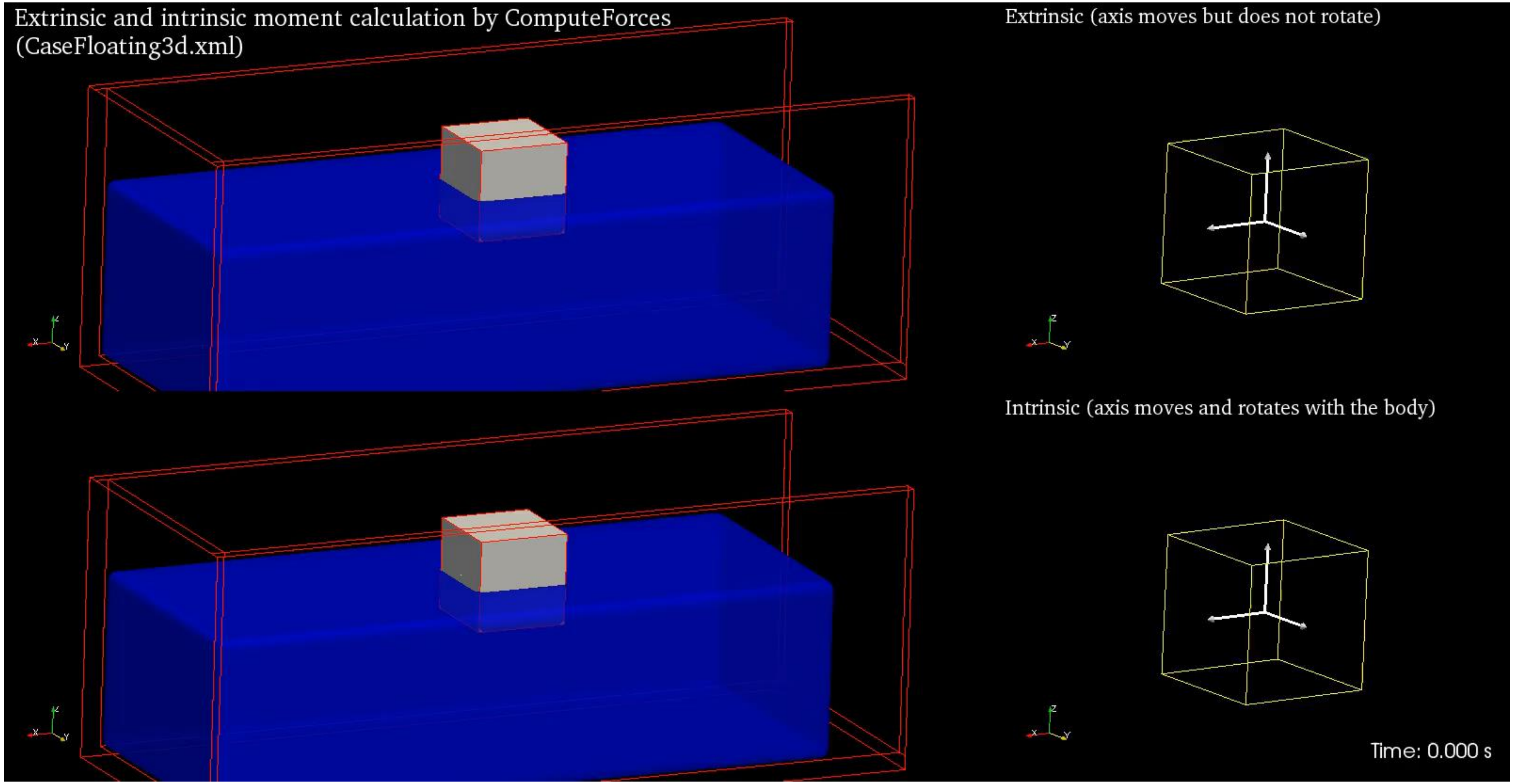
GenCase improvements for huge complex geometries and large testcases

- New algorithms to support external geometries with a **huge number of triangles (+30M)**.
- Split processing of large files to **avoid out-of-memory problems**.
- Major **performance improvements on advanced drawing mode** for large geometries (**speedup 100x**).
- Significant **performance improvements on automatic normal calculation** for large geometries (**speedup 5x**).
- Supports **output files larger than 4 GB** to generate hundreds of millions of particles.

Main novelties in v5.4 for post-processing

Performance improvements and new options

- **Extrinsic and intrinsic moment** force calculation (*ComputeForces*).



Main novelties in v5.4 for post-processing

Performance improvements and new options

- **Extrinsic and intrinsic moment** force calculation (*ComputeForces*).
- **Improved area selection** for flow calculation by XML file with **many options** (*FlowTool*).

```xml
<flowtool_boxes>
    <box8pt name="Domain1">
        <point x="0.0" y="0.2" z="0.0" />
        <point x="0.0" y="0.0" z="0.0" />
        <point x="0.1" y="0.0" z="0.0" />
        <point x="0.1" y="0.2" z="0.0" />
        <point x="0.0" y="0.2" z="0.4" />
        <point x="0.0" y="0.0" z="0.4" />
        <point x="0.1" y="0.0" z="0.4" />
        <point x="0.1" y="0.2" z="0.4" />
    </box8pt>
    <box2pt name="Domain2">
        <point x="1.2783" y="-0.7755" z="1.1631" />
        <point x="1.3" y="-0.7" z="1.2" />
    </box2pt>
    <boxsize name="Domain3">
        <point x="1.2783" y="-0.7755" z="1.1631" />
        <size x="0.0792" y="0.1135" z="0.0621" />
    </boxsize>
    <boxangle name="Domain4">
        <point x="1.2783" y="-0.7755" z="1.1631" />
        <angle x="3" y="-4" z="5" />
        <size x="0.0792" y="0.1135" z="0.0621" />
    </boxangle>
    <boxanglediv name1="Domain5a" name2="Domain5b">
        <divide axis="y" />
        <point x="1.2783" y="-0.7755" z="1.1631" />
        <angle x="3" y="-4" z="5" />
        <size x="0.0792" y="0.1135" z="0.0621" />
    </boxanglediv>
</flowtool_boxes>
```

Main novelties in v5.4 for post-processing

Performance improvements and new options

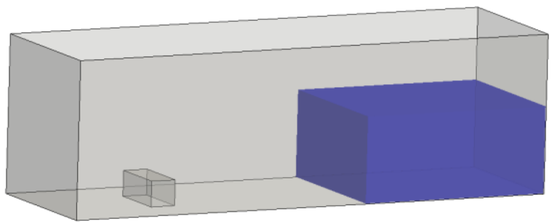
- Moment force calculation on **intrinsic and extrinsic** axis (*ComputeForces*)
- **Improved area selection** for flow calculation by XML file with **many options** (*FlowTool*).
- **Much faster motion processing** since it is computed during the simulation and the reading of output data is avoided. (*FloatingInfo* & *BoundaryVtk*). Almost instantaneous execution!
- **More floating body information** on fluid forces and external forces (*FloatingInfo*).
    - External linear and angular forces (moorings and imposed forces).
    - Linear and angular forces from fluid.
    - Linear and angular acceleration before constrints.
- **Higher frequency output** on moving and floating bodies (*BoundaryVtk* & *FloatingInfo*).

```
<parameter key="TimeOutExtra" value="0.001" comment="Time out for extra output data on motion and floating information"
```

DualSPHysics performance

Testcase for performance test

Dam break flow impacting an obstacle (experiment by kleefsman et al., 2005). 2 physical seconds of simulation.



Speed-up: 533x on **RTX 4090**
511x on **L40S**
over
CPU Intel Core i7-8700K
(12 threads)

SPH HIGHLY PARALLELISED