Updates on modified Dynamic Boundary Conditions (mDBC)

AARON ENGLISH

Outline

Motivation

Boundary Conditions in DualSPHysics

- Dynamic Boundary Condition (DBC)
- modified Dynamic Boundary Conditions (mDBC)

Updates to mDBC

- Numerical checks
- Pressure cloning
- No-slip

Example Test Cases

Conclusions and future work

Motivation

Journal papers

Reference paper (link)

Domínguez JM, Fourtakas G, Altomare C, Car
Estévez I, Mokos A, Vacondio R, Crespo AJC,
2022. DualSPHysics: from fluid dynamics to r
Particle Mechanics, 9(5), 867-895. doi:10.10(

Zotero database of journal papers using DualSPHysics:
https://www.zotero.org/groups/2862487/dualsphysics/library

2024

Feng R, Fourtakas G, Rogers BD, Lombardi D. 2024. Computer Methods in Applied Mechanics and Engineering, 419, 116581. doi:10.1016/j.cma.2023.116581.

Yang Y, English A, Rogers BD, Stansby PK, Stagonas D, Buldakov E, Draycott S. 2024. Numerical modelling of a vertical cylinder with dynamic response in steep and breaking waves using smoothed particle hydrodynamics. Journal of Fluids and Structures, 125, 104049. doi:10.1016/j.jfluidstructs.2023.104049.

Cen C, Fourtakas G, Lind S, Rogers BD. 2024. A single-phase GPU-accelerated surface tension model using SPH. Computer Physics Communications, 295, 109012. doi:10.1016/j.cpc.2023.109012.

Ricci F, Vacondio R, Tafuni A. 2024. Multiscale Smoothed Particle Hydrodynamics based on a domain-decomposition strategy. Computer Methods in Applied Mechanics and Engineering, 418(A), 116500. doi:10.1016/j.cma.2023.116500.

2023

Laha S, Fourtakas G, Kuamr Das P, Keshmiri A. 2023. Fluid–structure interaction modeling of bi-leaflet mechanical heart valves using smoothed particle hydrodynamics. Physics of Fluids, 35, 121902. doi:10.1063/5.0172043.

Yang Y, Stansby PK, Rogers BD, Buldakov E, Stagonas D, Draycott S. 2023. The loading on a vertical cylinder in steep and breaking waves on sheared currents using smoothed particle hydrodynamics, Physics of Fluids, 35(9), 097132. doi.10.1063/5.0160021.

Antona R, Vacondio R, Avesani D, Righetti M, Renzi M. 2023. A WENO SPH scheme with improved transport velocity and consistent divergence operator. Computational Particle Mechanics. doi.10.1007/s40571-023-00681-z.

Tagliafierro B, Karimirad M, Altomare C, Göteman M, Martínez-Estévez I, Capasso S, Domínguez JM, Viccione G, Gómez-Gesteira M, Crespo AJC. 2023. Numerical validations and investigation of a semi-submersible floating offshore wind turbine platform interacting with ocean waves using an SPH framework. Applied Ocean Research, 141, 103757. doi:10.1016/j.apor.2023.103757.

King J, Lind S, Rogers BD, Stansby PK, Vacondio R. (2023). Large eddy simulations of bubbly flows and breaking waves with smoothed particle hydrodynamics. Journal of Fluid Mechanics, 972, A24. doi:10.1017/jfm.2023.649.

Sun JZ, Zou L, Govender N, Martínez-Estévez I, Crespo AJC, Sun Z, Domínguez JM. 2023. A resolved SPH-DEM coupling method for analysing the interaction of polyhedral granular materials with fluid. Ocean Engineering, 287 (2), 115938. doi:10.1016/j.oceaneng.2023.115938.

Altomare C, Scandura P, Cáceres I, van der A D, Viccione G. 2023. Large-scale wave breaking over a barred beach: SPH numerical simulation and comparison with experiments. Coastal Engineering, 185, 104362. doi:10.1016/j.coastaleng.2023.104362.

El Rahi J, Martínez-Estévez I, Tagliafierro B, Domínguez JM, Crespo AJC, Stratigaki V, Suzuki T, Troch P. 2023. Numerical investigation of wave-induced flexible vegetation dynamics in 3D using a coupling between DualSPHysics and the FEA module of Project Chrono. Ocean Engineering, 285, 115227. doi:10.1016/j.oceaneng.2023.115227.

Ricci F, Vacondio R, Tafuni, A. 2023. Direct numerical simulation of three-dimensional isotropic turbulence with smoothed particle hydrodynamics. Physics of Fluids, 35, 065148. doi:10.1063/5.0152154.

Yang Y, Draycott S, Stansby PK, Rogers BD. 2023. A numerical flume for waves on variable sheared currents using smoothed particle hydrodynamics (SPH) with open boundaries. Applied Ocean Research, 135, 103527. doi:10.1016/j.apor.2023.103527.

Mitsui J, Altomare C, Crespo AJC, Dominguez JM, Martínez-Estévez I, Suzuki T, Kubota S, Gómez-Gesteira M. 2023. DualSPHysics modelling to analyse the response of Tetrapods against solitary wave. Coastal Engineering, 183, 104315. doi:10.1016/j.coastaleng.2023.104315.

Capasso S, Tagliafierro B, Mancini S, Martínez-Estévez I, Altomare C, Dominguez JM, Viccione G. 2023. Regular Wave Seakeeping Analysis of a Planing Hull by Smoothed Particle Hydrodynamics: A Comprehensive Validation. Journal of Marine Science and Engineering, 11(4), 700. doi:10.3390/jmse11040700.

Martínez-Estévez I, Tagliafierro B, El Rahi J, Dominguez JM, Crespo AJC, Troch P, Gómez-Gesteira M. 2023. Coupling an SPH-based solver with an FEA structural solver to simulate free surface flows interacting with flexible structures. Computer Methods in Applied Mechanics and Engineering, 410, 115989. doi:10.1016/j.cma.2023.115989.

Pringgana G, Rogers BD, Cunningham L. 2023. Mitigating tsunami effects on buildings via novel use of discrete onshore protection systems. Coastal Engineering Journal, 65:1. doi:10.1080/21664250.2023.2170690.

Kotsarinis K, Green MD, Simonini A, Debarre O, Magin T, Tafuni A. 2023. Modeling sloshing damping for spacecraft: A smoothed particle hydrodynamics application. Aerospace Science and Technology, 108090. doi: 10.1016/j.ast.2022.108090.

Majtan E, Cunningham LS, Rogers BD. 2023. Numerical study on the structural response of a masonry arch bridge subject to flood flow and debris impact. Structures, 48, 782-797. doi:10.1016/j.istruc.2022.12.100.

Aslami MH, Rogers BD, Stansby PK, Bottacin-Busolin A. 2023. Simulation of floating debris in SPH shallow water flow model with tsunami application. Advances in Water Resources, 171, 104363. doi:10.1016/j.advwatres.2022.104363.

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

Martínez-Estévez I, Domínguez JM, Tagliafierro B, Canelas RB, García-Feal O, Crespo AJC, Gómez-Gesteira M. 2023. Coupling of an SPH-based solver with a multiphysics library. Computer Physics Communications, 283, 108581. doi:10.1016/j.cpc.2022.108581.

Novak G, Pengal P, Silva AT, Domínguez JM, Tafuni A, Cetina M, Zagar D. 2023. Ecological Modelling. Interdisciplinary design of a fish ramp using migration routes analysis, 475, 110189. doi:10.1016/j.ecolmodel.2022.110189.

Motivation

	Delegates Session 1	**Chair: Alejandro Crespo**
10:35–10:50	Latest outcomes of DualSPHysics' use in renewable energy simulations: good and bad news	Bonaventura Tagliafierro
10:50–11:05	Sensitivity analysis of snap loads on floating bodies using DualSPHysics with MoodyCore and MoorDyn+	Gael Verao Fernández
11:05–11:20	Numerical modelling of novel offshore and nearshore structures using DualSPHysics	Annelie Baines
11:20–11:35	Capturing WECs' extreme response: experimental validation of DualSPHysics and STAR-CCM+	Oronzo Dell'Edera
	Delegates Session 2	**Chair: Corrado Altomare**
11:40–11:55	3-D modelling of riverine flood and floating debris at arch bridges	Benedict Rogers
11:55–12:10	3-D simulations of turbulent flow past a cylindrical bridge pier	Gorazd Novak
12:10–12:25	Fluid-debris-structure interaction in extreme hydrodynamic events using DualSPHysics and CHRONO	Gioele Ruffini
12:25–12:40	Simulation of tsunami-borne debris interaction with structures in DualSPHysics coupled with CHRONO	Kenshiro Ishiki
12:40–14:40	**Lunch** (location here)	
	Developers Talks	
14:45–15:10	*Novelties and new appl*	
15:10–15:35	*Updates on multi-phase*	
15:35–16:00	*Updates on modified D*	
16:00–16:40	**Coffee break**	
	Delegates Session 3	
16:40–16:55	Preliminary analysis of	
16:55–17:10	Hydrodynamics of oscil	
17:10–17:25	SPH-FSI model of the r	
17:25–17:40	Simulating flexible vege	
17:40–17:55	Modeling flexible veget	

GC#1: Convergence, consistency and stability
Leaders: J.J. Monaghan, D. Violeau and R. Vignjevic

GC#2: Boundary conditions
Leaders: A. Souto-Iglesias

GC#3: Adaptivity
Leaders: B.D. Rogers and R. Vacondio

GC#4: Coupling to other models
Leaders: S. Marrone, C. Altomare and D. Le Touzé

GC#5: Applicability to industry
Leaders: J-C. Marongiu and M. De Leffe

09:35–10:15	**KEYNOTE: Industrial use of DualSPHysics: past, present and future**	**Georgios Fourtakas**
10:15–10:55	**Coffee break**	
	Delegates Session 4	**Chair: Annelie Baines**
10:55–11:10	Measuring flap gates wastewater flow	Tanguy Pouzol
11:10–11:25	Implementation of Mohr-Coulomb criterion in the non-newtonian fluid model	Chiaki Tsurudome
11:25–11:40	KRÆKEN: easy DualSPHysics GUI to study urban hydrology	Priscille Béguin
11:40–11:55	A numerical method for studying interaction problems between soil, rocks and structures with SPH and DDA	Guangqi Chen
	Delegates Session 5	**Chair: Bonaventura Tagliafierro**
12:00–12:15	Modelling of arbitrary sheared currents and focused waves using DualSPHysics for offshore applications	Aaron English
12:15–12:30	SPH numerical modelling of a U-OWC wave energy converter	Beatrice Mina
12:30–12:45	SPH numerical investigation of attenuating waves by an array of submerged resonators	Lucas Calvo
12:45–13:00	Performance assessment of a point-absorber WEC in waves & currents environments using DualSPHysics	Giacomo Viccione

DualSPHysics users (and SPH users in general) are wanting to simulate more and more complex problems with either difficult fluid structure interactions or complex shaped geometries

Aim: to deliver a free engineering tool with boundary conditions that are as good and easy to use as possible

Boundary Conditions in DualSPHysics

DualSPHysics currently contains two options for the boundary condition:

Option #1: Dynamic Boundary Condition (DBC)

```
<parameter key="Boundary" value="1" comment="Boundary method 1:DBC, 2:mDBC (default=1)" />
```

- Quick and easy to set up
- Has some drawbacks including the “gap”

Option #2 : modified Dynamic Boundary Condition (mDBC)

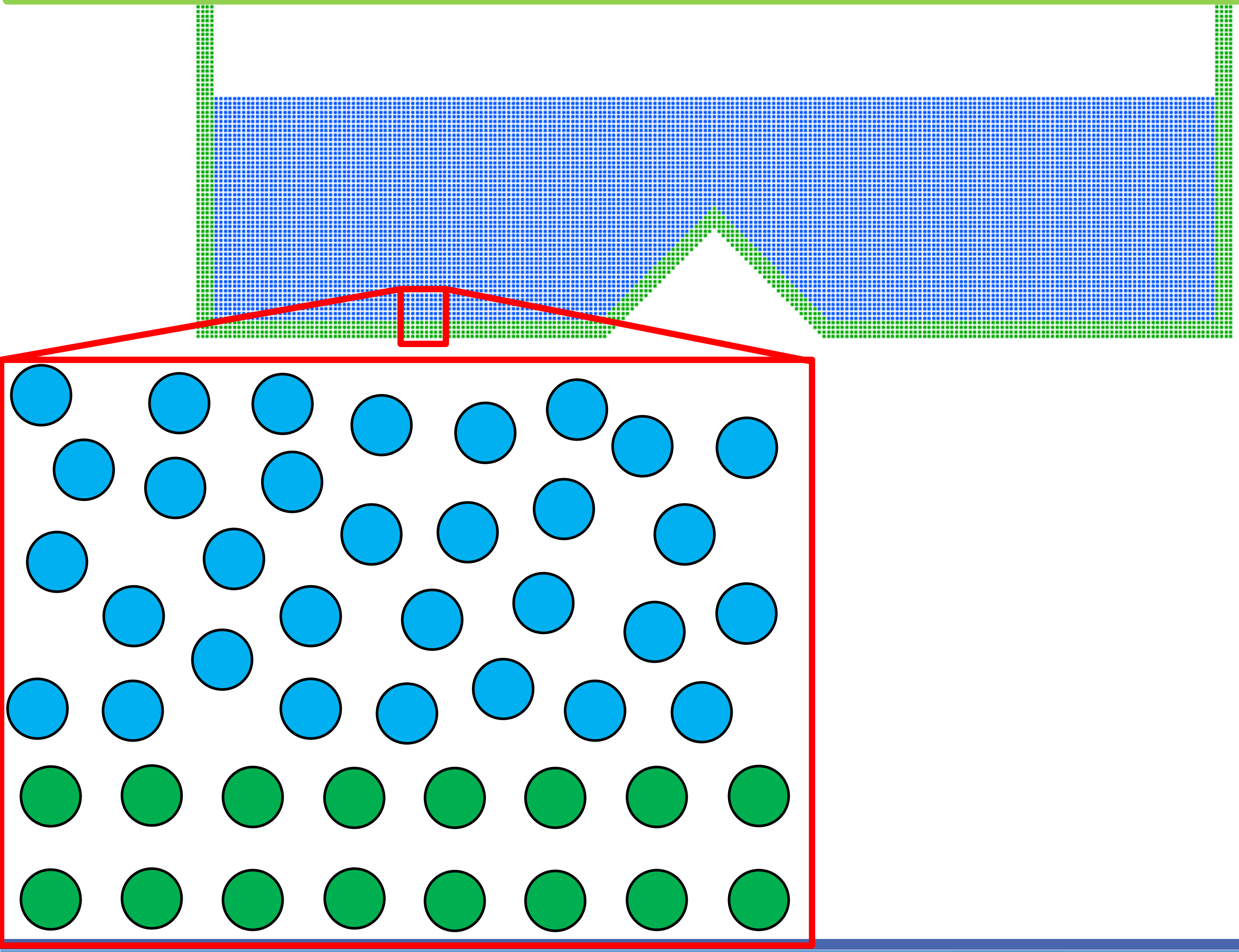
```
<parameter key="Boundary" value="2" comment="Boundary method 1:DBC, 2:mDBC (default=1)" />
```

- More difficult to set up
- Removes the gap but still has some issues

Option #3 : Improved mDBC formulation

DBC

```xml
<parameter key="Boundary" value="1" comment="Boundary method 1:DBC, 2:mDBC (default=1)" />
```



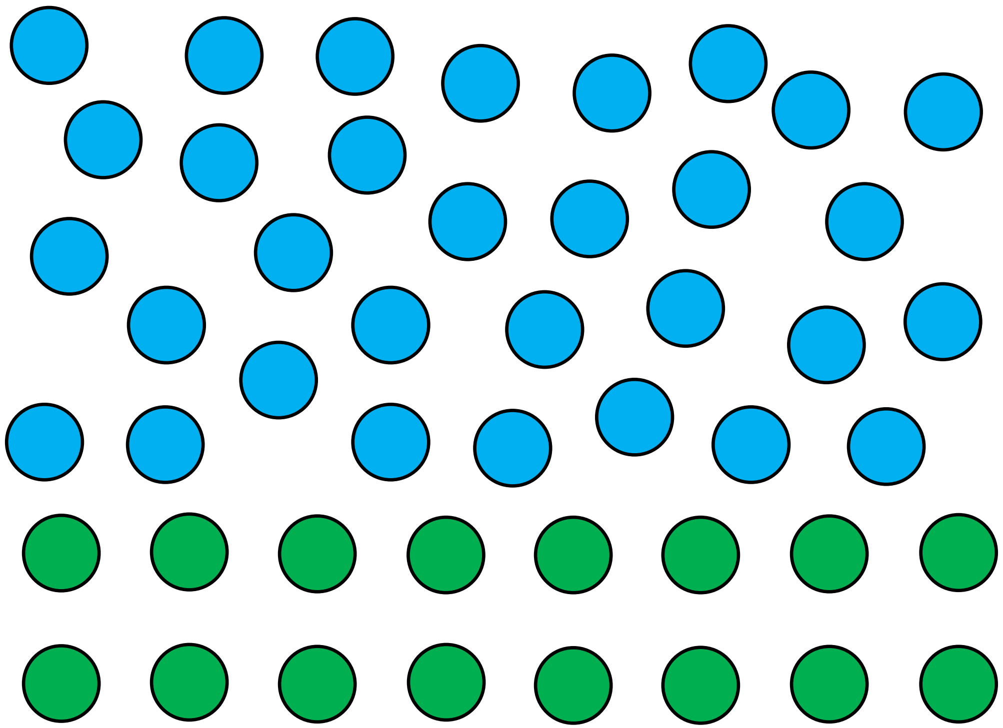
```xml
<?xml version="1.0" encoding="UTF-8" ?>
<casedef>
    <constantsdef>
        <gravity x="0" y="0" z="-9.81" comment="Gravitational acceleration" units_comment="m/s^2" />
        <rhop0 value="1000" comment="Reference density of the fluid" units_comment="kg/m^3" />
        <rhopgradient value="2" comment="Initial density gradient 1:Rhop0, 2:Water column, 3:Max. water height (default=2)" />
        <hswl value="0" auto="true" comment="Maximum still water level to calculate speedofsound using coefsound" units_comment="metres (m)" />
        <gamma value="7" comment="Politropic constant for water used in the state equation" />
        <speedsystem value="0" auto="true" comment="Maximum system speed (by default the dam-break propagation is used)" />
        <coefsound value="20" comment="Coefficient to multiply speedsystem" />
        <speedsound value="0" auto="true" comment="Speed of sound to use in the simulation (by default speedofsound=coefsound*speedsystem)" />
        <hdp value="1.3" comment="Alternative option to calculate the smoothing length (h=hdp*dp)" />
        <cflnumber value="0.2" comment="Coefficient to multiply dt" />
    </constantsdef>
    <mkconfig boundcount="240" fluidcount="9" />
    <geometry>
        <definition dp="0.01" units_comment="metres (m)">
            <pointmin x="-1" y="0" z="-1" />
            <pointmax x="2.5" y="0" z="2" />
        </definition>
        <commands>
            <list name="GeometryForNormals">
                <setactive drawpoints="0" drawshapes="1"/>
                <setshapemode>actual | bound</setshapemode>
                <setmkbound mk="0" />
                <drawextrude closed="false">
                    <point x="0.005" y="-0.1" z="0.7" />
                    <point x="0.005" y="-0.1" z="0.005" />
                    <point x="0.85" y="-0.1" z="0.005" />
                    <point x="1.1" y="-0.1" z="0.255" />
                    <point x="1.35" y="-0.1" z="0.005" />
                    <point x="2.195" y="-0.1" z="0.005" />
                    <point x="2.195" y="-0.1" z="0.7" />
                    <extrude x="0" y="0.2" z="0" />
                </drawextrude>
                <shapeout file="hdp" />
                <resetdraw />
            </list>
            <mainlist>
                <!-- Actual geometry at dp/2 -->
                <runlist name="GeometryForNormals" />
                <!-- Boundary particles -->
                <setdrawmode mode="full" />
                <setmkbound mk="0" />
                <drawextrude closed="false">
                    <point x="0" y="0" z="0.7" />
                    <point x="0" y="0" z="0" />
                    <point x="0.85" y="0" z="0" />
                    <point x="1.1" y="0" z="0.25" />
                    <point x="1.35" y="0" z="0" />
                    <point x="2.2" y="0" z="0" />
                    <point x="2.2" y="0" z="0.7" />
                    <extrude x="0" y="0.1" z="0" />
                    <layers vdp="0,-0.5,-1,-1.5,-2,-3" />
                </drawextrude>
                <!-- Fluid particles -->
                <setmkfluid mk="0" />
                <fillbox x="0.1" y="0" z="0.1">
                    <modefill>void</modefill>
                    <point x="-0.1" y="-1" z="-0.1" />
                    <size x="2.3" y="2" z="0.6" />
                </fillbox>
            </mainlist>
        </commands>
    </geometry>
    <normals>
        <distanceh value="3.0" comment="Maximum distance (H*distanceh) to compute normals data (default=2)" />
        <geometryfile file="[CaseName]_hdp_Actual.vtk" comment="File with boundary geometry (VTK format)" />
    </normals>
</casedef>
<execution>
    <parameters>
        <parameter key="SavePosDouble" value="0" comment="Saves particle position using double precision (default=0)" />
        <parameter key="Boundary" value="2" comment="Boundary method 1:DBC, 2:mDBC (default=1)" />
        <parameter key="SlipMode" value="2" comment="Boundary slip mode 1:Vel=0, 2:No-Slip (default=1)"/>
        <parameter key="StepAlgorithm" value="2" comment="Step Algorithm 1:Verlet, 2:Symplectic (default=1)" />
        <parameter key="VerletSteps" value="40" comment="Verlet only: Number of steps to apply Euler timestepping (default=40)" />
        <parameter key="Kernel" value="2" comment="Interaction Kernel 1:Cubic Spline, 2:Wendland (default=2)" />
        <parameter key="ViscoTreatment" value="1" comment="Viscosity formulation 1:Artificial, 2:Laminar+SPS, 3:Laminar (default=1)" />
        <parameter key="Visco" value="0.01" comment="Viscosity value. Typically 0.01 for Artificial and 1e-6 m^2/s (water kinematic viscosity) for Laminar" />
        <parameter key="ViscoBoundFactor" value="1" comment="Multiply viscosity value with boundary (default=1)" />
        <parameter key="DensityDT" value="3" comment="Density Diffusion Term 0:None, 1:Molteni, 2:Fourtakas, 3:Fourtakas(full) (default=0)" />
        <parameter key="DensityDTvalue" value="0.1" comment="DDT value (default=0.1)" />
        <parameter key="Shifting" value="0" comment="Shifting mode 0:None, 1:Ignore bound, 2:Ignore fixed, 3:Full (default=0)" />
        <parameter key="ShiftCoef" value="-2" comment="Coefficient for shifting computation (default=-2)" />
        <parameter key="ShiftTFS" value="1.5" comment="Threshold to detect free surface. Typically 1.5 for 2D and 2.75 for 3D (default=0)" />
        <parameter key="RigidAlgorithm" value="1" comment="Rigid Algorithm 0:collision-free, 1:SPH, 2:DEM, 3:Chrono (default=1)" />
        <parameter key="FtPause" value="0.0" comment="Time to freeze the floatings at simulation start (warmup) (default=0)" units_comment="seconds" />
        <parameter key="CoefDtMin" value="0.05" comment="Coefficient to calculate minimum time step dtmin=coefdtmin*h/speedsound (default=0.05)" />
        <parameter key="DtIni" value="0" comment="Initial time step. Use 0 to default use (default=h/speedsound)" units_comment="seconds" />
        <parameter key="DtMin" value="0" comment="Minimum time step. Use 0 to default use (default=coefdtmin*h/speedsound)" units_comment="seconds" />
        <parameter key="DtFixed" value="0" comment="Fixed Dt value. Use 0 to disable (default=disabled)" units_comment="seconds" />
        <parameter key="DtFixedFile" value="NONE" comment="Dt values are loaded from file. Use NONE to disable (default=disabled)" units_comment="milliseconds (ms)" />
        <parameter key="DtAllParticles" value="0" comment="Velocity of particles used to calculate DT. 1:All, 0:Only fluid/floating (default=0)" />
        <parameter key="TimeMax" value="15" comment="Time of simulation" units_comment="seconds" />
        <parameter key="TimeOut" value="0.01" comment="Time out data" units_comment="seconds" />
        <parameter key="PartsOutMax" value="1" comment="Allowed %/100 of fluid particles out the domain (default=1)" units_comment="decimal" />
        <parameter key="RhopOutMin" value="700" comment="Minimum rhop valid (default=700)" units_comment="kg/m^3" />
        <parameter key="RhopOutMax" value="1300" comment="Maximum rhop valid (default=1300)" units_comment="kg/m^3" />
        <simulationdomain comment="Defines domain of simulation (default=Uses minimum and maximum position of the generated particles)">
            <posmin x="default" y="default" z="default" comment="e.g.: x=0.5, y=default-1, z=default-10%" />
            <posmax x="default" y="default" z="default" />
        </simulationdomain>
    </parameters>
</execution>
</case>
```

DBC

```
<parameter key="Boundary" value="1" comment="Boundary method 1:DBC, 2:mDBC (default=1)" />
```

$$\frac{d\rho_i}{dt} = \sum_j m_j \mathbf{u}_{ij} \cdot \nabla W_{ij}$$

$$\mathbf{u}_i = \mathbf{0}$$

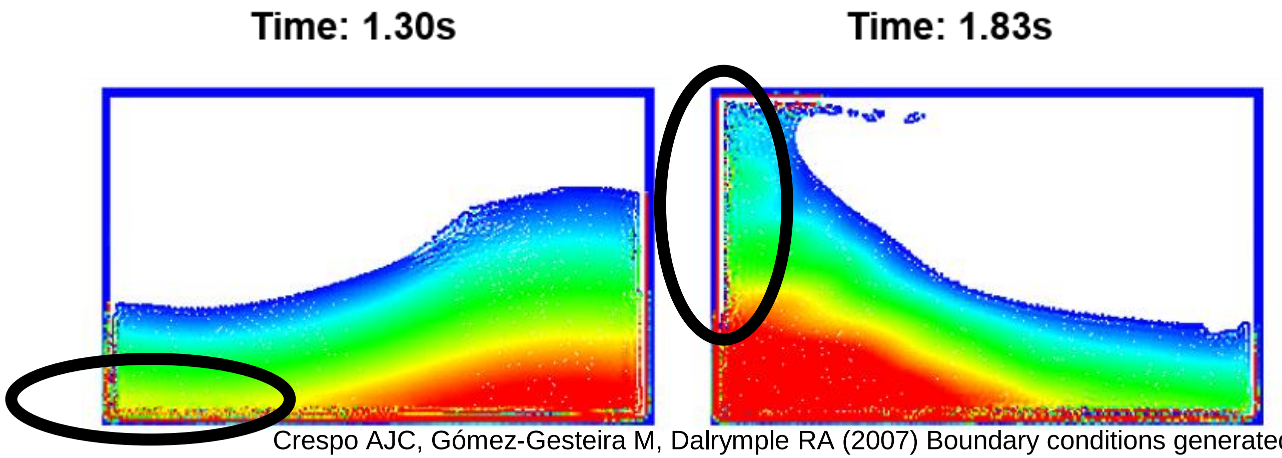


ADVANTAGES

- high computational efficiency
- ability to deal with very complex geometries
- can be used for real engineering problems

DISADVANTAGES

- unphysical density/pressure values of the boundary particles
- high repulsive force resulting in a separation distance (GAP)

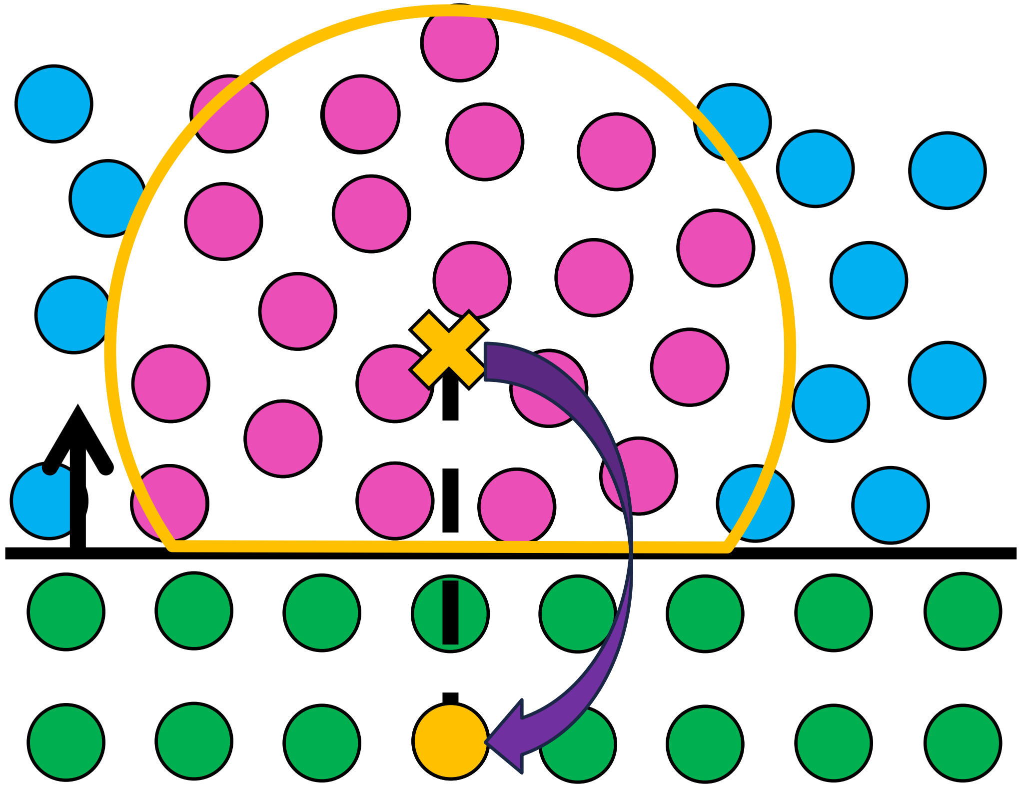


Crespo AJC, Gómez-Gesteira M, Dalrymple RA (2007) Boundary conditions generated by dynamic particles in SPH methods. Comput Mater Contin 5:173–184. https://doi.org/10.3970/cmc.2007.005.17

mDBC

```
<parameter key="Boundary" value="2" comment="Boundary method 1:DBC, 2:mDBC (default=1)" />
```

First create a ghost node in the fluid domain by mirroring our boundary particle position across the boundary interface (hdp) into the fluid using the boundary normal



To correct the truncated kernel we use the multi-dimensional first-order Taylor series approximation of Liu and Liu (2006) to retrieve first order kernel and particle consistency

$$\boldsymbol{A}_g \cdot \left[\rho_g \;\; \partial_x\rho_g \;\; \partial_y\rho_g \;\; \partial_z\rho_g\right]^{\mathrm{T}} = \mathbf{b}_{\mathrm{g}}$$

$$\mathbf{b}_{\mathrm{g}} = \left[\sum_j \rho_j V_j W_{gj} \;\; \sum_j \rho_j V_j \partial_x W_{gj} \;\; \sum_j \rho_j V_j \partial_y W_{gj} \;\; \sum_j \rho_j V_j \partial_z W_{gj}\right]^{\mathrm{T}}$$

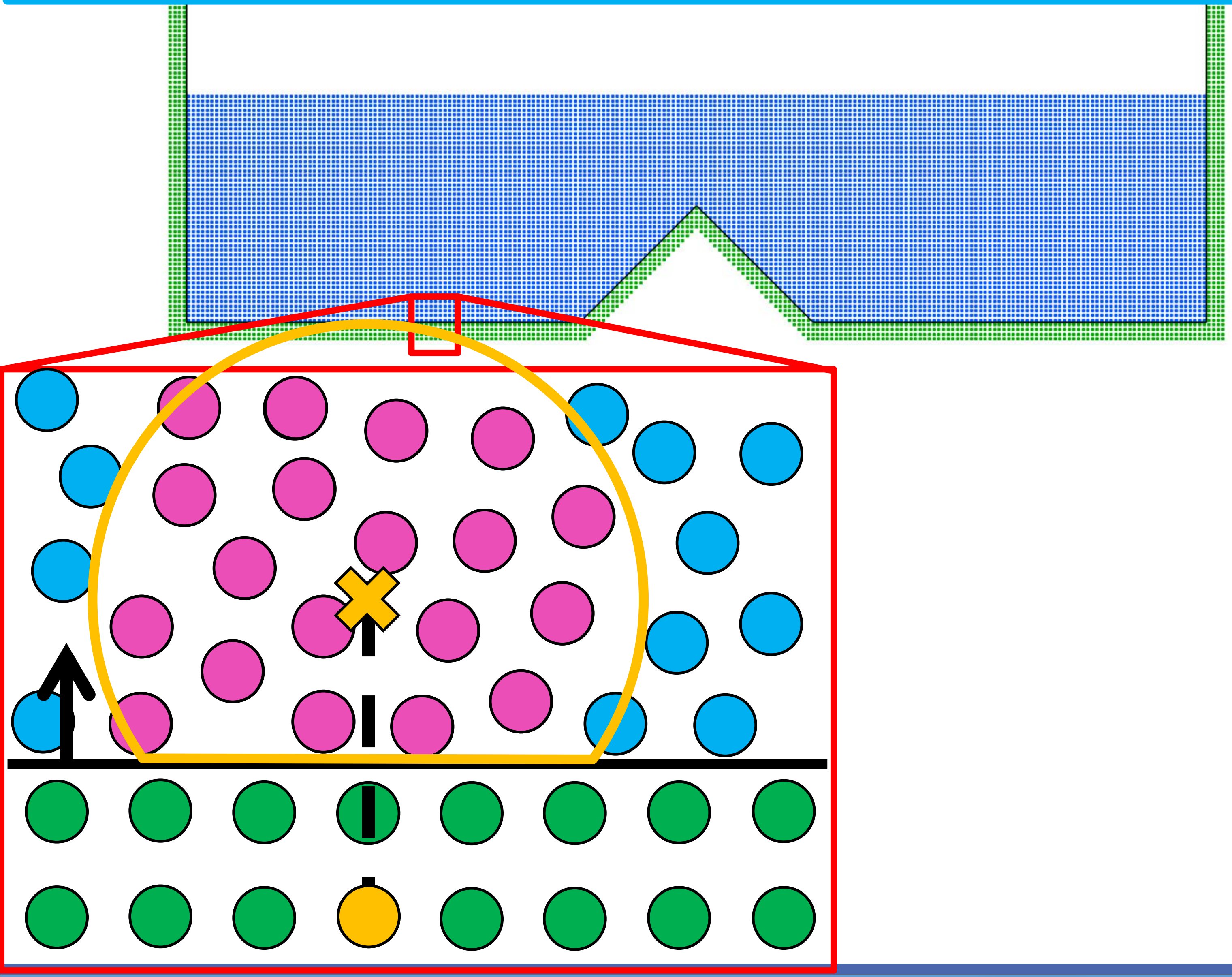
$$\boldsymbol{A}_g = \begin{bmatrix} \sum_j V_j W_{gj} & \sum_j (x_j - x_g) V_j W_{gj} & \sum_j (y_j - y_g) V_j W_{gj} & \sum_j (z_j - z_g) V_j W_{gj} \\ \sum_j V_j \partial_x W_{gj} & \sum_j (x_j - x_g) V_j \partial_x W_{gj} & \sum_j (y_j - y_g) V_j \partial_x W_{gj} & \sum_j (z_j - z_g) V_j \partial_x W_{gj} \\ \sum_j V_j \partial_y W_{gj} & \sum_j (x_j - x_g) V_j \partial_y W_{gj} & \sum_j (y_j - y_g) V_j \partial_y W_{gj} & \sum_j (z_j - z_g) V_j \partial_y W_{gj} \\ \sum_j V_j \partial_z W_{gj} & \sum_j (x_j - x_g) V_j \partial_z W_{gj} & \sum_j (y_j - y_g) V_j \partial_z W_{gj} & \sum_j (z_j - z_g) V_j \partial_z W_{gj} \end{bmatrix}$$

$$\rho_b = \rho_g + (\mathbf{r}_{\mathrm{b}} - \mathbf{r}_{\mathrm{g}}) \cdot \left[\partial_x\rho_g \;\; \partial_y\rho_g \;\; \partial_z\rho_g\right]^{\mathrm{T}}$$

English, A., Domínguez, J.M., Vacondio, R. *et al.* Modified dynamic boundary conditions (mDBC) for general-purpose smoothed particle hydrodynamics (SPH): application to tank sloshing, dam break and fish pass problems. *Comp. Part. Mech.* **9**, 1–15 (2022). https://doi.org/10.1007/s40571-021-00403-3

mDBC

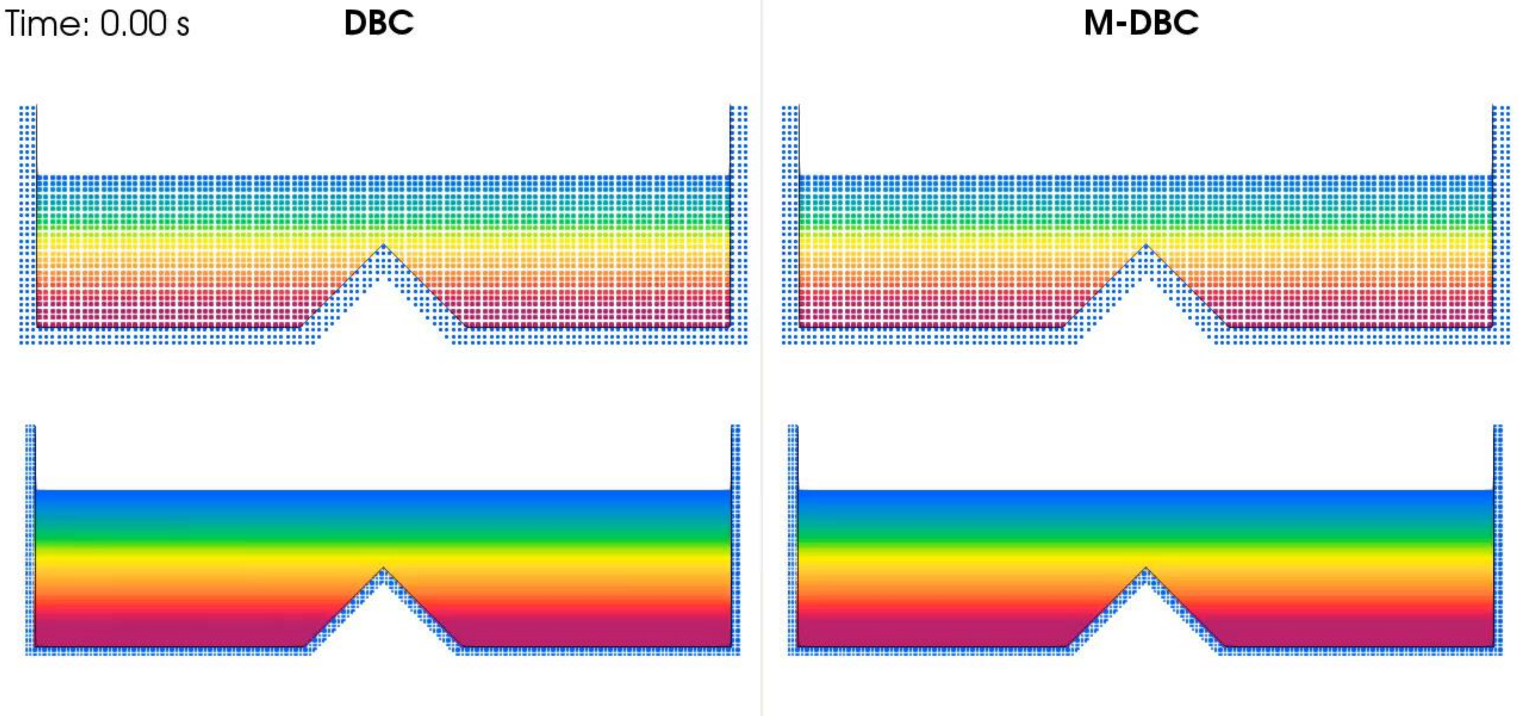
`<parameter key="Boundary" value="2" comment="Boundary method 1:DBC, 2:mDBC (default=1)" />`

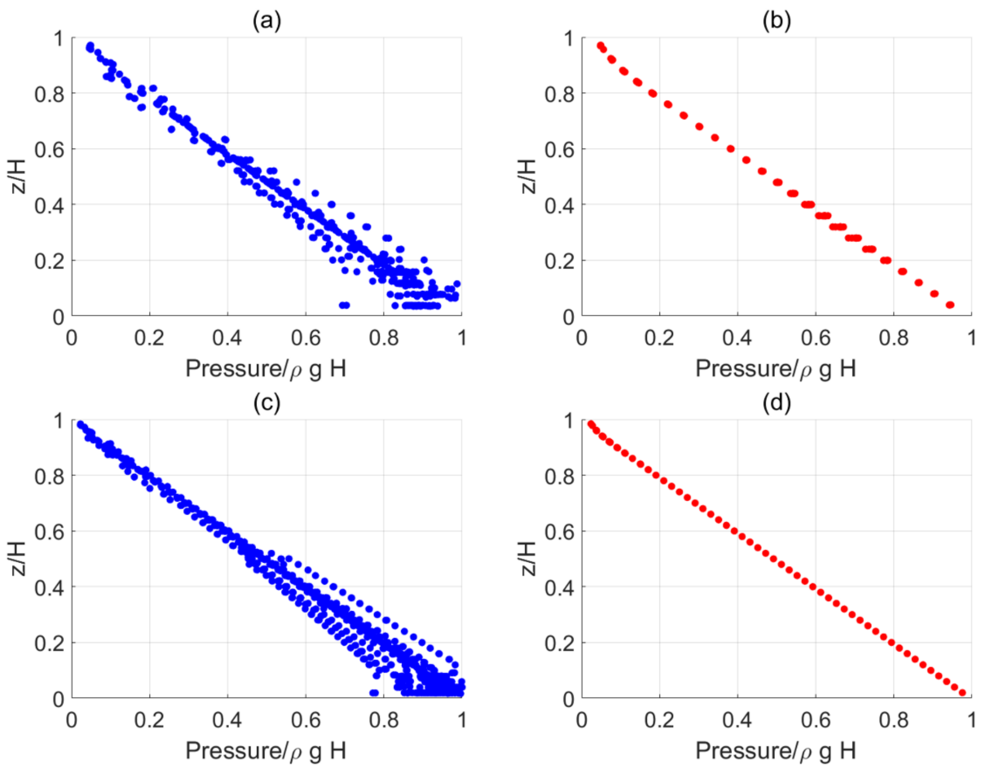


```xml
<?xml version="1.0" encoding="UTF-8" ?>
<casedef>
    <constantsdef>
        <gravity x="0" y="0" z="-9.81" comment="Gravitational acceleration" units_comment="m/s^2" />
        <rhop0 value="1000" comment="Reference density of the fluid" units_comment="kg/m^3" />
        <rhopgradient value="2" comment="Initial density gradient 1:Rhop0, 2:Water column, 3:Max. water height (default=2)" />
        <hswl value="0" auto="true" comment="Maximum still water level to calculate speedofsound using coefsound" units_comment="metres (m)" />
        <gamma value="7" comment="Politropic constant for water used in the state equation" />
        <speedsystem value="0" auto="true" comment="Maximum system speed (by default the dam-break propagation is used)" />
        <coefsound value="20" comment="Coefficient to multiply speedsystem" />
        <speedsound value="0" auto="true" comment="Speed of sound to use in the simulation (by default speedofsound=coefsound*speedsystem)" />
        <hdp value="1.3" comment="Alternative option to calculate the smoothing length (h=hdp*dp)" />
        <cflnumber value="0.2" comment="Coefficient to multiply dt" />
    </constantsdef>
    <mkconfig boundcount="240" fluidcount="9" />
    <geometry>
        <definition dp="0.01" units_comment="metres (m)">
            <pointmin x="-1" y="0" z="-1" />
            <pointmax x="2.5" y="0" z="2" />
        </definition>
            <list name="GeometryForNormals">
                <setactive drawpoints="0" drawshapes="1"/>
                <setshapemode>actual | bound</setshapemode>
                <setmkbound mk="0" />
                <drawextrude closed="false">
                    <point x="0.005" y="-0.1" z="0.7" />
                    <point x="0.005" y="-0.1" z="0.005" />
                    <point x="0.85" y="-0.1" z="0.005" />
                    <point x="1.1" y="-0.1" z="0.255" />
                    <point x="1.35" y="-0.1" z="0.005" />
                    <point x="2.195" y="-0.1" z="0.005" />
                    <point x="2.195" y="-0.1" z="0.7" />
                    <extrude x="0" y="0.2" z="0" />
                </drawextrude>
                <shapeout file="hdp" />
                <resetdraw />
            </list>
            <mainlist>
                <runlist name="GeometryForNormals" />
                <setdrawmode mode="full" />
                <setmkbound mk="0" />
                <drawextrude closed="false">
                    <point x="0" y="0" z="0.7" />
                    <point x="0" y="0" z="0" />
                    <point x="0.85" y="0" z="0" />
                    <point x="1.1" y="0" z="0.25" />
                    <point x="1.35" y="0" z="0" />
                    <point x="2.2" y="0" z="0" />
                    <point x="2.2" y="0" z="0.7" />
                    <extrude x="0" y="0.1" z="0" />
                    <layers vdp="0,-0.5,-1,-1.5,-2,-3" />
                </drawextrude>
                <!-- Fluid particles -->
                <setmkfluid mk="0" />
                <fillbox x="0.1" y="0" z="0.1">
                    <modefill>void</modefill>
                    <point x="-0.1" y="-1" z="-0.1" />
                    <size x="2.3" y="2" z="0.6" />
                </fillbox>
            </mainlist>
    </geometry>
    <normals>
        <distanceh value="3.0" comment="Maximum distance (H*distanceh) to compute normals data (default=2)" />
        <geometryfile file="[CaseName]_hdp_Actual.vtk" comment="File with boundary geometry (VTK format)" />
    </normals>
<execution>
        <parameter key="SavePosDouble" value="0" comment="Saves particle position using double precision (default=0)" />
        <parameter key="Boundary" value="2" comment="Boundary method 1:DBC, 2:mDBC (default=1)" />
        <parameter key="SlipMode" value="2" comment="Boundary slip mode 1:Vel=0, 2:No-Slip (default=1)"/>
        <parameter key="StepAlgorithm" value="2" comment="Step Algorithm 1:Verlet, 2:Symplectic (default=1)" />
        <parameter key="VerletSteps" value="40" comment="Verlet only: Number of steps to apply Euler timestepping (default=40)" />
        <parameter key="Kernel" value="2" comment="Interaction Kernel 1:Cubic Spline, 2:Wendland (default=2)" />
        <parameter key="ViscoTreatment" value="1" comment="Viscosity formulation 1:Artificial, 2:Laminar+SPS, 3:Laminar (default=1)" />
        <parameter key="Visco" value="0.01" comment="Viscosity value. Typically 0.01 for Artificial and 1e-6 m^2/s (water kinematic viscosity) for Laminar" />
        <parameter key="ViscoBoundFactor" value="1" comment="Multiply viscosity value with boundary (default=1)" />
        <parameter key="DensityDT" value="3" comment="Density Diffusion Term 0:None, 1:Molteni, 2:Fourtakas, 3:Fourtakas(full) (default=0)" />
        <parameter key="DensityDTvalue" value="0.1" comment="DDT value (default=0.1)" />
        <parameter key="Shifting" value="0" comment="Shifting mode 0:None, 1:Ignore bound, 2:Ignore fixed, 3:Full (default=0)" />
        <parameter key="ShiftCoef" value="-2" comment="Coefficient for shifting computation (default=-2)" />
        <parameter key="ShiftTFS" value="1.5" comment="Threshold to detect free surface. Typically 1.5 for 2D and 2.75 for 3D (default=0)" />
        <parameter key="RigidAlgorithm" value="1" comment="Rigid Algorithm 0:collision-free, 1:SPH, 2:DEM, 3:Chrono (default=1)" />
        <parameter key="FtPause" value="0.0" comment="Time to freeze the floatings at simulation start (warmup) (default=0)" units_comment="seconds" />
        <parameter key="CoefDtMin" value="0.05" comment="Coefficient to calculate minimum time step dtmin=coefdtmin*h/speedsound (default=0.05)" />
        <parameter key="DtIni" value="0" comment="Initial time step. Use 0 to default use (default=h/speedsound)" units_comment="seconds" />
        <parameter key="DtMin" value="0" comment="Minimum time step. Use 0 to default use (default=coefdtmin*h/speedsound)" units_comment="seconds" />
        <parameter key="DtFixed" value="0" comment="Fixed Dt value. Use 0 to disable (default=disabled)" units_comment="seconds" />
        <parameter key="DtFixedFile" value="NONE" comment="Dt values are loaded from file. Use NONE to disable (default=disabled)" units_comment="milliseconds (ms)" />
        <parameter key="DtAllParticles" value="0" comment="Velocity of particles used to calculate DT. 1:All, 0:Only fluid/floating (default=0)" />
        <parameter key="TimeMax" value="15" comment="Time of simulation" units_comment="seconds" />
        <parameter key="TimeOut" value="0.01" comment="Time out data" units_comment="seconds" />
        <parameter key="PartsOutMax" value="1" comment="Allowed %/100 of fluid particles out the domain (default=1)" units_comment="decimal" />
        <parameter key="RhopOutMin" value="700" comment="Minimum rhop valid (default=700)" units_comment="kg/m^3" />
        <parameter key="RhopOutMax" value="1300" comment="Maximum rhop valid (default=1300)" units_comment="kg/m^3" />
        <simulationdomain comment="Defines domain of simulation (default=Uses minimum and maximum position of the generated particles)">
            <posmin x="default" y="default" z="default" comment="e.g.: x=0.5, y=default-1, z=default-10%" />
            <posmax x="default" y="default" z="default" />
        </simulationdomain>
    </parameters>
</execution>
</case>
```

mDBC

```
<parameter key="Boundary" value="2" comment="Boundary method 1:DBC, 2:mDBC (default=1)" />
```

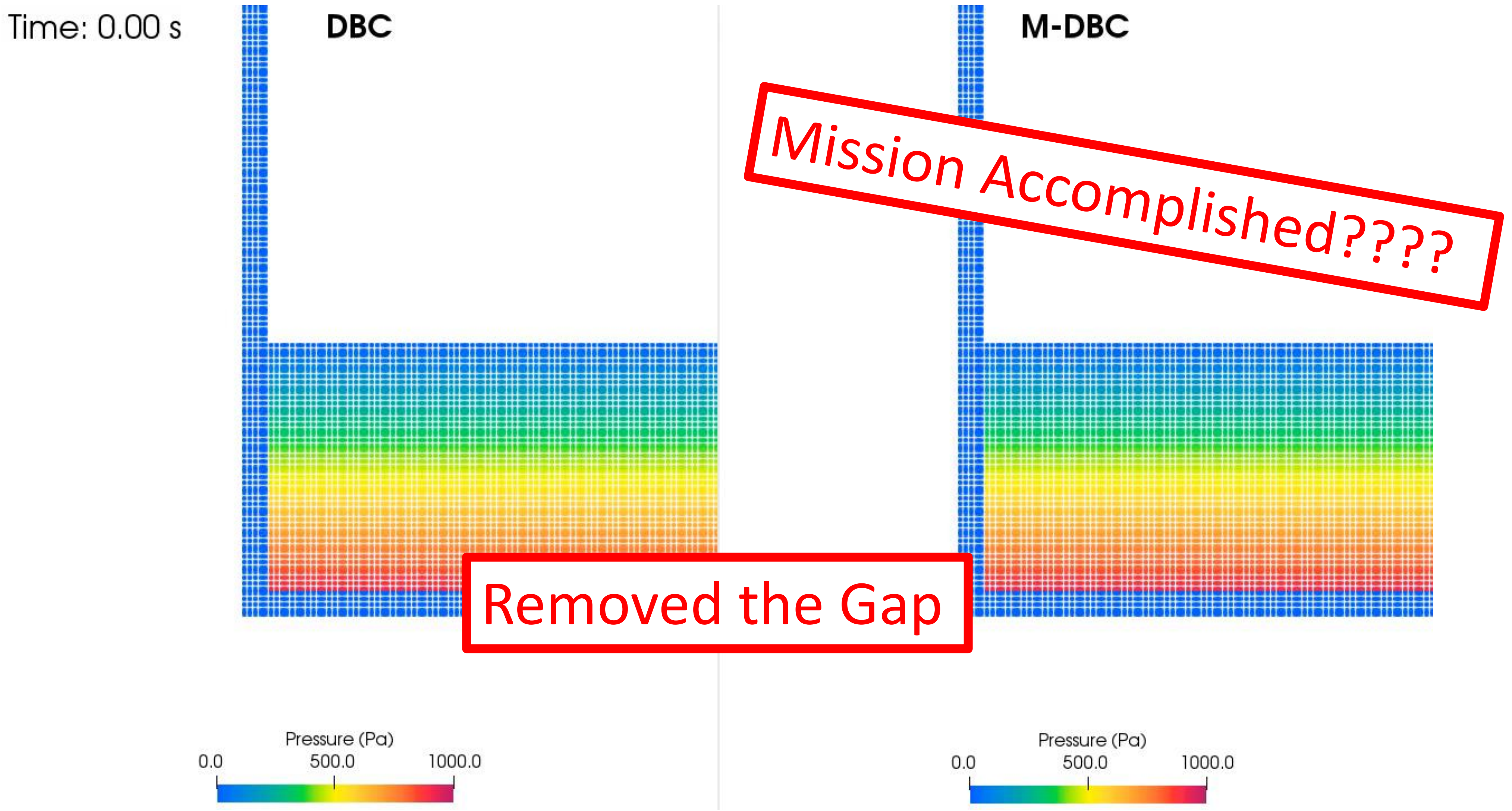




Improved pressure/density in the fluid and boundary

mDBC

```
<parameter key="Boundary" value="2" comment="Boundary method 1:DBC, 2:mDBC (default=1)" />
```



mDBC <parameter key="Boundary" value="2" comment="Boundary method 1:DBC, 2:mDBC (default=1)" />

Not Quite

Drawbacks or issues

Only vel = 0 was available as a no-slip option in the DualSPHysics release and no free slip option available (kept behind exceptions)

mDBC could only run on the predictor step when using the Symplectic time step scheme in DualSPHysics

No free surface detection included to “turn off” boundary particles when very few fluid particles

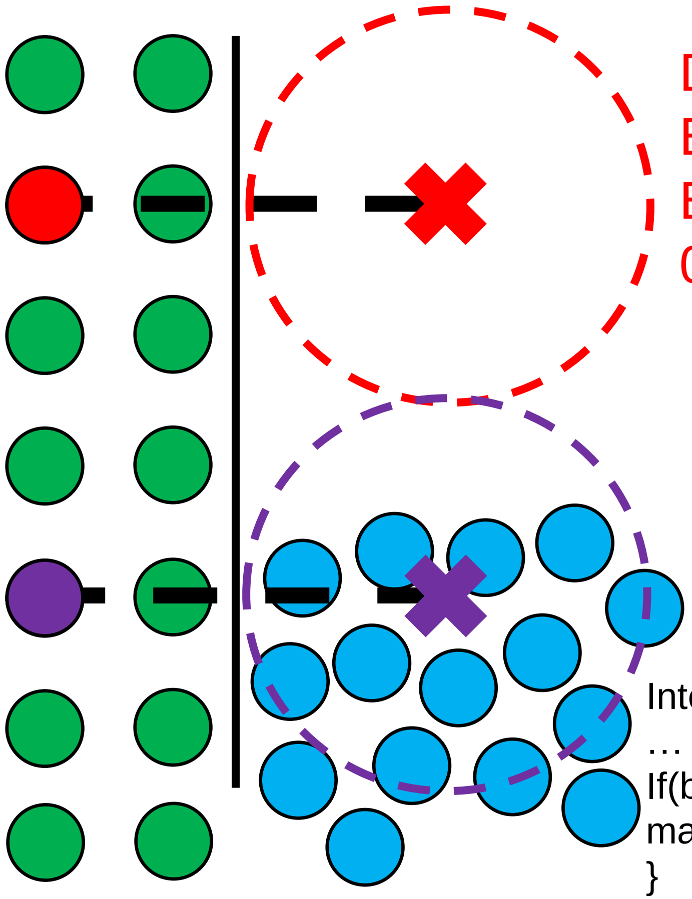
- Issues with isolated fluid particles
- Particles dancing on beach

Only check if the matrix is invertible

mDBC updates – numerical checks

Boundary particles turned on or off

Check if a ghost node is submerged through the divergence of position



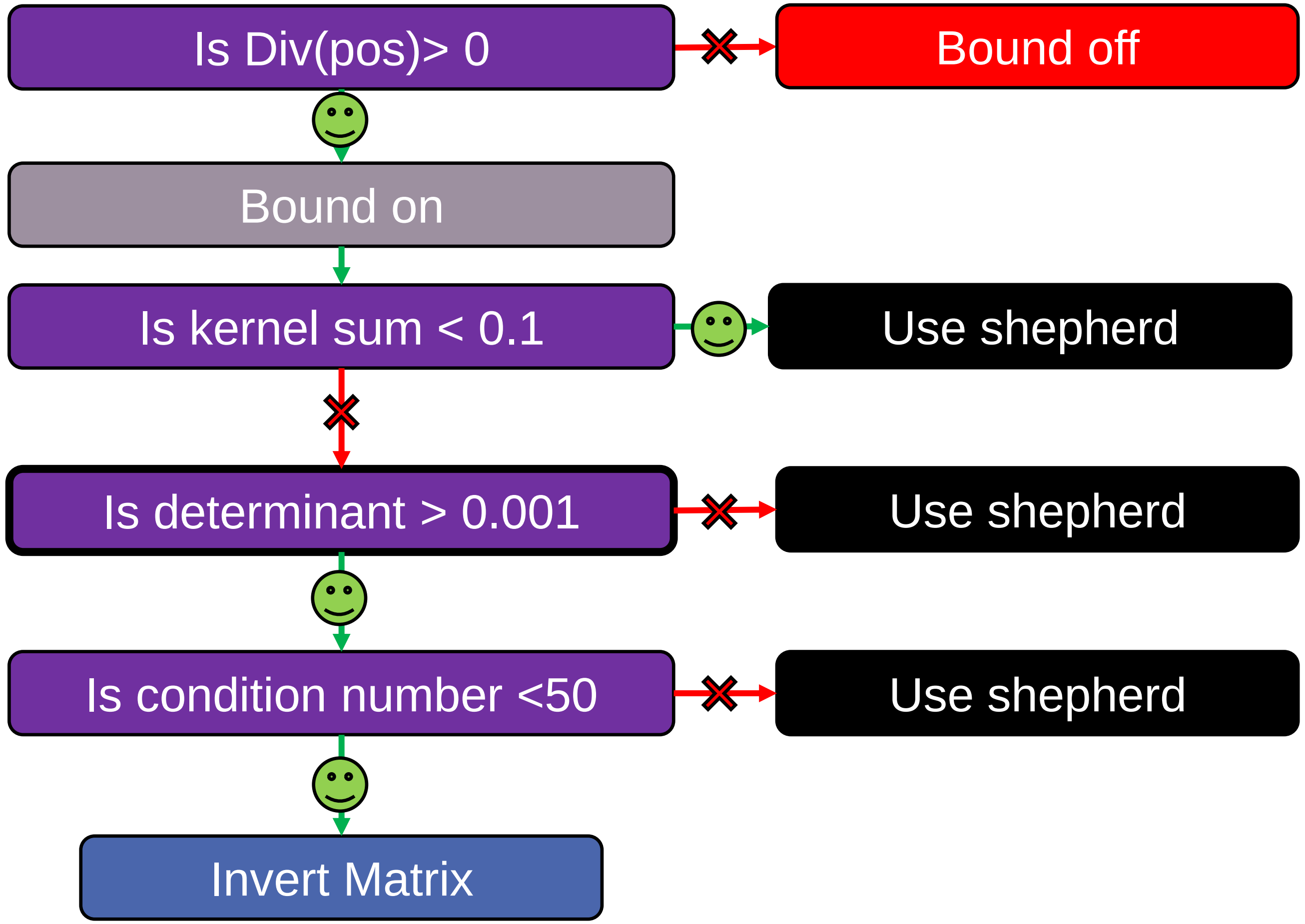
Div(pos)<=0
Bound off
Boundonoff=0

Div(pos)>0
Bound on
Boundonoff=1

```
InteractionForcesFluid
…
If(boundp2){
massp2 = Boundonoff*massbound
}
```

Matrix checks

Use the condition number of the matrix to decide on the correction method used

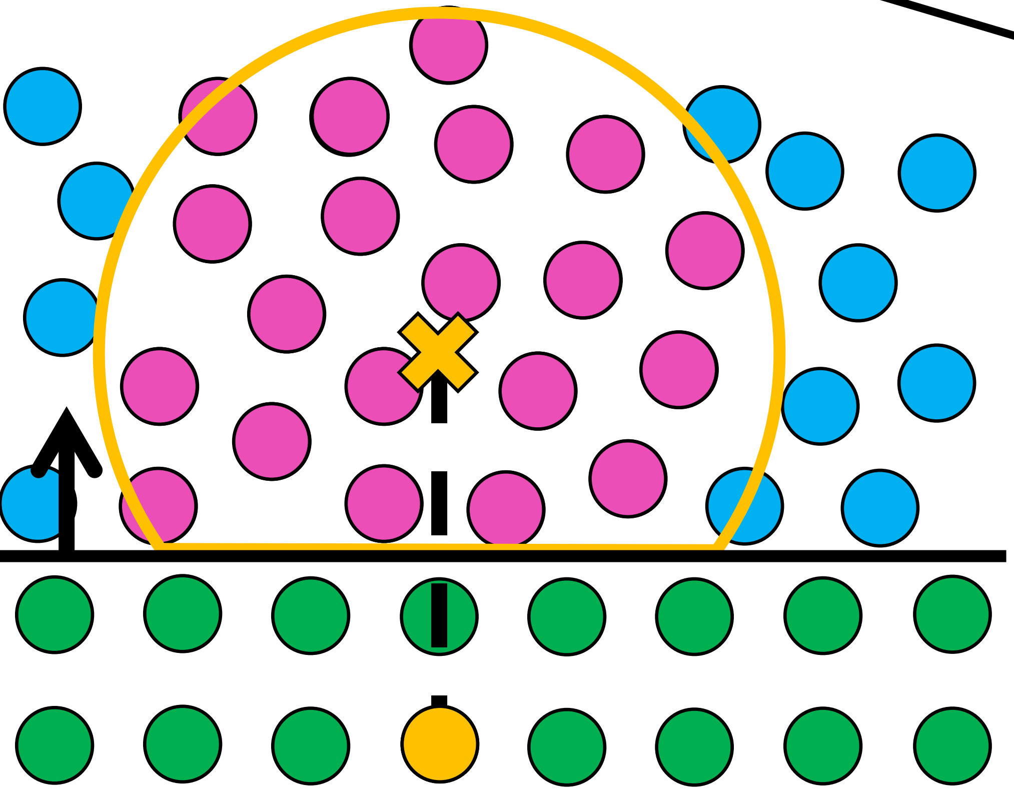


mDBC updates – pressure cloning

Use the density and the clone particle procedure of Antuono et al 2023.

Matrix inversion
Invert the matrix $\boldsymbol{A}_g$ to find the density -> ρ_g

Shepherd
Use a shepherd sum to find the density -> ρ_g



ρ_g

$$P_g = c_0^2(\rho_g - \rho_0)$$

Use and approximation to the momentum equation

$$P_b = P_g + \rho_0[(\boldsymbol{g} - \boldsymbol{a}_b)\cdot \boldsymbol{n_b}][(\boldsymbol{x}_g - \boldsymbol{x}_b)\cdot \boldsymbol{n_b}]$$

$$\rho_b = \rho_0 + \frac{P_b}{c_0^2}$$

Antuono M., Pilloton C., Colagrossi A., Durante D. Clone particles: A simplified technique to enforce solid boundary conditions in SPH Comput. Methods Appl. Mech. Engrg., 409 (2023), Article 115973

mDBC updates – no-slip

$$\boldsymbol{u}_g = \frac{\sum_j \boldsymbol{u}_j V_j W_{gj}}{\sum_j V_j W_{gj}}$$

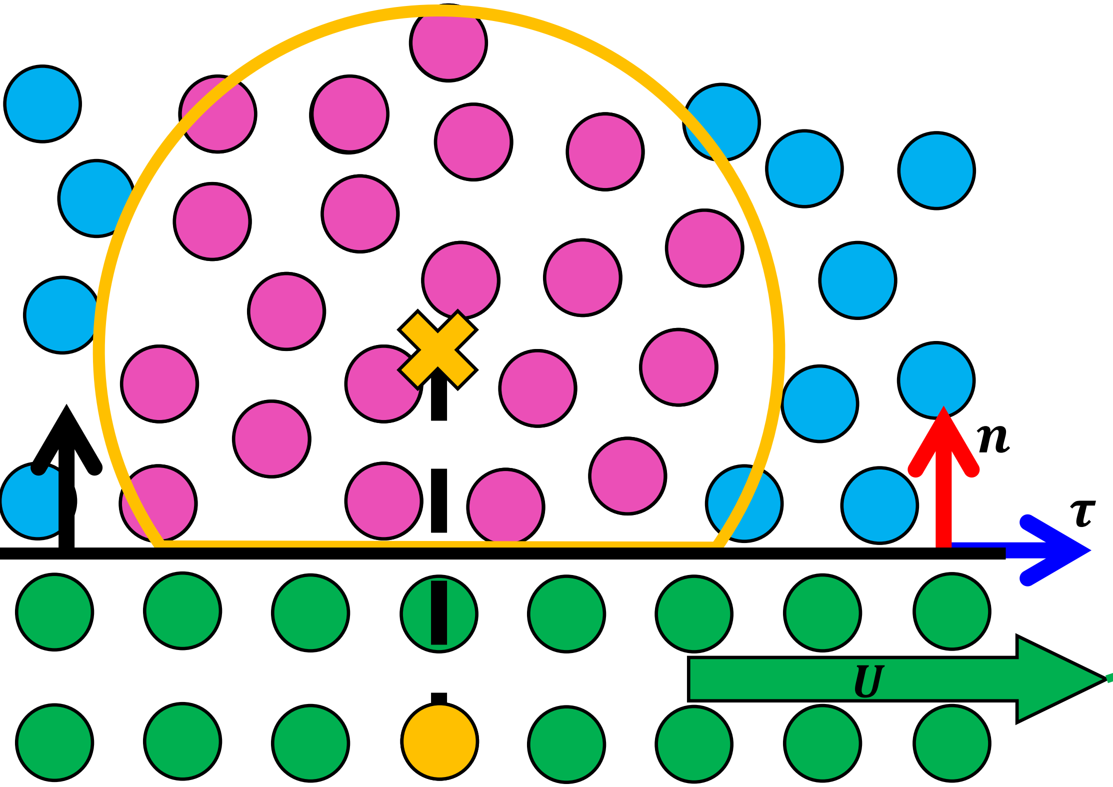
$$\frac{d\boldsymbol{u}_i}{dt} = -\sum_j m_j \left(\frac{P_i + P_j}{\rho_i \, \rho_j}\right) \nabla W_{ij} + \sum_j \frac{4\nu \, m_j \, \boldsymbol{r}_{ij} \cdot \nabla W_{ij}}{(\rho_i + \rho_j)\,(\boldsymbol{r}_{ij}^2 + 0.01h^2)} \boldsymbol{u}_{ij} + \boldsymbol{g}_i$$

$$\boldsymbol{u}_b = 2\boldsymbol{U} - \boldsymbol{u}_g$$

$$\boldsymbol{u}_b \cdot \boldsymbol{\tau} = (2\boldsymbol{U} - \boldsymbol{u}_g) - \left((2\boldsymbol{U} - \boldsymbol{u}_g) \cdot \boldsymbol{n}\right)$$

~~$\boldsymbol{u}_b \cdot \boldsymbol{n} = (2\boldsymbol{U} - \boldsymbol{u}_g) \cdot \boldsymbol{n}$~~

$$\frac{d\rho_i}{dt} = \rho_i \sum_j \frac{m_j}{\rho_j} \boldsymbol{u_{ij}} \cdot \nabla W_{ij} + DDT$$



mDBC updates – extra details

The new mDBC now updates the boundary particle density and velocity during **both the predictor and correct steps of the Symplectic time step scheme** where the previous approach only updated during the predictor step

Pressure of **boundary particles is now allowed to become negative** where the previous approaches would set the pressure to zero in the event of negative pressure. This is key for flows past objects where negative pressure keeps the flow attached to the back of the object

It is essential when using the new mDBC options that the **ViscousBoundFactor is given a value of 1** regardless of viscosity treatment used

```
<parameter key="ViscoTreatment" value="2" comment="Viscosity formulation 1:Artificial, 2:Laminar+SPS, 3:Lamina
<parameter key="Visco" value="0.1" comment="Viscosity value. Typically 0.01 for Artificial and 1e-6 m^2/s (wat
<parameter key="ViscoBoundFactor" value="1" comment="Multiply viscosity value with boundary (default=1)" />
```

mDBC updates – new xml option

```
<parameter key="Boundary" value="2" comment="Boundary method 1:DBC, 2:mDBC (default=1)" />
<parameter key="SlipMode" value="2" comment="Boundary slip mode 1:Vel=0, 2:No-Slip (default=1)"/>
```

There is a new flag included in the xml files to activate all these features, **`SlipMode`**:

- To run DBC, select options:

`Boundary=1: DBC`

- To run the original mDBC, select options:

`Boundary=2: mDBC, SlipMode=1: Vel=0`

This is a completely unmodified version of mDBC without any added numerical checks

- To run the new mDBC, select options: (BETA)

`Boundary=2: mDBC, SlipMode=2: No-Slip`

The new mDBC with extra numerical checks, no-slip mirrored velocity and pressure cloning

Testcases


```xml
<?xml version="1.0" encoding="UTF-8" ?>
<case>
    <casedef>
        <constantsdef>
            <gravity x="0" y="0" z="-9.81" comment="Gravitational acceleration" units_comment="m/s^2" />
            <rhop0 value="1000" comment="Reference density of the fluid" units_comment="kg/m^3" />
            <rhopgradient value="2" comment="Initial density gradient 1:Rhop0, 2:Water column, 3:Max. water height (default=2)" />
            <hswl value="0" auto="true" comment="Maximum still water level to calculate speedofsound using coefsound" units_comment="metres (m)" />
            <gamma value="7" comment="Politropic constant for water used in the state equation" />
            <speedsystem value="0" auto="true" comment="Maximum system speed (by default the dam-break propagation is used)" />
            <coefsound value="20" comment="Coefficient to multiply speedsystem" />
            <speedsound value="0" auto="true" comment="Speed of sound to use in the simulation (by default speedofsound=coefsound*speedsystem)" />
            <hdp value="1.3" comment="Alternative option to calculate the smoothing length (h=hdp*dp)" />
            <cflnumber value="0.2" comment="Coefficient to multiply dt" />
        </constantsdef>
        <mkconfig boundcount="240" fluidcount="9" />
        <geometry>
            <definition dp="0.01" units_comment="metres (m)">
                <pointmin x="-1" y="0" z="-1" />
                <pointmax x="2.5" y="0" z="2" />
            </definition>
            <commands>
                <list name="GeometryForNormals">
                    <setactive drawpoints="0" drawshapes="1"/>
                    <setshapemode>actual | bound</setshapemode>
                    <setmkbound mk="0" />
                    <drawextrude closed="false">
                        <point x="0.005" y="-0.1" z="0.7" />
                        <point x="0.005" y="-0.1" z="0.005" />
                        <point x="0.85" y="-0.1" z="0.005" />
                        <point x="1.1" y="-0.1" z="0.255" />
                        <point x="1.35" y="-0.1" z="0.005" />
                        <point x="2.195" y="-0.1" z="0.005" />
                        <point x="2.195" y="-0.1" z="0.7" />
                        <extrude x="0" y="0.2" z="0" />
                    </drawextrude>
                    <shapeout file="hdp" />
                    <resetdraw />
                </list>
                <mainlist>
                    <!-- Actual geometry at dp/2 -->
                    <runlist name="GeometryForNormals" />
                    <!-- Boundary particles -->
                    <setdrawmode mode="full" />
                    <setmkbound mk="0" />
                    <drawextrude closed="false">
                        <point x="0" y="0" z="0.7" />
                        <point x="0" y="0" z="0" />
                        <point x="0.85" y="0" z="0" />
                        <point x="1.1" y="0" z="0.25" />
                        <point x="1.35" y="0" z="0" />
                        <point x="2.2" y="0" z="0" />
                        <point x="2.2" y="0" z="0.7" />
                        <extrude x="0" y="0.1" z="0" />
                        <layers vdp="0,-0.5,-1,-1.5,-2,-3" />
                    </drawextrude>
                    <!-- Fluid particles -->
                    <setmkfluid mk="0" />
                    <fillbox x="0.1" y="0" z="0.1">
                        <modefill>void</modefill>
                        <point x="-0.1" y="-1" z="-0.1" />
                        <size x="2.3" y="2" z="0.6" />
                    </fillbox>
                </mainlist>
            </commands>
        </geometry>
        <normals>
            <distanceh value="3.0" comment="Maximum distance (H*distanceh) to compute normals data (default=2)" />
            <geometryfile file="[CaseName]_hdp_Actual.vtk" comment="File with boundary geometry (VTK format)" />
        </normals>
    </casedef>
    <execution>
        <parameters>
            <parameter key="SavePosDouble" value="0" comment="Saves particle position using double precision (default=0)" />
            <parameter key="Boundary" value="2" comment="Boundary method 1:DBC, 2:mDBC (default=1)" />
            <parameter key="SlipMode" value="2" comment="Boundary slip mode 1:Vel=0, 2:No-Slip (default=1)"/>
            <parameter key="StepAlgorithm" value="2" comment="Step Algorithm 1:Verlet, 2:Symplectic (default=1)" />
            <parameter key="VerletSteps" value="40" comment="Verlet only: Number of steps to apply Euler timestepping (default=40)" />
            <parameter key="Kernel" value="2" comment="Interaction Kernel 1:Cubic Spline, 2:Wendland (default=2)" />
            <parameter key="ViscoTreatment" value="1" comment="Viscosity formulation 1:Artificial, 2:Laminar+SPS, 3:Laminar (default=1)" />
            <parameter key="Visco" value="0.01" comment="Viscosity value. Typically 0.01 for Artificial and 1e-6 m^2/s (water kinematic viscosity) for L
            <parameter key="ViscoBoundFactor" value="1" comment="Multiply viscosity value with boundary (default=1)" />
            <parameter key="DensityDT" value="3" comment="Density Diffusion Term 0:None, 1:Molteni, 2:Fourtakas, 3:Fourtakas(full) (default=0)" />
            <parameter key="DensityDTvalue" value="0.1" comment="DDT value (default=0.1)" />
            <parameter key="Shifting" value="0" comment="Shifting mode 0:None, 1:Ignore bound, 2:Ignore fixed, 3:Full (default=0)" />
            <parameter key="ShiftCoef" value="-2" comment="Coefficient for shifting computation (default=-2)" />
            <parameter key="ShiftTFS" value="1.5" comment="Threshold to detect free surface. Typically 1.5 for 2D and 2.75 for 3D (default=0)" />
            <parameter key="RigidAlgorithm" value="1" comment="Rigid Algorithm 0:collision-free, 1:SPH, 2:DEM, 3:Chrono (default=1)" />
            <parameter key="FtPause" value="0.0" comment="Time to freeze the floatings at simulation start (warmup) (default=0)" units_comment="seconds"
            <parameter key="CoefDtMin" value="0.05" comment="Coefficient to calculate minimum time step dtmin=coefdtmin*h/speedsound (default=0.05)" />
            <parameter key="DtIni" value="0" comment="Initial time step. Use 0 to default use (default=h/speedsound)" units_comment="seconds" />
            <parameter key="DtMin" value="0" comment="Minimum time step. Use 0 to default use (default=coefdtmin*h/speedsound)" units_comment="seconds"
            <parameter key="DtFixed" value="0" comment="Fixed Dt value. Use 0 to disable (default=disabled)" units_comment="seconds" />
            <parameter key="DtFixedFile" value="NONE" comment="Dt values are loaded from file. Use NONE to disable (default=disabled)" units_comment="mi
            <parameter key="DtAllParticles" value="0" comment="Velocity of particles used to calculate DT. 1:All, 0:Only fluid/floating (default=0)" />
            <parameter key="TimeMax" value="15" comment="Time of simulation" units_comment="seconds" />
            <parameter key="TimeOut" value="0.01" comment="Time out data" units_comment="seconds" />
            <parameter key="PartsOutMax" value="1" comment="Allowed %/100 of fluid particles out the domain (default=1)" units_comment="decimal" />
            <parameter key="RhopOutMin" value="700" comment="Minimum rhop valid (default=700)" units_comment="kg/m^3" />
            <parameter key="RhopOutMax" value="1300" comment="Maximum rhop valid (default=1300)" units_comment="kg/m^3" />
            <simulationdomain comment="Defines domain of simulation (default=Uses minimum and maximum position of the generated particles)">
                <posmin x="default" y="default" z="default" comment="e.g.: x=0.5, y=default-1, z=default-10%" />
                <posmax x="default" y="default" z="default" />
            </simulationdomain>
        </parameters>
    </execution>
</case>
```

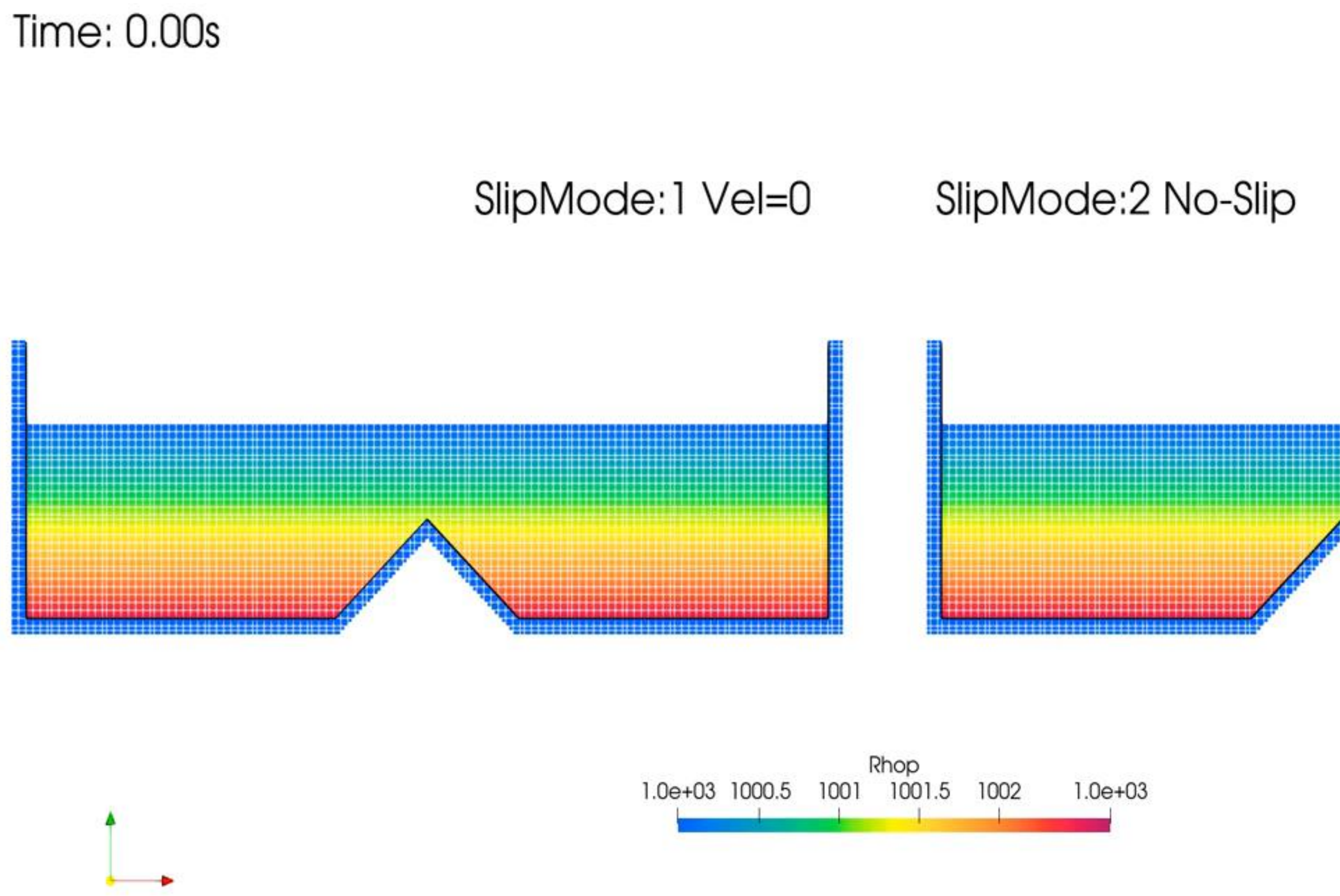
Testcases

Name	Date modified	Type	Size
chrono	18/10/2023 12:32	File folder	
flexstruc	18/10/2023 12:32	File folder	
inletmesh	06/03/2024 12:15	File folder	
inletoutlet			
main			
mdbc			
moordyn			
motion			
mphase_liquidgas			
mphase_nnewtonian			
others			
vresolution			
wavecoupling			

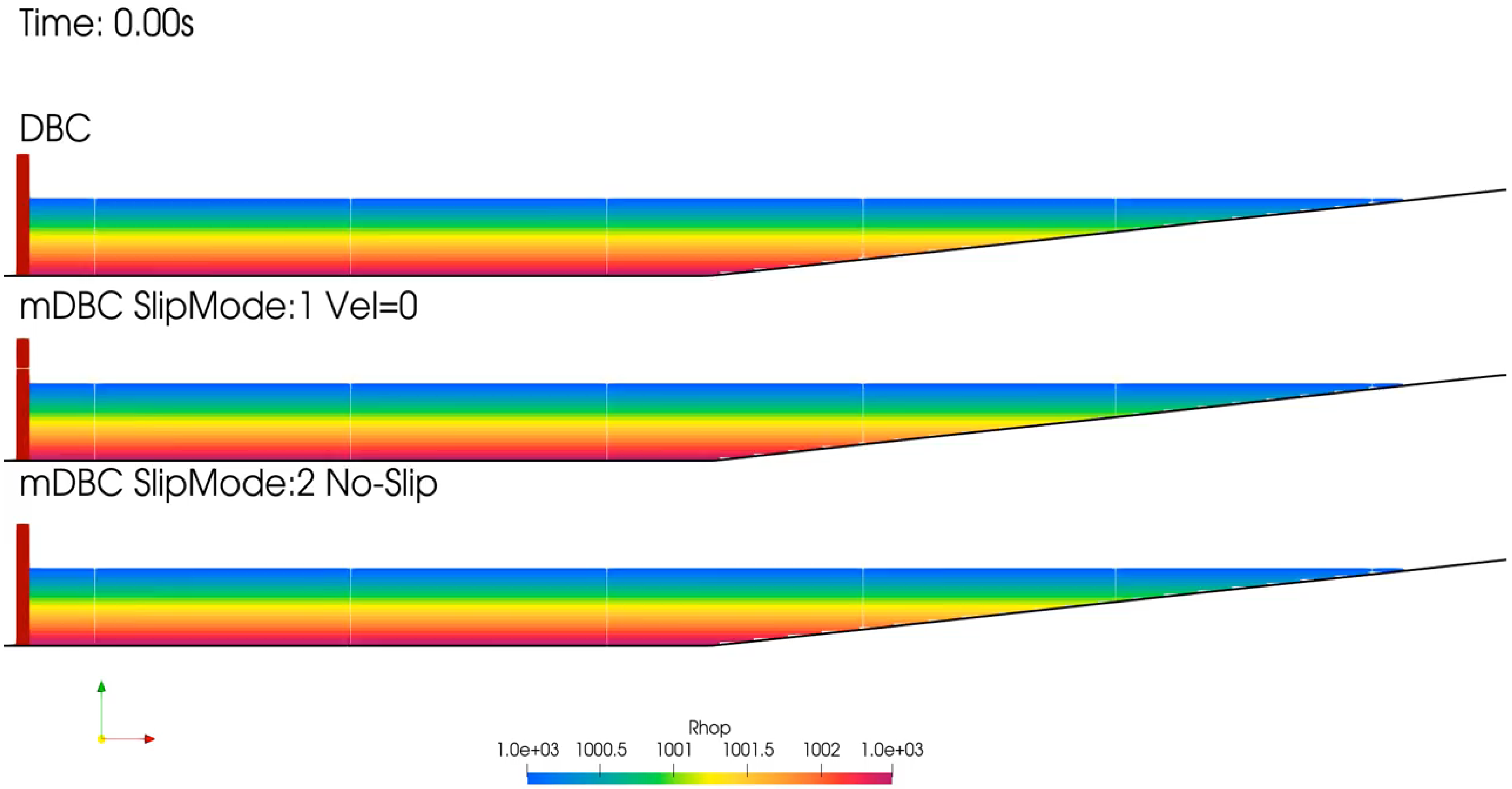
Name	Date modified	Type	Size
_InitializeExamples	06/03/2024 12:15	File folder	
01_StillWedge	11/03/2024 17:49	File folder	
02_Poiseuille	11/03/2024 17:49	File folder	
03_Sloshing	11/03/2024 17:49	File folder	
04_Dambreak	11/03/2024 17:49	File folder	
05_FlowCylinder	11/03/2024 17:49	File folder	
06_WaveTank	08/03/2024 13:26	File folder	
07_WavesCylinder	06/03/2024 12:15	File folder	
08_FloatingWaves	06/03/2024 12:15	File folder	
09_FloatingDuck	06/03/2024 12:15	File folder	
10_NoSlip	06/03/2024 15:05	File folder	

Name	Date modified	Type	Size
101_StillWater	11/03/2024 17:49	File folder	
102_Poiseuille	11/03/2024 17:49	File folder	
103_SloshingTank	08/03/2024 14:29	File folder	
104_DamBreak	11/03/2024 17:49	File folder	
105_FlowCylinder	11/03/2024 17:49	File folder	
106_WaveTank	11/03/2024 17:49	File folder	

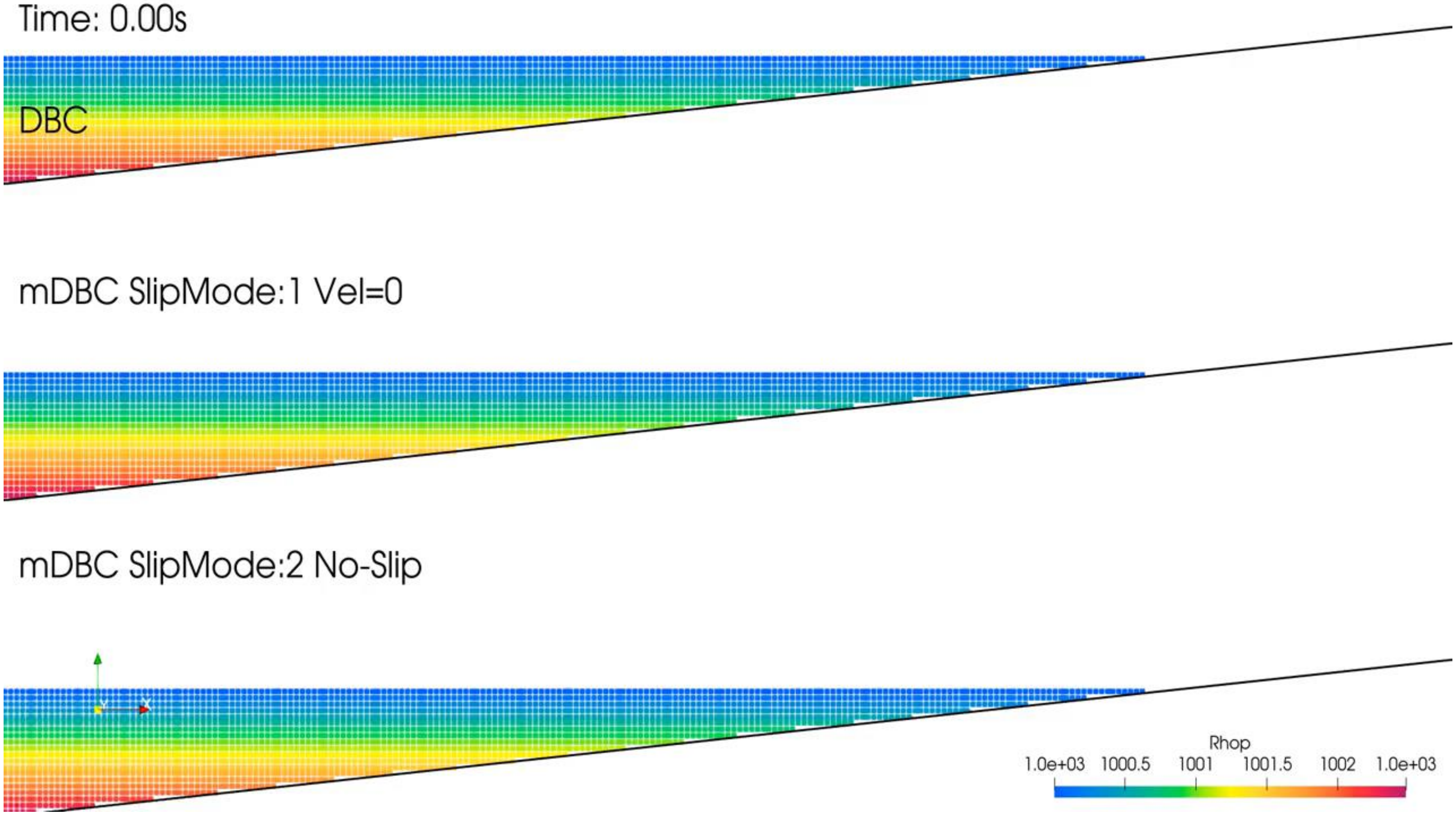
Testcases – Stillwater



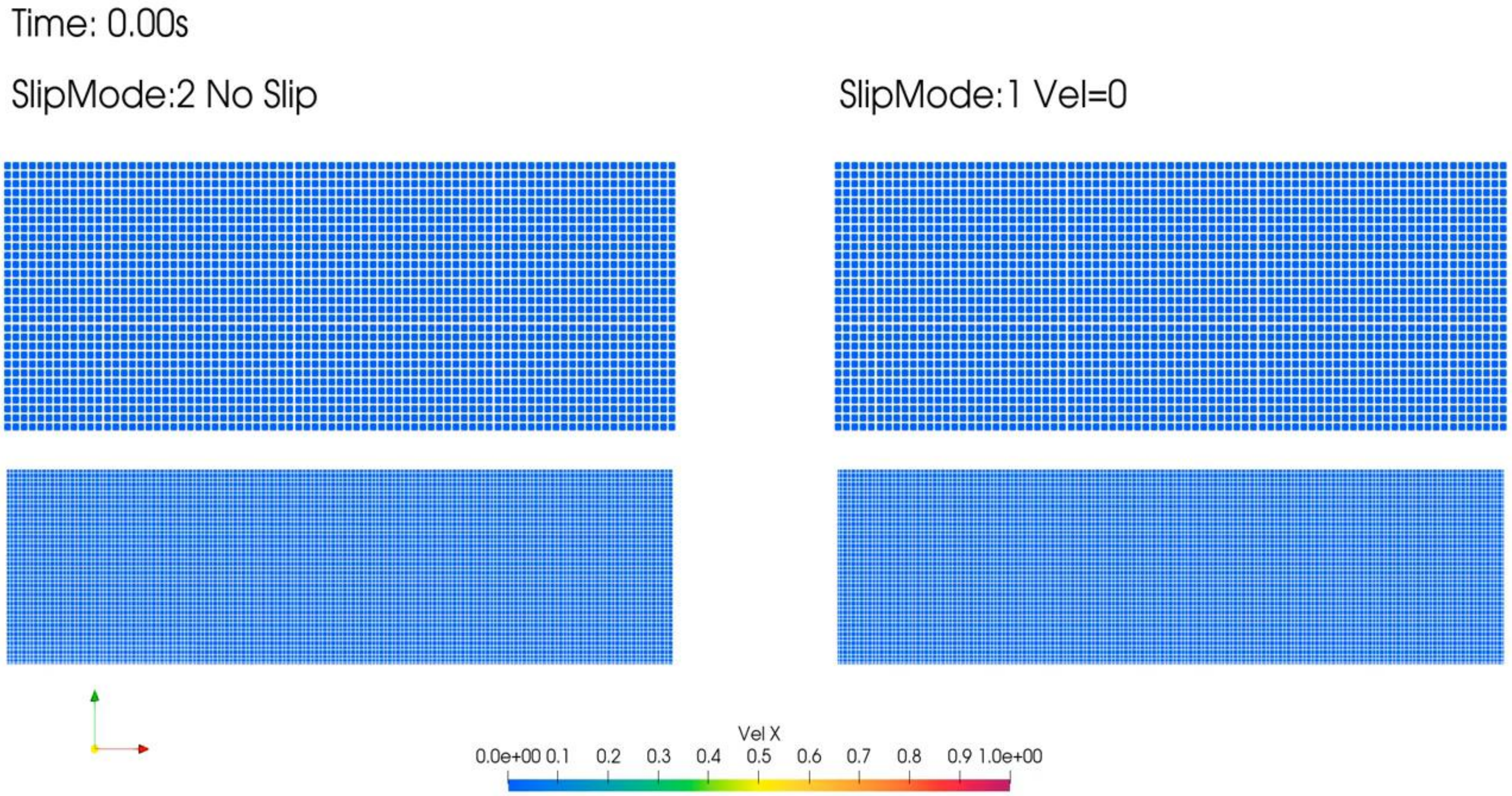
Testcases – 2D Wave tank



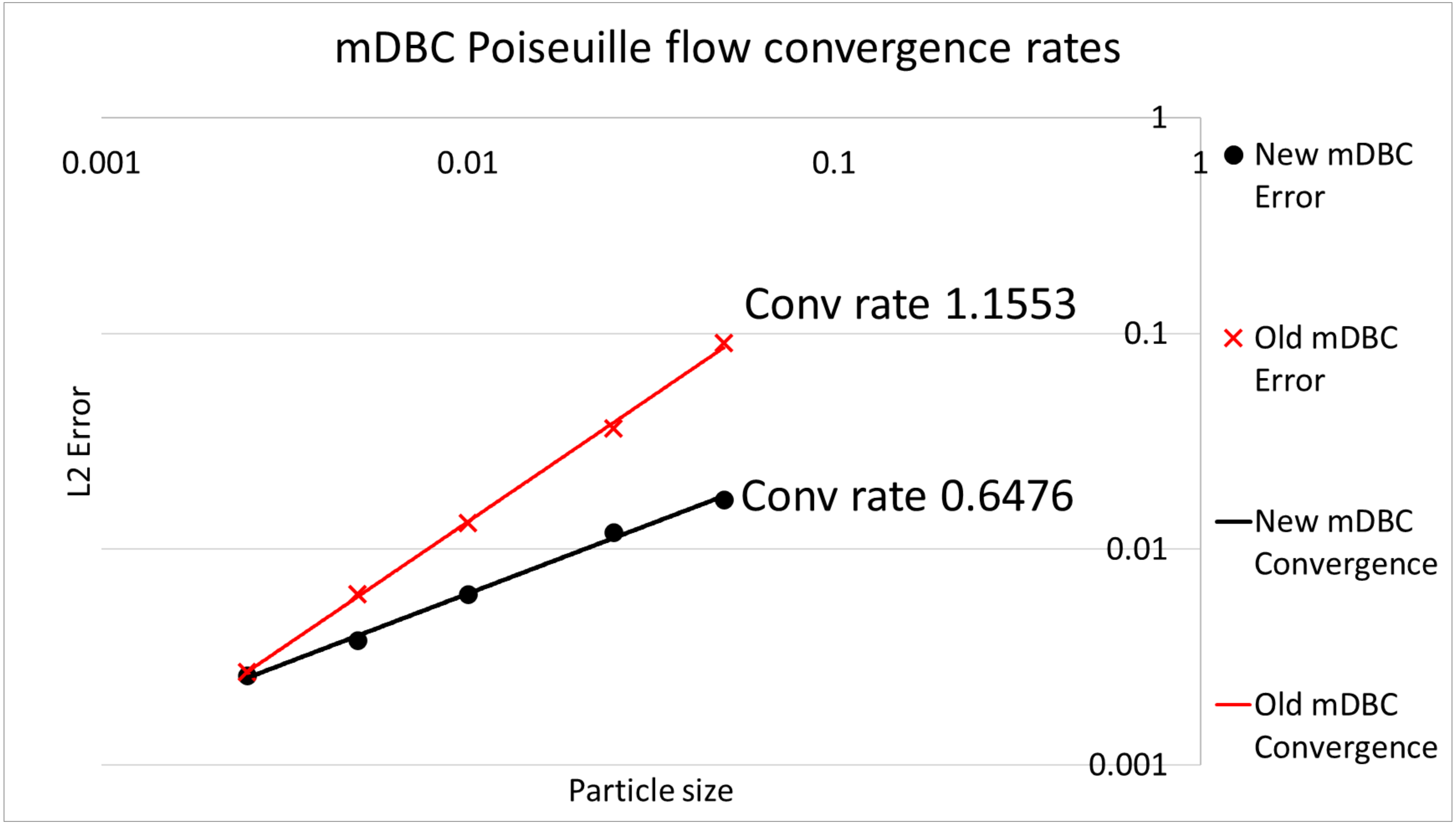
Testcases – 2D Wave tank



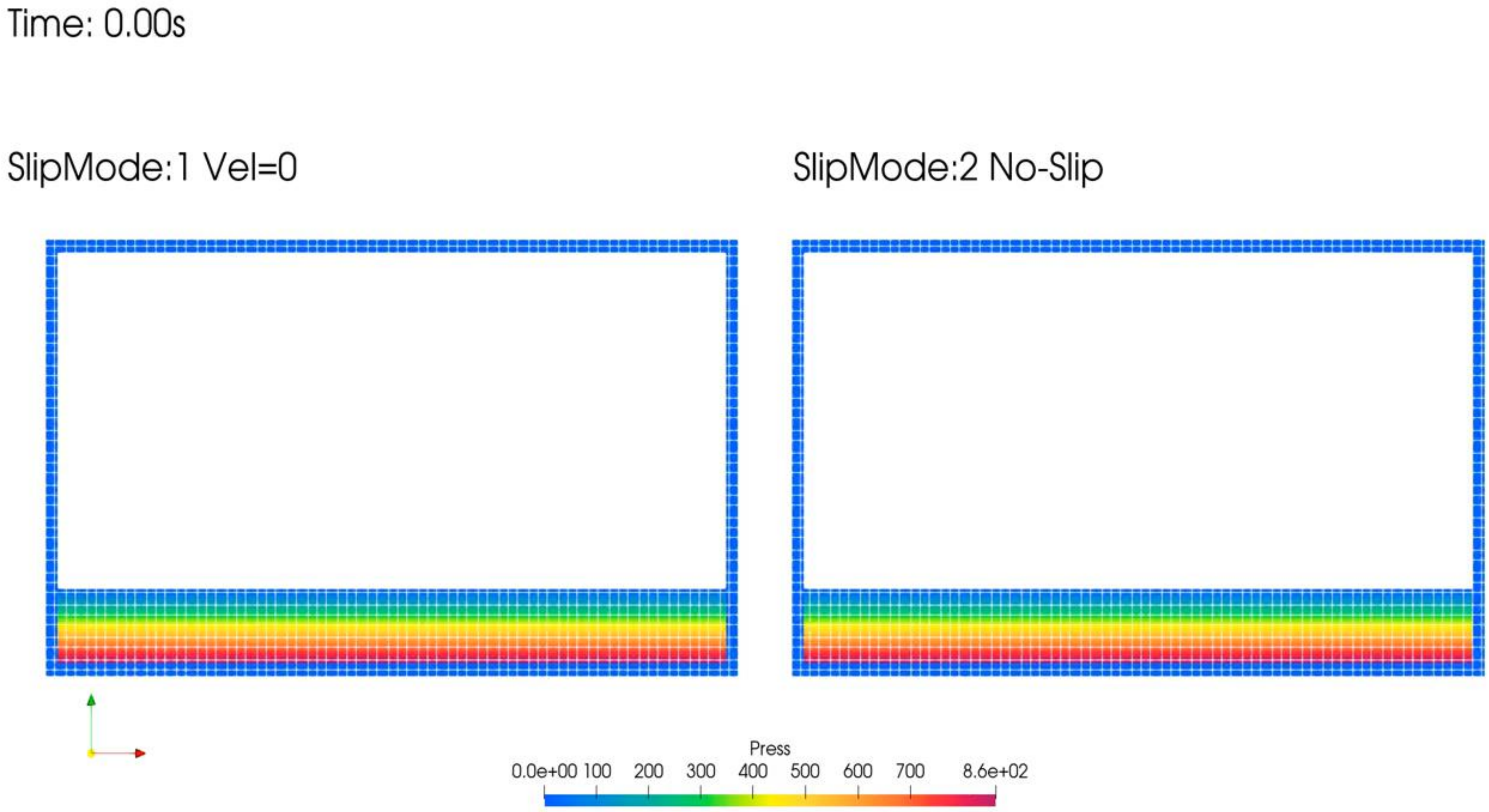
Testcases – Poiseuille flow



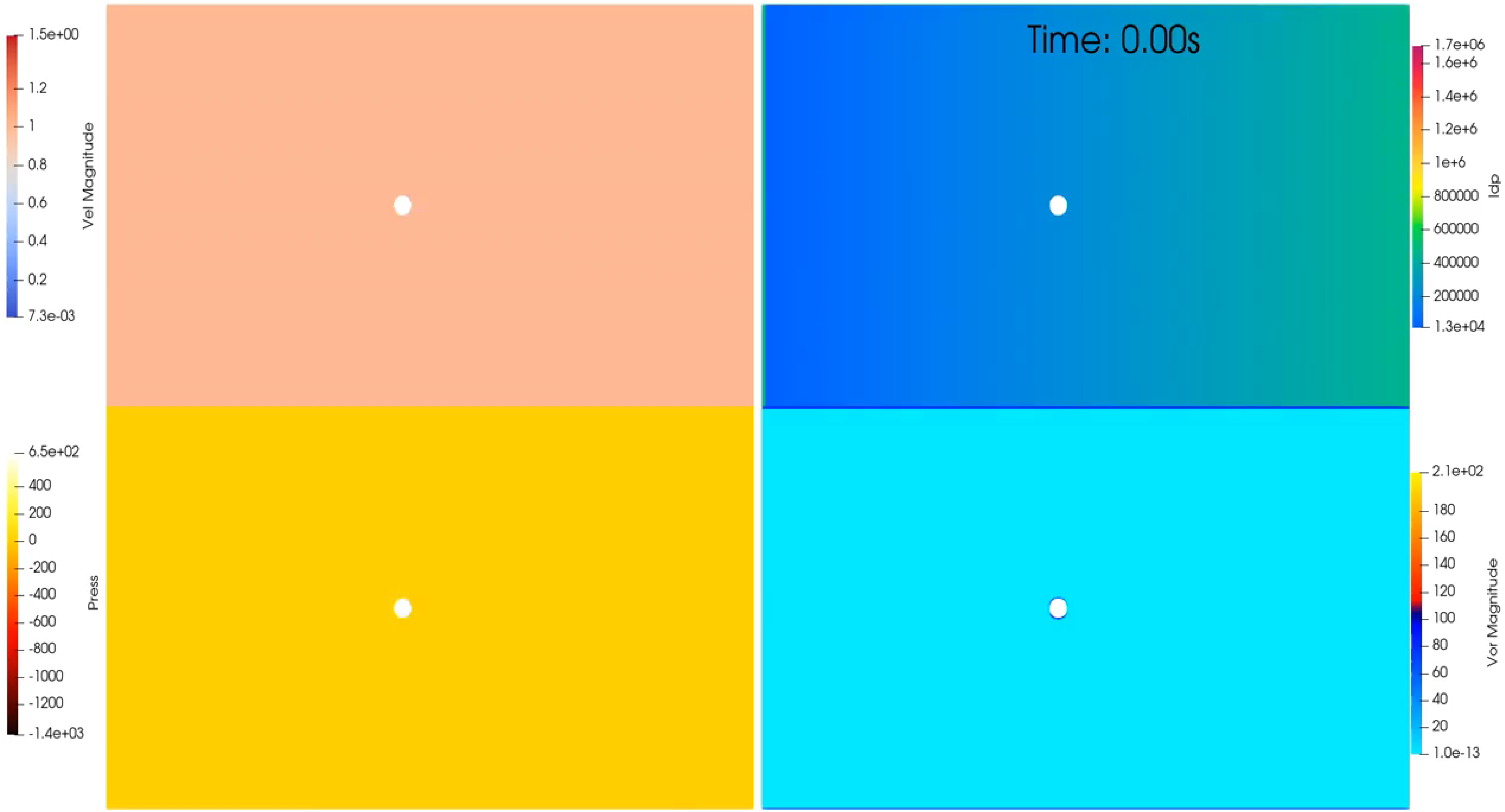
Testcases – Poiseuille flow



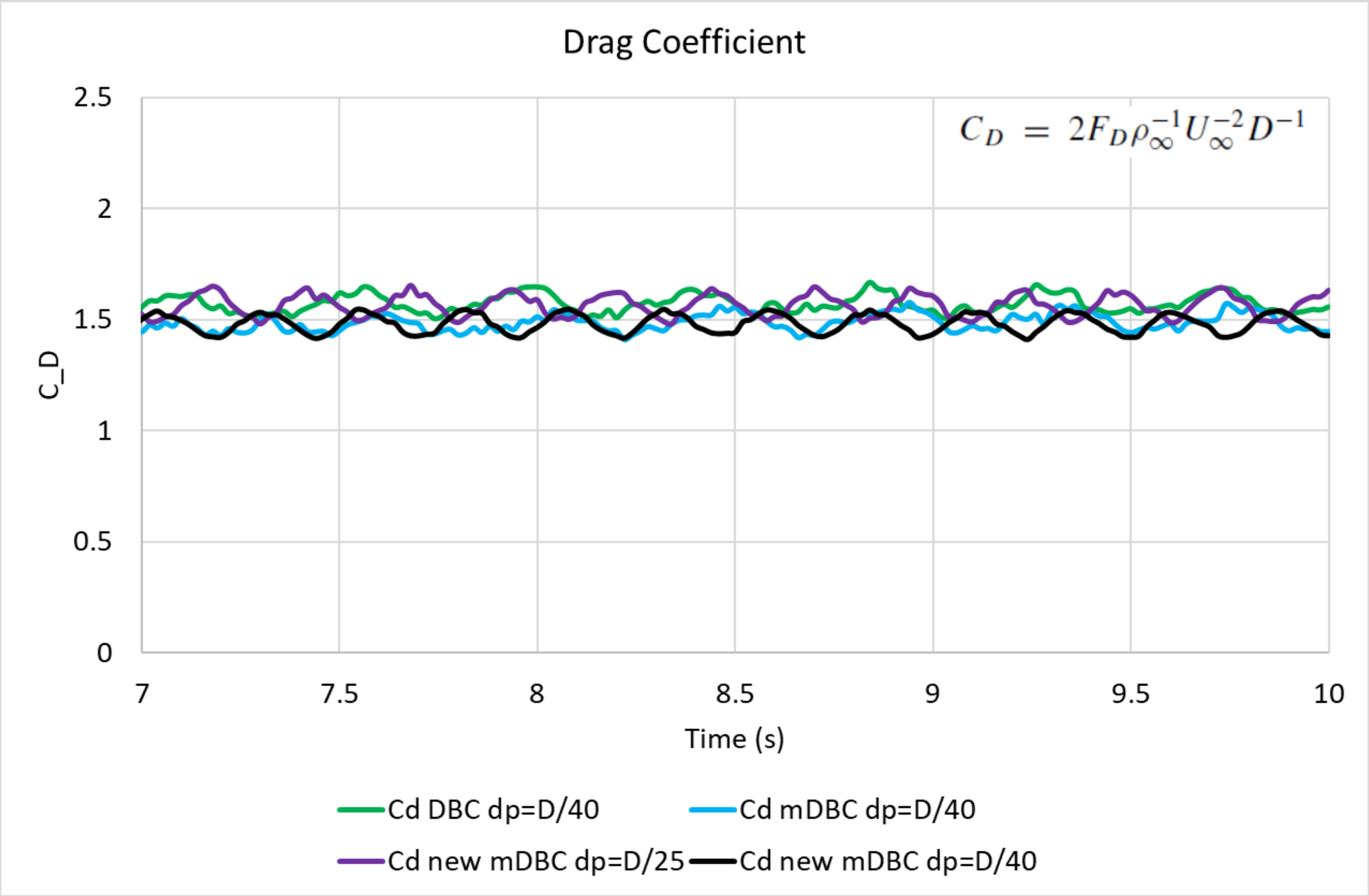
Testcases – Sloshing tank



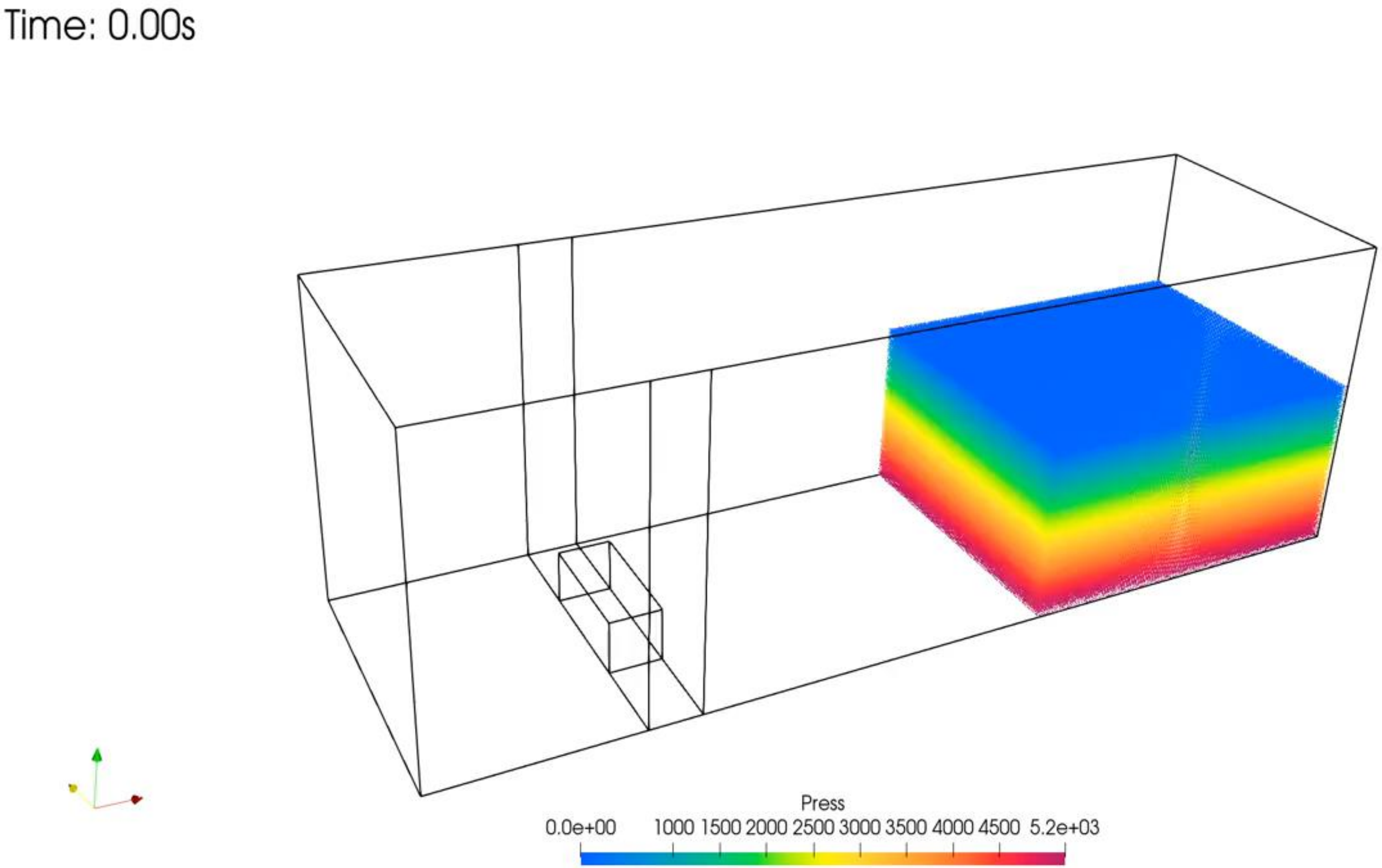
Testcases – Flow past cylinder



Testcases – Flow past cylinder



Testcases – 3D Dambreak



Conclusions

- Highlighted a number of issues with the boundary methods previously included in DualSPHysics

- Developed a new mDBC approach that overcomes many of the issues of the old mDBC method including:
  - Extra numerical checks
  - More physical pressure and density extrapolation method
  - More accurate no-slip condition
  - No more particles dancing on the beach
  - Negative boundary pressures behind obstacles

- Presented a selection of test cases with simple implementation of the new method

Future work

- Free slip velocity treatment
- Fix any bugs or leaks
- Integration with floating objects, Chrono and Flexible structures

Thanks for your attention – Any questions?