Proceedings of the International Symposium on Engineering Lessons Learned from
the 2011 Great East Japan Earthquake, March 1-4, 2012, Tokyo, Japan

TSUNAMI IMPACT ON COASTAL STRUCTURES: PARTITIONED SOLUTION PROCEDURE FOR FLUID–STRUCTURE INTERACTION WITH FREE–SURFACE FLOW

A. Ibrahimbegovic[1], C. Kassiotis[2] and H. G. Matthies[3]

January 16, 2012

[1] LMT-Cachan, ENS Cachan/UPMC/CNRS/PRES UniverSud Paris, France
ai@lmt.ens-cachan.fr
[2]Saint-Venant Laboratory for Hydraulics, Université Paris-Est (Joint Research Unit EDF R&D, CETMEF, École des Ponts ParisTech)
6 quai Watier, BP 49, 78401 Chatou, France
christophe.kassiotis@enpc.fr
[3] Institut for Scientific Computing, TU-Braunschweig, Germany
wire@tu-bs.de

Abstract

In this work we discuss a way to compute the interaction between free-surface flows and non-linear structures. The approach chosen rely on a partitioned strategy that allows to solve strongly coupled fluid-structure interaction problem. The software coupling is ensured in an efficient way using the Communication Template Library (CTL). Numerical examples presented herein concern 2D validations case but also 3D problems with a large number of equations to be solved.

INTRODUCTION

In the 2011 Tohoku Disaster, the huge tsunami, caused by an ocean-ridge earthquake of magnitude 9.0, was the primary source of the destruction and death toll. Moreover, this disaster is characterized by the vast affected region and by the chain of events where one disaster (earthquake) triggered another (tsunami). Such chains of events pose many challenges to overcome, and many lessons to learn from the 2011 Tohoku Disaster. In particular, the unprecedented risks brought about by combining and superimposing the disaster threats requires a new approach to understanding the way of mitigating the consequences, which is best spelled by the word resiliency that is currently being used in earthquake engineering community.

In most of the current applications, the fluid loading is applied statically on the construction, taking into account its dynamics and the interaction effects with only security coefficients. Obviously, the problem is coupled (the more a coastal protection resists, the higher the fluid level, and therefore the loading is), and there is a need to compute both the resistance of structures and the flow main characteristics, especially when the failure occurs.

In this work we focus on fluid-structure interaction problems, with free-surface flows. The free-surface flow problem is described by the Navier-Stokes equations with two phases (water and air), set in an ALE framework [Farhat *et al.* , 2001, Demirdžić & Perić, 1988, Hirt *et al.* , 1997] and discretized with a VOF strategy [Dutykh & Mitsotakis, 2009, Ubbink & Issa, 1999, Sobey, 1998, Stoker, 1992] For the structure part, it is natural to follow material point motion in a Lagrangian formulation and a discretization with FEM [Ibrahimbegović, 2009].

For the coupled problem, the monolithic approach is abandoned in favor of the partitioned approach [Farhat & Lesoinne, 2000], [Le Tallec & Mouro, 2001], [Matthies & Steindorf, 2003], [Causin *et al.* , 2005], [Fernández & Moubachir, 2005], [Matthies *et al.* , 2006], [Deparis *et al.* , 2006], [Förster *et al.* , 2007] and

[Fernández *et al.* , 2008]. The latter is preferred for its modularity and the possibility of re-using existing software. The partitioned approach used here is based on a simple fixed-point strategy based on the Block Gauss-Seidel algorithm (DFMT-BGS) with an adaptive relaxation parameter [Küttler & Wall, 2008] that shows sufficient performances for the example proposed. The properties as well as stability of the implicit coupling DFMT-BGS algorithms used herein are presented in detail in [Kassiotis *et al.* , 2010a].

In this work, a general fluid-structure interaction framework based on existing software was used. This framework was built using the middleware Communication Template Library (CTL) [Niekamp *et al.* , 2009] which offers good performances, and can therefore be used for scientific computing of large systems. An important feature is the possibility to couple types of software product that were initially not programmed to be coupled (here `FEAP` for the structure and `OpenFOAM` for the fluid), even if they are based on different discretization techniques (respectively FV for the fluid and FE for the structure) and were programmed in different languages, `C++` and `Fortran`. For more details on the implementation, see [Kassiotis *et al.* , 2010b]

The outline of this paper is as follow: in the subsequent section we present the chosen formulations for the structure and fluid sub-problem. In Section 3, we describe the coupling between the fluid and the structure sub-problems. In Section 4, we give and comment the results of illustrative numerical examples dealing with free-surface flow impacting a structure in two and three-dimensions as well as comparison with existing works. The concluding remarks are given in the last section.

DESCRIPTION OF THE FREE SURFACE FLOWS AND THE STRUCTURES

We will not here gives the detailled PDE for the structure and the fluid, but give directly the semi-discrete form of the problem. For the structure, it can be set in a matrix notation by using the real valued vectors $\mathbf{u}_s \in \mathbb{R}^{n_{d-o-f}}$ that give a discrete approximation of the 3D displacement vector field $\boldsymbol{u}_s$:

$$\mathcal{R}_s(\mathbf{u}_s;\boldsymbol{\lambda}) := \mathbf{M}_s\ddot{\mathbf{u}}_s + \mathbf{f}_s^{\text{int}}(\mathbf{u}_s) - \mathbf{f}_s^{\text{ext}}(\boldsymbol{\lambda}) = \mathbf{0} \tag{1}$$

where $\mathbf{M}$ is the mass matrix, $\mathbf{f}_s^{\text{int}}$ with a geometrically nonlinear problem, and $\mathbf{f}_s^{\text{ext}}$ the consistent nodal forces. Here the $\boldsymbol{\lambda}$ represents the boundary forces computed from the fluid flow problem and imposed on the fluid-structure interface. In order to complete the discretization process, the time integration of the structure problem can be carried out by using standard time-stepping schemes [Ibrahimbegović, 2009]. In particular, the Generalized HHT-α method is used herein.

The semi-discrete form of the discretized fluid problem can be written in a matrix form as follows. The discrete fluid mesh motion considers that $\mathbf{u}_m$ is imposed by the motion of the interface $\mathbf{u}$:

$$\mathcal{R}_m(\mathbf{u}_m;\mathbf{u}) := \mathbf{K}_m\mathbf{u}_m - \mathbf{D}_m\mathbf{u} = \mathbf{0} \tag{2}$$

where $\mathbf{D}_m$ is a projection/restriction operator and $\mathbf{K}_m$ governs the extension of the boundary displacement. The (discrete) volume fraction $\boldsymbol{\iota}$, the 3 components of velocity $\mathbf{v}$ and pressure p are coupled through a set of non-linear equations. Written in a matrix forms, it gives the following semi-discrete problem:

$$\begin{aligned}
&\mathcal{R}_f(\boldsymbol{\iota},\mathbf{v}_f,\mathsf{p}_f;\mathbf{u}_m)\\
&:= \begin{bmatrix}
\mathbf{M}_\iota\dot{\boldsymbol{\iota}} + \mathbf{N}_\iota(\mathbf{v}_f - \dot{\mathbf{u}}_m)\boldsymbol{\iota}\\
\mathbf{M}_f(\boldsymbol{\iota})\dot{\mathbf{v}}_f + \mathbf{N}_f(\boldsymbol{\iota},\mathbf{v}_f - \dot{\mathbf{u}}_m)\mathbf{v} + \dots\\
\qquad \dots \mathbf{K}_f(\boldsymbol{\iota})\mathbf{v}_f + \mathbf{B}_f\mathsf{p}_f - \mathbf{f}_f(\boldsymbol{\iota})\\
\mathbf{B}_f^T\mathbf{v}_f
\end{bmatrix}\\
&= \mathbf{0}
\end{aligned} \tag{3}$$

where $\mathbf{M}_\iota$ and $\mathbf{N}_\iota$ are the matrices associated to the advection problem of the fluid volume fraction, $\mathbf{M}_f$ is a positive definite mass matrix, $\mathbf{N}_f$ is an unsymmetric advection matrix, $\mathbf{K}_f$ is the conduction matrix describing the diffusion terms, and $\mathbf{B}_f$ is for the gradient matrix, whereas $\mathbf{f}_f$ is the discretized nodal loads on the flow. This matrix form also takes into account the boundary conditions; special care has to be taken concerning the discretization of boundary conditions – and especially normal flux – when using the Finite Volume Method [Ghidaglia & Pascal, 2005].

One way to solve the flow problem is to consider a monolithic solver handling all equations simultaneously. Another way is to consider a split between the mesh motion, the volume fraction advection, the momentum and the continuity equations, and to use an operator split-like procedure often referred

to the *segregated approach* [Patankar, 1980]. This approach is favored for its computational efficiency compared to the monolithic scheme. Indeed, even with a simple fixed point iteration strategy its cost is less important than that of the monolithic approach for large size problems [Ferziger & Perić, 2002]. In the work presented herein, the segregated approach will be used because of its efficiency.

COUPLING STRATEGY AND IMPLEMENTATION

By enforcing the continuity of primal variables at the interface we can eliminate the energy errors that characterize the explicit interface matching. When coupling incompressible flow with structure, the implicit interface matching is required for stability reason, as proved in [Kassiotis *et al.* , 2010a]. This ought to be done by iterating on the following residual to reduce its value below the chosen tolerance:

$$\boldsymbol{r}_{N+1} := \boldsymbol{u}_{s,N+1} - \boldsymbol{u}_{f,N+1} \simeq 0 \quad \leq \texttt{TOL} \tag{4}$$

In this way we obtain an implicit algorithm requiring more than one iteration to enforce the interface matching condition. The chosen order of iterations, corresponds to the Block-Gauß-Seidel algorithm for fluid-structure interaction problem [Matthies *et al.* , 2006]. Let us note that not only the value at synchronization points T_n or T_{n+1}, but also the interpolated evolution of variables have to be exchanged in the entire time-interval $t \in [T_n, T_{n+1}]$ when the time steps are not matching between fluid and structure sub-problems.

Contrary to explicit algorithms which generate spurious energy at the interface, the present implicit interface matching algorithm enforce the same evolution of the primal variables at the fluid-structure interface. In other words, an iterative solution for primal (displacements) continuity as well as the dual (forces) equilibrium equations at the interface is performed by using the Picard iteration:

$$\boldsymbol{u}_{N+1}^{(k+1)} = \mathcal{G}\left(\boldsymbol{u}_{N+1}^{(k)}\right); \qquad \mathcal{G} = {\mathcal{S}_s}^{-1} \circ -\mathcal{S}_f \tag{5}$$

where $\mathcal{S}_f$ and $\mathcal{S}_s$ are the Steklov-Poincaré operators for fluid and structure defined as traditionally defined in [Deparis *et al.* , 2006]. These operators can be formulated using transfer operators and equations (1), (3) and (2):

$$\mathcal{S}_s = \mathcal{T}_s^{\boldsymbol{\lambda}} \circ \mathcal{R}_s \circ \mathcal{T}_s^{\boldsymbol{u}}; \qquad \mathcal{S}_f = \mathcal{T}_f^{\boldsymbol{\lambda}} \circ \mathcal{R}_f \circ \mathcal{R}_m \circ \mathcal{T}_s^{\boldsymbol{u}} \tag{6}$$

where the transfer of structure displacement to fluid-structure interface displacement is $\mathcal{T}_s^{\boldsymbol{u}}$, the transfer of fluid displacement to interface displacement $\mathcal{T}_s^{\boldsymbol{\lambda}}$, the transfer of structure stresses to the interface $\mathcal{T}_f^{\boldsymbol{u}}$ and the transfer of fluid stresses to the interface $\mathcal{T}_f^{\boldsymbol{\lambda}}$.

The Picard iterations will continue until convergence of interface residual is achieved:

$$\boldsymbol{r}_{N+1}^{(k)} = \boldsymbol{u}_{s,N+1}^{(k)} - \boldsymbol{u}_{f,N+1}^{(k)} = \mathcal{G}\left(\boldsymbol{u}_{N+1}^{(k)}\right) - \boldsymbol{u}_{N+1}^{(k)} \tag{7}$$

It is clear that this fixed-point algorithm based on Picard iterations has the main drawback that the search directions for $\boldsymbol{u}$ and $\boldsymbol{\lambda}$ variables at the interface do not exploit any information from the fixed-point function $\mathcal{G}$ nor the Steklov-Poincaré operators $\mathcal{S}_f$ and $\mathcal{S}_s$. Therefore, quite a few iterations may be needed to reach the convergence.

The stability of such a coupling algorithm is studied in [Kassiotis *et al.* , 2010a]. We give a formal proof of potential numerical instability due to the *added-mass effect* also observed in [Förster *et al.* , 2007, Le Tallec & Mouro, 2001]. In order to improve the convergence of the DFMT-BGS method, we can use a relaxed update:

$$\boldsymbol{u}_{N+1}^{(k+1)} = \boldsymbol{u}_{N+1}^{(k)} + \omega^{(k)}\ \boldsymbol{r}_{N+1}^{(k)} \tag{8}$$

Our favorite choice for constructing $\omega^{(k)}$ is using a secant methods which can keep the cost of each iteration as low as possible. The Aitken's relaxation strategy has been extensively used in fluid-structure interaction [Deparis *et al.* , 2006, Küttler & Wall, 2008], and shown sufficient performances to be used in the following.

The use of different different solvers, for the fluid and the structure part, do not provide in general a matching mesh at the interface. Furthermore, even for matching meshes, as the geometries of the domains are not the same on both sides of the interface, an optimal numbering of the nodes can lead to different orders for the interface nodes. In the examples proposed herein, only this latter point is of interest. Last

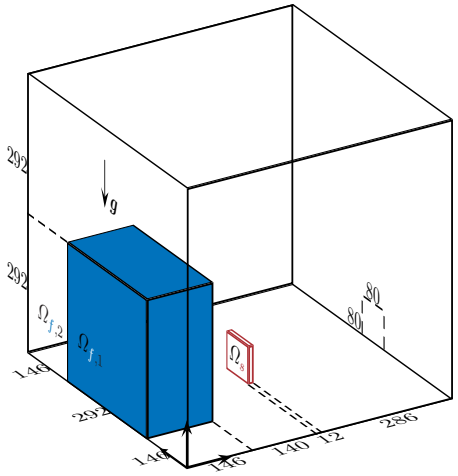


Figure 1: Three-dimensional water column impacting an obstacle: geometry (given in mm) and boundary conditions

but not least, different discretization techniques (Finite Element versus Finite Volume) or different order p of the polynomials can be used for constructing solution to fluid-structure interaction problem. In the domain of FE applied to mechanical engineering, extensive literature can be found on how to build a consistent interpolation for both subproblems at the interface [Felippa & Park, 2004]. For the fluid-structure interaction problems, an interesting review can be found in [de Boer *et al.* , 2007]. In our framework, it was decided not to favor any mesh-based representation of the interface, since, in the most general case, the fluid problem can also be solved by a meshfree-based method [Dalrymple & Rogers, 2006]. Namely, an interpolation strategy relying on radial basis function is here chosen. This method has already been employed for FSI in [Monaghan, 1992, Beckert & Wendland, 2001].

The algorithm presented here is simple to implement. We use for this work the Communication Template Library (CTL, see [Markovič *et al.* , 2005, Niekamp *et al.* , 2009]) that allows to re-use existing codes in a generic way, either called as libraries on the same computer, or as remote executables through network. With the CTL, we are able to couple existing stand-alone software, in a quite straightforward way, even if they are programmed in different langages (`Fortran` for the structure part, `C++` for the fluid part), and to conserve the inner parallelism of each component. For more details on the implementation, the reader is invited to see [Kassiotis *et al.* , 2010b].

NUMERICAL SIMULATION

The problem solved is a 3D generalization of dam-breaking event that brings about a sloshing wave impact on a rigid [Bullock *et al.* , 2007, Ubbink & Issa, 1999] or flexible structure as presented in 2D in [Walhorn *et al.* , 2005, Baudille & Biancolini, 2005]. At initial time $t = 0s$, a the three-dimensional water column starts falling down under the gravity loading and eventually hits the obstacle placed in the way. The flexible obstacle is a slender plate-like body made of elastic material that can undergo large deformation. The chosen dimension of the problem, as well as the boundary conditions are given in Fig. 1. Let us note that we propose to use open boundary conditions far from the obstacle in order to avoid the water bounces-back and hits again the structure after breaking off the walls. For that reason, only the left and bottom planes of the fluid domain are defined as non-slipping walls, while the others are defined with boundary condition of atmospheric pressure.

The material properties are chosen as follows: the density and the kinematic viscosity are $\rho_{f,1} = 1 \times 10^3 kg.m^{-3}$ and $\nu_{f,1} = 1 \times 10^5 m.s^{-1}$ for the high density fluid (water in the reservoir), versus $\rho_{f,2} = 1 \times 10^3 kg.m^{-3}$ and $\nu_{f,2} = 1 \times 10^6 m.s^{-1}$ for the low density fluid (air in the remaining part of the domain). The mesh motion problem is solved by using a Laplacian smoothing material where the diffusion coefficient is a quadratic inverse function of the distance to the interface between solid and fluid.

The results are computed for two meshes with the chosen discretization and the number of cells given in Tab. 1. For the finest grid, around 64, 60 and 40 cells are used in $\boldsymbol{e}_x$, $\boldsymbol{e}_y$ and $\boldsymbol{e}_z$ direction. The mesh

is refined gradually refined around the structure, and initially the cell dimensions are between 8.9×10^{-8} and 3.2×10^{-5}. For this finest grid, the maximum skewness of the mesh observed is 2.947, that does not generate too large errors. The fluid is handled by second order space discretization and a Van Leer limiter is used for the advection terms. The time integration scheme employed in this computation is implicit Euler. For such a scale of modeling it is not required to consider surface tension between the two fluids. For this problem the fluid computation is parallelized, but reduction of the CPU time is obtained by using a Generalized Algebraic-MultiGrid (GAMG) linear solver. Note that small time steps are required for the explicit solution of the phase function indicator equation, as well as the half-implicit nature of the coupling between the momentum predictor and the pressure corrector.

For the structure part, we propose here to use three-dimensional elements with quadratic shape functions, where each element has 27 nodes. The material properties used for the solid are: a neo-Hookean elastic material with Young's modulus $E_s = 1 \times 10^6 Pa$ and Poisson's ratio $\nu_s = 0$ and a density $\rho_s = 2500 kg \cdot m^{-3}$, which can represent finite deformation. The time integration is carried out by a Generalized-α scheme with the same parameters as the one used for the 2D case.

	fluid d-o-f	solid d-o-f	number of time steps
Coarse	63×10^3	1.1×10^3	1×10^5
Fine	520×10^3	6.6×10^3	1×10^5

Table 1: Number of d-o-f for coarse and fine discretization of the three-dimensional dam-breaking problem

The computation of the coupled problem (with total number of d-o-f given in Tab. 1), is carried out by an implicit iterative scheme. The results of fluid and solid computations are matched for a time step of 1×10^{-4} for the coarse and 2×10^{-5} for the fine discretization. The coupling scheme used is DFMT-BGS with Aitken's relaxation. The initial parameter is $\omega = 0.25$ and the predictor is of order 1. The absolute tolerance considered is:

$$\|\boldsymbol{r}_N^{(k)}\| \leq 1 \times 10^{-6} \tag{9}$$

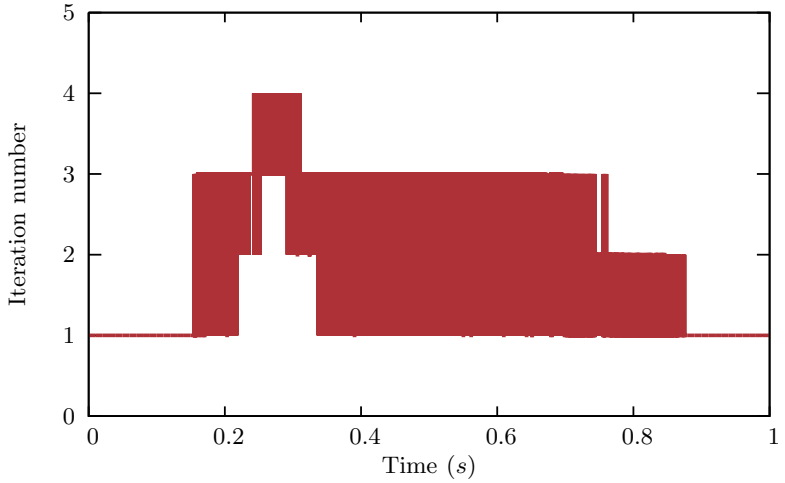
The number of iterations required to reach the convergence criteria is given in Fig. 2(a). Note that there is no iteration required before the water hits the structure since the effect of air flow can almost be deemed negligible with respect to the structure. The number of iterations required for the coarse and fine mesh is small. This propertie is observed for most of the simulation carried out with free-surface flow when the convergence of the pressure solver is easily reached. For the finest grid, the total time required to perform the whole coupled simulation on a single $3.0 GHz$ Intel processor is $279 \times 10^3 s$.

In Fig. 3, the high density fluid domain is represented, as well as some part of the fluid mesh and the structure displacement. The first $0.1s$ of the simulations, the water column falls under the gravity loading. There is no effect whatsoever on the structure until the high density flow reaches its bottom. The maximum amplitude of the motion is obtained at $t = 0.25s$, before the solid comes back to its initial position and oscillates after the shock.

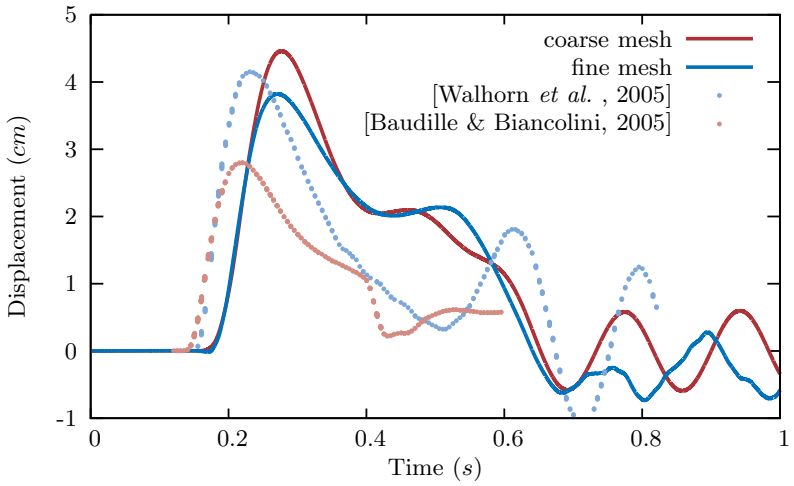
In Fig. 2(b) the motion of the extremity of the solid obstacle is plotted. Contrary of the two-dimensional example, small drops of high density fluid are not interacting with the obstacle after the main shock. Therefore, the motion of the flexible structure remains fairly smoothed and it is rather well described with the coarsest grid. We present for comparison the results obtain with 2D models using either a monolithic stabilized-FEM strategy [Walhorn *et al.* , 2005] or tight coupling strategy between a FV and FEM solver [Baudille & Biancolini, 2005]. The use of 2D models ad well as the difference in strategy explain the differences observed (our results are closer for a 2D simulation, as presented in [Kassiotis *et al.* , 2010c]).

CONCLUSION

The proposed solution method allows to perform coupled simulations and obtain reliable solution to complex fluid-structure interaction by using the existing codes, that were initially developed to support either fluid or structure motion computation. This is achieved thanks to the use of the component technology [Kassiotis *et al.* , 2010b, Niekamp *et al.* , 2009] providing the coupling between existing software products. Therefore, the proposed solution method for fluid-structure interaction can utilize very different discretization strategies to obtain the optimal accuracy; The case in point concerns FE for the



(a) Number of iterations in order to make the DFMT-BGS algorithm converge for the three-dimensional dam-breaking problem



(b) Three-dimensional dam break example: obstacle displacement in $\boldsymbol{e}_x$ direction of center point of the top face $(40; 6; 80)$ and comparison with 2D models [Baudille & Biancolini, 2005, Walhorn *et al.* , 2005]

Figure 2: Results of three-dimensional simulations

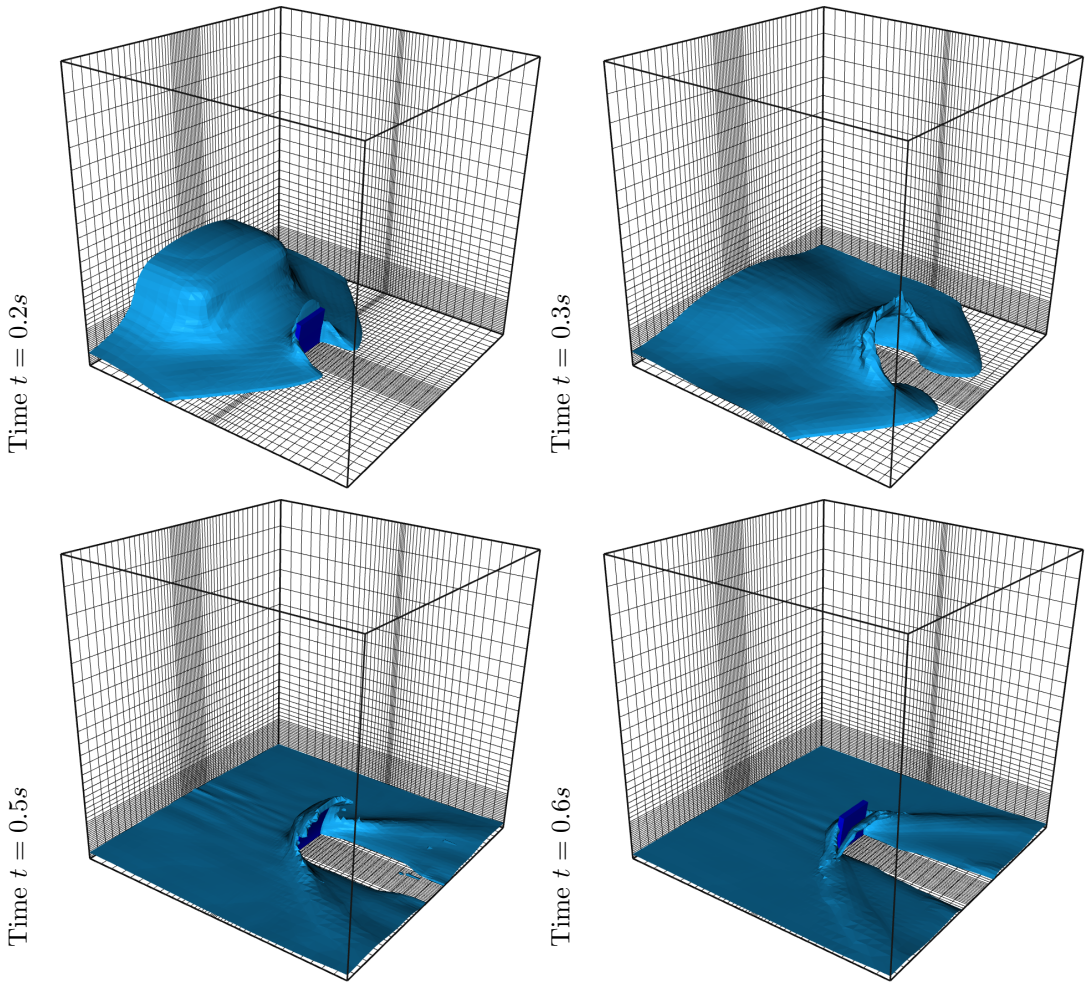


Figure 3: Tri-dimensional dam break problem. Evolution of the free surface and motion of the structure.

structure and FV for the fluid. The use of these popular methods for the fluid and solid parts allows to benefit from the advanced features of the two families of methods, each developed by the experts from the corresponding domain. Accordingly, on the fluid side, it is possible to use a very efficient semi-implicit solver for incompressible flow (PISO), inverse techniques (Algebraic Multigrid) or advanced models for free-surface flows. A very good performance of the proposed technology for fluid-structure interaction is illustrated with 2D and 3D models for dam breaking examples, which also involve flexible obstacles.

Even if the convergence is quite easy to achieve, the use of the recently proposed semi-implicit coupling strategy [Astorino *et al.* , 2009] based on the predictor-corrector work traditionnaly used for pressure correction could by a good way to decrease the computational cost without loosing accuracy. An other point of interest is the extension of this work to more complex behavior, for both the fluid (turbulence, non Newtonian flows, debris flows...) and the structure (non-linear behavior, cracks and failure).

REFERENCES

[Astorino *et al.* , 2009] Astorino, M., Chouly, F., & Fernández, M. Á. 2009. An added-mass free semi-implicit coupling scheme for fluid-structure interaction. *Comptes Rendus Mathematique*, **347**(1-2), 99–104.

[Baudille & Biancolini, 2005] Baudille, R., & Biancolini, M. E. 2005. Modelling FSI problems in FLUENT: a dedicated approach by means of UDF programming. *In: Proceedings of the European Automotive CFD Conference, Frankfurt, Germany.*

[Beckert & Wendland, 2001] Beckert, A., & Wendland, H. 2001. Multivariate interpolation for fluid-structure-interaction problems using radial basis functions. *Aerospace Science and Technology*, **5**(2), 125–134.

[Bullock *et al.* , 2007] Bullock, GN, Obhrai, C., Peregrine, DH, & Bredmose, H. 2007. Violent breaking wave impacts. Part I: Results from large-scale regular wave tests on vertical and sloping walls. *Coastal Engineering*, **54**(8), 602–617.

[Causin *et al.* , 2005] Causin, P., Gerbeau, J.-F., & Nobile, Fabio. 2005. Added-mass effect in the design of partitioned algorithms for fluid-structure problems. *Computer Methods in Applied Mechanics and Engineering*, **194**(42-44), 4506–4527.

[Dalrymple & Rogers, 2006] Dalrymple, RA, & Rogers, BD. 2006. Numerical modeling of water waves with the SPH method. *Coastal engineering*, **53**(2-3), 141–147.

[de Boer *et al.* , 2007] de Boer, A., van Zuijlen, A. H., & Bijl, H. 2007. Review of coupling methods for non-matching meshes. *Computer Methods in Applied Mechanics and Engineering*, **196**(8), 1515–1525.

[Demirdžić & Perić, 1988] Demirdžić, I., & Perić, Milovan. 1988. Space conservation law in finite volume calculations of fluid flow. *International Journal for Numerical Methods in Fluid*, **8**(9).

[Deparis *et al.* , 2006] Deparis, Simone, Discacciati, Marco, Fourestey, Gilles, & Quarteroni, Alfio. 2006. Fluid-structure algorithms based on Steklov-Poincaré operators. *Computer Methods in Applied Mechanics and Engineering*, **195**(41-43), 5797–5812.

[Dutykh & Mitsotakis, 2009] Dutykh, Denis, & Mitsotakis, Dimitrios. 2009. On the relevance of the dam break problem in the context of nonlinear shallow water equations. *Discrete and Continous Dynamical System – Series A*, **Accepted**.

[Farhat & Lesoinne, 2000] Farhat, Charbel, & Lesoinne, M. 2000. Two efficient staggered algorithms for the serial and parallel solution of three-dimensional nonlinear transient aeroelastic problems. *Computer Methods in Applied Mechanics and Engineering*, **182**, 499–515.

[Farhat *et al.* , 2001] Farhat, Charbel, Geuzaine, Philippe, & Grandmont, Céline. 2001. The Discrete Geometric Conservation Law and the Nonlinear Stability of ALE Schemes for the Solution of Flow Problems on Moving Grids. *Journal of Computational Physics*, **174**(2), 669–694.

[Felippa & Park, 2004] Felippa, C.A., & Park, K.C. 2004. Synthesis tools for structural dynamics and partitioned analysis of coupled systems. *NATO Advanced Research Workshop (eds. A. Ibrahimbegović and B. Brank)*, 50–111.

[Fernández & Moubachir, 2005] Fernández, M. Á., & Moubachir, M. 2005. A Newton method using exact Jacobians for solving fluid–structure coupling. *Computers and Structures*, **83**(2-3), 127–142.

[Fernández *et al.* , 2008] Fernández, M. Á., Gerbeau, J.-F., Gloria, A., & Vidrascu, M. 2008. Domain decomposition based Newton methods for fluid-structure interaction problems. *Pages 67–82 of: ESAIM: Proceedings*, vol. 22. edpsciences.org.

[Ferziger & Perić, 2002] Ferziger, Joel H., & Perić, Milovan. 2002. *Computational Methods for Fluid Dynamics*. 3^{rd} edn. Berlin, Germany: Springler-Verlag.

[Förster *et al.* , 2007] Förster, Christiane, Ramm, Wolfgang A., & Ramm, Ekkehard. 2007. Artificial added mass instabilities in sequential staggered coupling of nonlinear structures and incompressible viscous flows. *Computer Methods in Applied Mechanics and Engineering*, **196**, 1278–1291.

[Ghidaglia & Pascal, 2005] Ghidaglia, Jean-Michel, & Pascal, F. 2005. The normal flux method at the boundary for multidimensional finite volume approximations in CFD. *European Journal of Mechanics/B Fluids*, **24**(1), 1–17.

[Hirt *et al.* , 1997] Hirt, C. W., Amsden, A. A., & Cook, J. L. 1997. An arbitrary Lagrangian-Eulerian computing method for all flow speeds. *Journal of Computational Physics*, **135**(2), 203–216.

[Ibrahimbegović, 2009] Ibrahimbegović, Adnan. 2009. *Nonlinear solid mechanics: Theoretical formulations and finite element solution methods*. Springer.

[Kassiotis *et al.* , 2010a] Kassiotis, Christophe, Niekamp, Rainer, Ibrahimbegović, Adnan, & Matthies, Hermann G. 2010a. Partitioned procedure for strongly coupled fluid-structure problems. Part I: implicit algorithm, stability analysis and validation examples. *Computational Mechanics*, **Published online**.

[Kassiotis *et al.* , 2010b] Kassiotis, Christophe, Niekamp, Rainer, Ibrahimbegović, Adnan, & Matthies, Hermann G. 2010b. Partitioned procedure for strongly coupled fluid-structure problems. Part II: implementation aspects, re-using existing software as components and nested parallel computations by using CTL. *Computational Mechanics*, **Published online**.

[Kassiotis *et al.* , 2010c] Kassiotis, Christophe, Ibrahimbegović, Adnan, & Matthies, Hermann G. 2010c. Partitioned solution to fluid-structure interaction problems in application to free-surface flows. *European Journal of Mechanics – B/Fluids*, **29**, 510–521.

[Küttler & Wall, 2008] Küttler, Ulrich, & Wall, Wolfgang A. 2008. Fixed-point fluid-structure interaction solvers with dynamic relaxation. *Computational Mechanics*, **43**(1), 61–72.

[Le Tallec & Mouro, 2001] Le Tallec, P., & Mouro, J. 2001. Fluid structure interaction with large structural displacements. *Computer Methods in Applied Mechanics and Engineering*, **190**(24-25), 3039–3067.

[Markovič *et al.* , 2005] Markovič, Damijan, Niekamp, R., Ibrahimbegović, Adnan, Matthies, H. G., & Taylor, R. L. 2005. Multi-scale modeling of heterogeneous structures with inelastic constitutive behavior : Mathematical and physical aspects. *International Journal of Engineering Computations*, **22**, 664–683.

[Matthies & Steindorf, 2003] Matthies, Hermann G., & Steindorf, Jan. 2003. Partitioned strong coupling algorithms for fluid-structure interaction. *Computers and Structures*, **81**, 805–812.

[Matthies *et al.* , 2006] Matthies, Hermann G., Niekamp, Rainer, & Steindorf, Jan. 2006. Algorithms for strong coupling procedures. *Computer Methods in Applied Mechanics and Engineering*, **195**, 2028–2049.

[Monaghan, 1992] Monaghan, JJ. 1992. Smoothed particle hydrodynamics. *Annual review of astronomy and astrophysics*, **30**(1), 543–574.

[Niekamp *et al.* , 2009] Niekamp, Rainer, Markovič, Damijan, Ibrahimbegović, Adnan, Matthies, Hermann G., & Taylor, Robert L. 2009. Multi-scale modelling of heterogeneous structures with inelastic constitutive behavior: Part II–software coupling implementation aspects. *Engineering Computations*, **26**, 6–28.

[Patankar, 1980] Patankar, S. V. 1980. *Numerical heat transfer and fluid flow.* Washington, DC: Hemisphere Publishing Corporation.

[Sobey, 1998] Sobey, R. J. 1998. *Linear and Nonlinear Wave Theory.* Lecture Note. Leichtweiß-Institut für Wasserbau, Braunschweig.

[Stoker, 1992] Stoker, J. J. 1992. *Water Waves: The Mathematical Theory and Applications.* New-York: Wiley-Interscience.

[Ubbink & Issa, 1999] Ubbink, O., & Issa, RI. 1999. A method for capturing sharp fluid interfaces on arbitrary meshes. *Journal of Computational Physics*, **153**(1), 26–50.

[Walhorn *et al.* , 2005] Walhorn, Elmar, Kölke, Andreas, Hübner, Björn, & Dinkler, Dienter. 2005. Fluid-structure coupling within a monolithic model involving free surface flows. *Computers and Structures*, **83**(25-26), 2100–2111.