VII International Conference on Particle-Based Methods
PARTICLES 2021
P. Wriggers, M. Bischoff, E. Oñate, M. Bischoff, A. Düster & T. Zohdi (Eds)

COUPLING OF NON-NEWTONIAN MESHLESS FLOW WITH STRUCTURAL SOLVERS

MARTINA BASIC[1], BRANKO BLAGOJEVIC[1], BRANKO KLARIN[1] AND JOSIP BASIC[1]

[1] Faculty of Electrical Engineering, Mechanical Engineering and Naval Architecture
University of Split
R. Boskovica 32, 21000 Split, Croatia
e-mail: mandru00@fesb.hr, www.fesb.hr

Key words: Meshless, non-Newtonian, Rheology, Coupling, Fluid-structure interaction

Abstract. The goal of this study is to introduce a coupling process between a novel meshless scheme for non-Newtonian flows and an arbitrary FEM structural solver. The flow-solving method named Lagrangian Differencing Dynamics (LDD) is based on Lagrangian differences, volume–conservative advection, and direct interaction with triangulated geometry. The flow solver implementation named *Rhoxyz* is coupled with *CalculiX* solver for the structure through a bidirectional coupling tool named *preCICE*. The non-Newtonian solver is validated on a skewed lid-driven cavity experiment, and the coupling scheme is validated on a dam-break problem with an elastic gate.

1 INTRODUCTION

Myriad of problems include violent and strongly nonlinear fluid-structure interaction (FSI), where such loading arises that causes structural deformations. Eulerian methods that solve partial differential equations (PDEs) on a mesh have been successfully employed for various problems, but generating a topologically clean and adequate mesh for the investigated problem is time-consuming work and often requires user intervening. An Eulerian solver that simulates large mesh deformations has difficulties to remain stable for problems with high geometrical complexity. Moving mesh nodes within this process can even require remeshing of areas with large deformation in order to avoid tangling of mesh elements, loss of the orthogonality, etc. On the contrary, overset-mesh schemes and Lagrangian mesh-free methods and hybrid methods have been developed for the purpose of avoiding these problems that are encountered in conventional mesh–based methods. Lagrangian and meshless methods are often employed to simulate violent flows with complex free surface evolution. Therefore, they are naturally capable of coupling with structural solvers to simulate nonlinear FSI with large deformations. An example of mesh-free discretisation compared to the mesh equivalent is shown in Figure 1.

Recently a novel meshless Lagrangian method was proposed for numerical simulation of incompressible flows and estimation of hydrodynamic loads wave-ship interaction [1, 2].

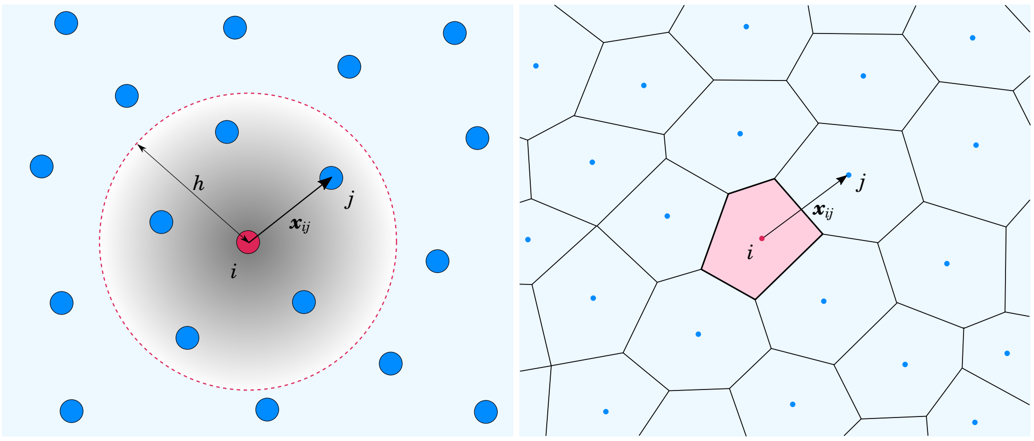

Figure 1: Mesh-free concept of space discretisation.

It was shown that the method accurately reproduces complex free surface patterns, as well as pressure distributions. Furthermore, the fluid is adjusting to moving boundaries represented by triangulated meshes while freely advecting about the boundaries by Lagrangian motion. The flow solver is volume–conservative, second–order accurate, and works directly on triangulated geometry. This property allows the method to directly use structural surface mesh as bounding geometry, and therefore to directly transfer loads between the flow solver and any structural solver [3].

Non-Newtonian fluids and granular flows may be simulated as continuum with large time steps by extending the above introduced method named Lagrangian Differencing Dynamics (LDD) [4]. In this paper non-Newtonian flow is simulated using the Power Law model, which is most commonly used model for purely viscous fluids. Viscous fluids are distinguished by the lack of the linear dependence of shear stress and shear strain rate that characterizes Newtonian fluids. Shear-thinning and shear-thickening fluids are the two main types of purely viscous fluids. These fluids are modelled in the Power Law model through the use of the flow-behaviour index n. The index can mathematically model three types of fluids. For $n < 1$, the effective viscosity decreases with increase of shear rate, i.e. it describes shear-thinning fluid. For $n > 1$, the model describes a shear-thickening fluid, and $n = 1$ describes a Newtonian fluid. Shear-thinning fluids are common in food, biological fluids, modern paints, nearly all polymer melts, polymer solutions, and other materials, whereas shear-thickening fluids can be found in body armours due to their impact hardening abilities.

Since the novel flow solver is ready to be coupled to a structural or a rigid-body solver due to the direct use of discrete geometry in fluid simulations, this study deals with establishing a bidirectional FSI scheme. The main aim is to investigate the coupling process between the non-Newtonian meshless Lagrangian method and Finite Element Method (FEM) solvers. The coupling process is validated using the LDD flow solver named *Rhoxyz*, but it should be extendable to any mesh-free Lagrangian solver. The solver is implemented in a way that points adjacent to walls are projected onto them, and boundary conditions for the pressure and velocity are imposed on those projections,

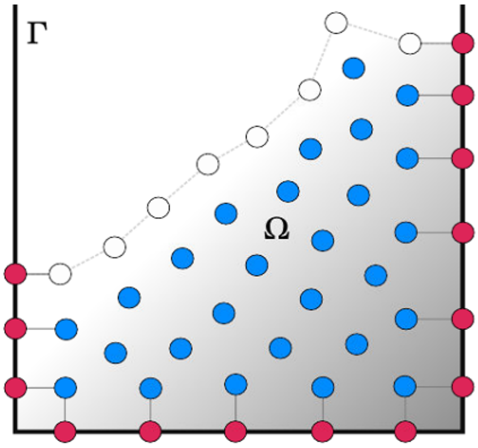

Figure 2: A schematic of the numerical domain with imposed no–slip (red) and free–surface (white) boundary conditions.

as shown in Figure 2. This makes transferring of deformations from the structural solver straightforward. *preCICE*, an open-source coupling library, is used to set-up partitioned bidirectional coupling of flow and the open-source FEM solver, named *CalculiX*. During the simulation the structure motion is imposed, the fluid stresses on the structure are applied in the structural equations-of-motion, and the deformations are brought back to the fluid solver. The coupling scheme is validated by simulating dam breaking with elastic gate clamped at one end.

2 METHODOLOGY

2.1 Governing equations

The Navier–Stokes equations in vector form for the incompressible fluid flow are described and solved. The conservation of momentum and mass is given as follows:

$$\frac{\mathrm{D}(\rho \boldsymbol{u})}{\mathrm{D}t} = -\nabla p + \nabla \cdot \boldsymbol{\tau} + \boldsymbol{F}_{ext}, \tag{1}$$

$$\nabla \cdot \boldsymbol{u} = 0, \tag{2}$$

where the advective derivative is expressed as $\mathrm{D}/\mathrm{D}t$, the velocity vector as $\boldsymbol{u}$, the fluid density as ρ, the fluid pressure as p, the stress tensor as $\boldsymbol{\tau}$, and $\boldsymbol{F}_{ext}$ as the vector of external forces. Eqs. (1) and (2) imply temporal and spatial dependency. The stress tensor $\boldsymbol{\tau}$ is identified for an incompressible fluid as:

$$\boldsymbol{\tau} = 2\,\mu\,(\boldsymbol{E})\;\boldsymbol{E}, \tag{3}$$

where μ is the variable dynamic viscosity of the fluid, which depends on$\boldsymbol{E}$, i.e. the strain rate that is defined as:

$$\boldsymbol{E} = \frac{1}{2}\left[\nabla \boldsymbol{u} + (\nabla \boldsymbol{u})^T\right], \tag{4}$$

where $\nabla \boldsymbol{u}$ indicates the velocity-gradient tensor of the flowing material. The shear rate is defined as:

$$\dot{\gamma} = \sqrt{2\boldsymbol{E} : \boldsymbol{E}^T}, \tag{5}$$

where the double-dot operator is defined as $\boldsymbol{E} : \boldsymbol{E}^T \equiv \mathrm{trace}(\boldsymbol{E}\boldsymbol{E}^T)$.

Power Law is a widely used and mathematically simple model that can approximately simulate the behaviour of a non-Newtonian fluid. In this generalized model for purely viscous fluids, the shear stress tensor is calculated as:

$$\boldsymbol{\tau} = k \left|\dot{\gamma}\right|^{n-1} \dot{\gamma}, \tag{6}$$

the model is defined by the effective viscosity as a function of the shear rate as follows:

$$\mu\left(\left|\dot{\gamma}\right|\right) = k \left|\dot{\gamma}\right|^{n-1}, \tag{7}$$

where k represents the flow-consistency index, and n is the flow-behaviour index. Depending on the flow-behaviour index n, it can mathematically model three types of fluids. $n < 1$ describes a shear-thinning fluid, $n > 1$ describes a shear-thickening fluid, and $n = 1$ describes a Newtonian fluid. The zero-shear viscosity is approached at very low shear rates, while the infinite shear viscosity is approached at very high shear rates.

2.2 Solving scheme

The method for solving incompressible flows is meshless, Lagrangian, volume–conservative and based on second–order accurate finite differences. Spatial operators, named Lagragian differences (LD), are based on generalised finite differences (FDs) derived by using weighted least–squares (WLS) [5], obtained within the compact sphere of all points shown in Figure (1). Flow is solved using the velocity–pressure decoupled scheme and the volume–conservative Lagrangian advection, obtained by solving a set of geometrical constraints [1]. Due to the fully Lagrangian description of the unsteady fluid flow, the method can handle and simulate violent FSI that includes complex free surface advection with fragmentation. The implementation of the method, named *Rhoxyz* (`http://rhoxyz.com`), has been validated for various problems in ship hydrodynamics [2, 1, 3]. The fluid solver works with geometry discretely described by triangles and quadrilaterals, and hence the discrete model of the structure may be directly used in the fluid simulation, which makes the transfer of loads straightforward. The method intrinsically handles violent fluid-structure interaction with free surface fragmentation, while providing second-order accurate pressure field [5, 1]. The method was recently extended to simulate non-Newtonian flows [4] by solving Eq. (1) in the implicit context using the Laplacian formulation of generalised Navier-Stokes equations.

2.3 Coupling scheme

The coupling scheme allows for coupling arbitrary structural solvers, which expose an Application Programming Interface (API) that enables sharing structure deformation during solving, and imposing forces to structure elements or nodes. In this study a

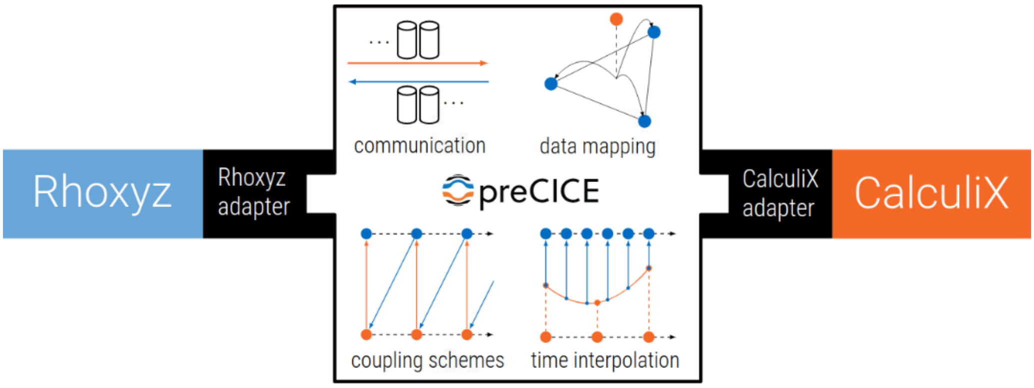

Figure 3: Peer-to-peer coupling capabilities with space and time interpolation, enabled for any participant solvers (in this case *Rhoxyz* and *CalculiX*).

validated open–source structural solver named *CalculiX* is employed, which is based on the Finite Element Method (FEM) [6]. Moreover, an open source coupling framework named *preCICE* (PREcise Code Interaction Coupling Environment) is used for bidirectional partitioned coupling of the structure and flow solvers [7]. The coupling scheme is schematically drawn in Figure 3, which renders how *preCICE* provides communication tools for for multi-physics simulation. The important ingredients for enabling massively parallel coupled simulations are: mapping of data between non-matching grids, peer-to-peer communication between solver processes, iterative methods for solving interface equations. In this study, the serial and explicit coupling scheme is used. At the start of each time step the solvers synchronise (wait for each other to reach the same point), seeing that the peer-to-peer communication channel must exchange data between the coupled solvers. The flow solver obtains and sends fluid force for each node of the patch mesh, while the structural solver sends deformations of the structure nodes. As implied above, non-matching interface discretisations of two solvers do not pose any issues; forces are conservatively interpolated from one solver to another performed by *preCICE*. Therefore, during this exchange the fluid solver obtains deformations of the structure nodes, i.e. deflections and velocity vectors of moved nodes, which are are used for imposing boundary conditions in the flow solver. Meanwhile, using the same communication channel the structural solver obtains fluid forces on each node of the structure mesh. In this study, the structure is made of an elastic isotropic material and discretised using eight-node brick elements (C3D8). The only thing needed for coupling from the structural solver-side is to define a set of nodes that define an interface for two-way transfer of information (e.g. nodes on the gate surface).

3 NUMERICAL EXPERIMENTS

3.1 Lid-driven cavity flow

The Power Law viscosity model is tested for a fluid that is circulating in a lid-driven skewed cavity flow. An experiment of the skewed cavity using the Power Law reported by Thohura *et al.* [8], is reproduced using the LDD method. The circulation pattern

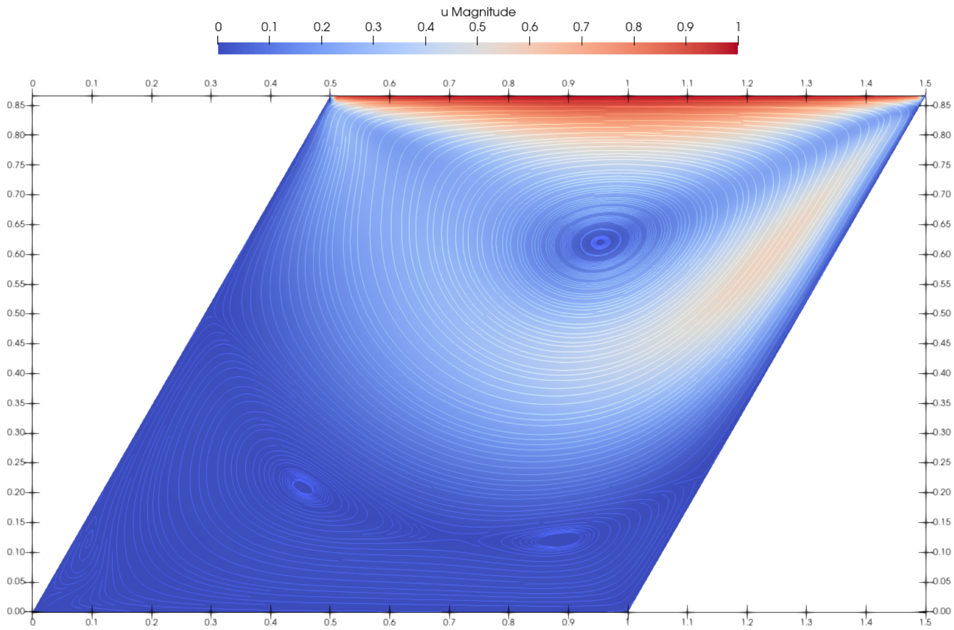

Figure 4: Velocity magnitude and streamlines for Power Law fluid flow in the skewed cavity, for $Re = 500$ and $n = 1.5$.

and vortex formation are highly dependent on the Reynolds number for any rheological behaviour. A relatively high Reynolds number was simulated, $Re = 500$, in order to show the stability and robustness of the LDD method. The Power Law flow-behaviour index was set to $n = 1.5$. The cavity in dimensionless units is 1×1 and the angle of the skewed-cavity is $\alpha = 60°$. The lid is moving with a steady, dimensionless velocity of $u_{lid} = 1$. The no-slip condition is applied to the lid and wall boundaries and an initial resolution of 200×200 points was used. The time step used for simulations is $\delta t = 10^{-3}$, and the calculation of a time-step took 26 ms in average on a GTX 980Ti GPU. $t = 20$ seconds of physical time was simulated until steady state of the simulation was reached. In the Figure 4, the plotted streamlines correspond very well to the reference data given in [8]. The position of the vortices is correctly captured, and vortex in the lower-right corner is more clearly represented in the LDD method than in the FVM. A good match with the results is obtained, demonstrating that the method is capable of simulating non-Newtonian Power Law fluids.

3.2 Dam break with elastic gate

In this section an experiment conducted by Antoci *et al.* [9] is reproduced. The experiment resembles to typical dam-breaking problem, but the gate is not rigid nor movable, but instead it is elastic and deformable. The rubber gate is clamped along its upper side to the rigid wall, and it deforms when subjected to fluid forces behind it. The tank space is filled with fluid column of length $A = 100$ mm and height $H = 140$ mm, while the rubber gate is supported by an rigid obstacle and its lower end touches the floor. The gate has thickness of $s = 5$ mm and height of $L = 79$ mm. The rubber gate is

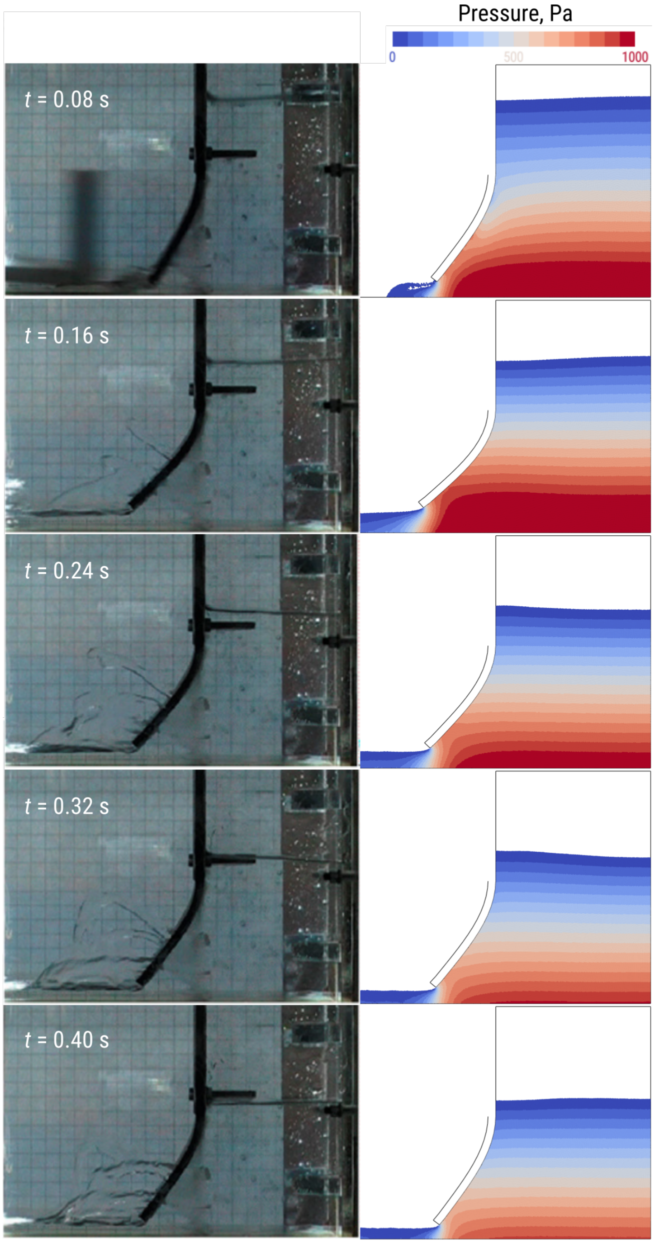

Figure 5: Dam break with a rubber gate; the simulation results are compared to the photographs taken during the experiment.

modeled using a elastic isotropic material with density $\rho_{gate} = 1100$ kg/m^3, and Young's modulus $E = 12$ MPa. Since some uncertainty occurs durting estimation of the Young modulus for rubber, future work will include proper rubber hyper-elastic properties. For the validation, the tank was filled with water, $\rho = 1000$ kg/m^3 and $\mu = 10^{-3}$ Pa·s, made of 56000 fluid points with initial point spacing 0.5 mm. The constant time-step was $\delta t = 2 \cdot 10^{-4}$ s, and the calculation of a time-step took 40 ms in average on a RTX 2080Ti GPU. The time-step reported in [9] for 6000 fluid SPH particles was $\delta t = 8 \cdot 10^{-6}$, which

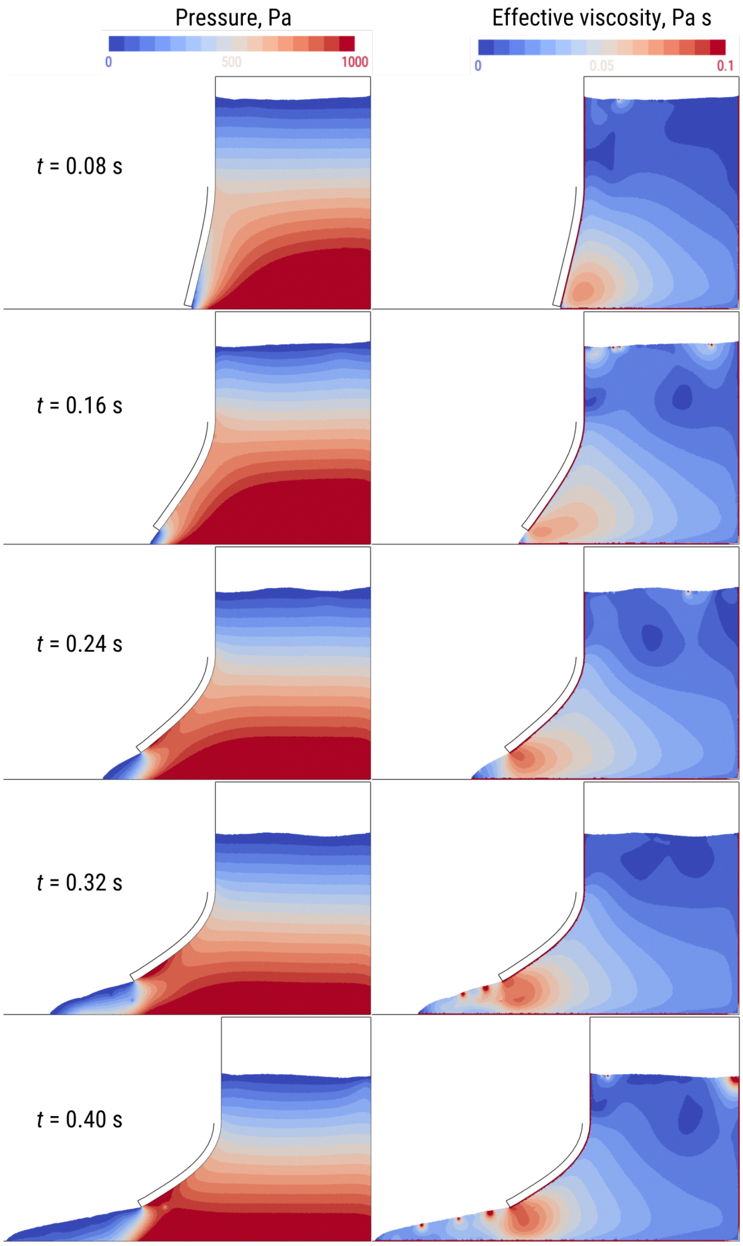

Figure 6: Dam break with a rubber gate, using a Power Law fluid with $n = 2$ and $\mu_0 = 10$ Pa·s.

emphasises the robustness of the implicit solving [1, 4]. Moreover, larger stable time-step values are expected for the implicit partitioned type of coupling that is available in *preCICE* [7], which will be assessed in future work.

The obstacle that supports the rubber gate is suddenly removed, which allows the hydrostatic condition to initially deform the lower end of the elastic plate, and this allows the water to flow under it. Analysis of the experiments showed that the resulting flow and the plate deformation can be studied as a two-dimensional phenomenon. Therefore two-dimensional flow is simulated, while the fluid forces are imposed on the gate modeled by one column of 28 brick (C3D8) elements. The solution captured during the simulation is shown in Figure 5, and compared to the photographs taken during the experiment. The evolution of the gate deformation and water level change is similar between the compared images. Furthermore, the free-surface shape (local elevation) evolution due to pressure gradients from the concentrated outflow is also properly simulated. The results shown that accurate prediction of the displacement of the elastic structure subjected to fluid pressure and of the resulting fluid flow can be obtained using the LDD method coupled with a FEM solver. More investigation is needed to include real rubber-like behaviour of the gate, and to analyse the disadvantages of explicit type of coupling.

In the second numerical experiment, the tank was filled with a Power Law fluid. The shear thickening effect of the flow was employed by setting the flow-behaviour index $n = 2$ and the flow-consistency index $k = 10$ Pa·s^2. The same initial spacing and time step was used, as defined in the text above. The solution captured during the simulation is shown in Figure 6, which renders the pressure field and effective-viscosity field as contour plots. Local maxima of the effective-viscosity scalar field are adequately reproduced at locations with high pressure gradient that generated significant velocity gradient. Some local deficiencies may be seen at the free surface, which will be assessed in future work.

4 CONCLUSIONS

The recently introduced Lagrangian mesh-free method for the simulation of incompressible fluids with a free surface, named Lagrangian Differencing Dynamics (LDD), is extended to simulate non-Newtonian fluids. It was validated that the method can accurately simulate lid-driven cavity non-Newtonian flow. The implementation of the LDD method available in the public domain, named *Rhoxyz*, was coupled with *Calculix* solver for the structure deformation using *preCICE*. The process of implementing the coupling scheme verified that this tool enables effortless coupling of arbitrary solvers without having to change solver algorithms and input files. The coupled scheme introduced in this paper was successfully validated using a dam-break problem with flexible gate, using water as fluid. By modifying the fluid properties, it was shown that the coupling scheme can handle non-Newtonian fluids using high time steps. The time-step values could be even higher if the implicit coupling scheme is used, which will be investigated in future work. Moreover, more complex simulations with three-dimensional structures will be investigated in future work.

REFERENCES

[1] Josip Bašić, Nastia Degiuli, Šime Malenica, and Dario Ban. Lagrangian finite-difference method for predicting green water loadings. *Ocean Engineering*, 2020.

[2] Josip Bašić, Branko Blagojević, Dario Ban, Boris Ljubenkov, and Nastia Degiuli. Lagrangian Finite Difference Method for Violent Fluid–Structure Interaction. In M. Abdel-Maksoud, editor, *Proceedings of the 32nd Symposium on Naval Hydrodynamics*, page 14, Hamburg, 2018.

[3] Martina Andrun, Josip Bašić, Branko Blagojević, and Branko Klarin. Simulating Hydroelastic Slamming by Coupled Lagrangian–FDM and FEM. In *Progress in Marine Science and Technology, Volume 5: HSMV 2020*, pages 135–142. Naples, Italy, oct 2020.

[4] Chong Peng, Matina Bašić, Branko Blagojević, Josip Bašić, and Wei Wu. A Lagrangian differencing dynamics method for granular flow modeling. *Computers and Geotechnics*, 137:104297, 2021.

[5] Josip Basic, Nastia Degiuli, and Dario Ban. A class of renormalised meshless Laplacians for boundary value problems. *Journal of Computational Physics*, 354:269–287, feb 2018.

[6] Guido Dhondt. *The Finite Element Method for Three-Dimensional Thermomechanical Applications*. John Wiley & Sons, Ltd, Chichester, UK, may 2004.

[7] Hans-Joachim Bungartz, Florian Lindner, Bernhard Gatzhammer, Miriam Mehl, Klaudius Scheufele, Alexander Shukaev, and Benjamin Uekermann. preCICE â A fully parallel library for multi-physics surface coupling. *Computers & Fluids*, 141:250–258, dec 2016.

[8] Sharaban Thohura, Md. Mamun Molla, and Md. Manirul Alam Sarker. Numerical Simulation of Non-Newtonian Power-Law Fluid Flow in a Lid-Driven Skewed Cavity. *International Journal of Applied and Computational Mathematics*, 5(1):14, feb 2019.

[9] Carla Antoci, Mario Gallati, and Stefano Sibilla. Numerical simulation of fluidâstructure interaction by SPH. *Computers & Structures*, 85(11-14):879–890, jun 2007.