# Fluid-Structure Interaction Problems Solution by Operator Split Methods and Efficient Software Development by Code-Coupling

Adnan Ibrahimbegovic[*1a]
Christophe Kassiotis[2], Rainer Niekamp[3]

[1]*Sorbonne Universités – UT Compiègne, Chaire de Mécanique,* [a] *Institut Universitaire de France*
[2] *ASN (detached from ENPC), France,* [3]*University of Duisburg-Essen, Institut fur Mechanik, Germany*

**Abstract.**   An efficient and general numerical strategy for fluid-structure interaction problems is presented where either the fluid or the structure part are represented by nonlinear models. This partitioned strategy is implemented under the form of code coupling that allows to (re)-use previous made developments in a more general multi-physics context. This strategy and its numerical implementation is verified on classical fluid-structure interaction benchmarks, and then applied to the impact of tsunamis waves on submerged structures.

**Keywords:** fluid-structure interaction, partitioned strategy, weak/strong coupling

## 1. Introduction

Interaction of fluid and structures is a key part for a variety of physical systems. The range of examples that come easily to mind in the engineering domain is vast and cover all the scales: from bio- and medical- (where the scale of problems is often less than the millimeter) to civil engineering (where it can be the hundreds of meters).

However, the development of numerical strategy to solve the already by themselves complex problems of solid and fluid was mostly carried by different research communities. This lead to efficient but often incompatible solvers.

It is interesting to note that the fluid-structure interaction problem is often tackled with the will to extend one of the methods traditionally applied to one of the sub-problems to the whole coupled. For example, one can cite fluid Eulerian description applied to solid or complex finite elements methods applied to fluid. If those strategies often lead to interesting results and new point of view, they often necessitate tremendous software implementation and basic testing. Furthermore, as one start from more or less "nothing" in a domain, the time scale before reaching high-level model coupling is often too long. Finally, as engineers habits are often hard to change, and their previous work as imperfect as it is, has to be conserved, those methods are restricted to the research area.

In this paper, we propose a totally different approach that rely on partitioned strategy (which is often used in for multi-physics problems) and its software armed hand: code coupling. The work herein propose to couple in a generic way a Finite Element Method Code for the solid part and to use Computational Fluid Dynamic solver with Finite Volume Method. This allows us at the first step to solve the complete Navier-Stokes equation within Arbitrary Lagrangian Eulerian (ALE) framework, and

---

Paper on plenary talk at ACEM16: *Corresponding author, Professor, E-mail: *adnan.ibrahimbegovic@utc.fr*
[2] Ph.D., E-mail: christophe.kassiotis@gmail.com
[3] Ph.D., E-mail: rainer.niekamp@uni-due.de

geometrically non-linear description of the structure motion. As the software coupled provide different models this is also possible to model more complex flows (with turbulence for instance) and solid materials (non-linear plasticity or damaging for instance).

Moreover, the ability to couple solving strategies and their associated codes will be emphasized by the coupling of three codes relying on different discretization schemes: the Finite Volume Method (FVM), Boundary Element Method (BEM) and Finite Element Method (FEM) are employed together in the presented work.

The outline of this paper is as follows. We begin by describing the solution strategies for the fluid problem in Sec. 2, the long-running wave propagation approach typical of tsunami propagation in Sec. 3 and the solid mechanics model in Sec. 4. As follow up, in Sec. 5 we describe the coupling strategy and its software implementation. In Sec. 6 we present the results of illustrate simulations; the first one is the classical benchmark fluid-structure interaction problem, and the second on introducing our modelling of waves' impact on structures.

## 2. Problem of the fluid flow

### 2.1 Fluid model

We consider a fluid domain $\Omega_f$ where an incompressible, viscous, isothermal and isotropic Newtonian flow takes place. The Eulerian description of this flow in term of continuity and momentum equilibrium equation gives:

$$\nabla \cdot \mathbf{v} = 0 \quad \text{in } \Omega_f \times [0, T]$$
$$\partial_t \mathbf{v} + \nabla \cdot \mathbf{v} \otimes \mathbf{v} - \nabla \cdot 2\nu \mathbf{D}(\mathbf{v}) = -\frac{1}{\rho}\nabla p \quad \text{in } \Omega_f \times [0, T] \tag{1}$$

where $p$ denotes the kinematic pressure field, $\mathbf{v}$ the velocity. The law of behavior of the flow links stress and velocity gradient $\mathbf{D}(\mathbf{v})$ through the kinematic viscosity $\nu_f$ which is the dynamic viscosity $\mu_f$ divided by the fluid density $\rho_f$.

Added to this equations, one has also to consider boundary equations at $\Gamma_f = \partial\Omega_f$. They can be of the Dirichlet or Neumann kind. One as also to set the initial conditions for the velocity field in the whole domain $\Omega_f$.

### 2.2 Finite Volume Method for the Navier-Stockes Equations

Two techniques are mainly used to solve this kind of problem: Finite Element Methods (FEM), in which predetermined shape functions are used to enforce this equation in a weak sense and Finite Volume Methods as used here in which the fields are averaged over a particular volume. For this method, the whole volume $\Omega_f$ is divided into a set of discrete volumes $(\delta\Omega_{f,i})_{i=1,N}$ such that the whole domain is covered ($\Omega_f = \cup_{i=1}^{N} \delta\Omega_{f,i}$) without any overlapping ($\delta\Omega_{f,i} \cap \delta\Omega_{f,j} = \emptyset$; $i \neq j$). The fluid flow equations are then integrated over each finite volume: The particular $\delta\Omega_{f,i}$ is the so-called control volume.

The Navier-Stokes equations (1) are integrated in each control volume. The use of Gauss's theorem transform each integration of a divergence term into surface integrated flux

$$\partial_t \int_{\delta\Omega_f} \mathbf{v} d\Omega + \oint_{\delta\Gamma_f} \mathbf{d}\Gamma \cdot \mathbf{v} \otimes \mathbf{v} - \oint_{\delta\Gamma_f} \mathbf{d}\Gamma \cdot 2\nu \mathbf{D}(\mathbf{v}) = \oint_{\delta\Gamma_f} \frac{1}{\rho} p \, \mathbf{d}\Gamma \tag{2}$$

The time integration of the spatially discretized equations rely on explicit methods to approximate time-derivatives. On the other hand, as the continuity equation as strictly to be enforced, an iterative split strategy named PISO is used (Jasak 1996).

### 2.3 Arbitrary Lagrangian-Eulerian description

Fluid-Structure interaction problems usually lead to unsteady moving domain for the fluid part. Traditional Computational Fluid Dynamic programs solve the fluid equations on a fixed-in-space Eulerian grid. A classical approach to overcome this difficulty is to consider the so-called Arbitrary Lagrangian Eulerian (ALE) method where the grid is moved arbitrary inside the fluid domain, following the movement of the solid boundary.

However, this leads to new difficulties: knowing the new shape of the solid boundary, how can one conserve the quality and the validity of the fluid mesh? One traditional and widely used method is to consider the mesh as a pseudo-structural system. The mesh points are linked with a kind of spring analogy, and one can describe the fluid mesh motion by analogy with a solid loaded at its boundary.

This pseudo-structural system is known not to succeed for too large motion cases. This is why one has often to consider non-linear pseudo structural systems and sometimes need to couple them with re-meshing tools. Even using additional concept to reduce the computational cost, this procedure remain expensive and does not solve all the problems. Some recent methods relying on the use of a linear Laplace equations, which is unconditionally bounded, present reliable mesh motion at a quit reasonable cost (Jasak and Tukovic 2007).

The ALE strategy leads in fact to a three-field coupling: the fluid, the structure and the fluid mesh motion. However, modern Fluid codes often propose an implementation of ALE, and fortunately the coupling between the fluid and the mesh motion need not to much improved compared to recent proposals (Jasak and Tukovic 2007).


## 3. Problem of fluid wave propagation

### 3.1 Potential flow description

To simulate and analyse this phenomenon, we generate solitary waves in a 3D numerical wave tank (NWT). The NWT solves fully non-linear potential flow equations with a free surface, using a high-order boundary element method and a mixed Eulerian–Lagrangian time updating. The former has allowed the accurate simulation of 3D overturning waves and the latter has led to at least a one-order of magnitude increase in the NWT computational efficiency. This made it possible to generate finely resolved 3D overturning waves and analyse their geometry and kinematics. For more detail about this method, see (Fochesato, Grilli and Dias 2007).

To be more precise, the surface wave problem consists in solving Laplace's equation on the velocity potential $\mathbf{v} = \nabla \phi$ in the whole domain filled with fluid $\Omega_\omega(t)$, as we assume in this domain inviscid and irrotational flow:

$$\nabla \phi = 0, \quad \forall \mathbf{x} = (x_1, x_2, x_3) \in \Omega_\omega(t) \tag{3}$$

The domain $\Omega_\omega(t)$ is bounded above by a moving free surface (interface between air and water) and below by a fixed solid boundary. The free surface is represented by $F : (\mathbf{x}, t) \rightarrow \eta(x_1, x_2, t) - x_3 = 0$ and the bottom is given by $G : (\mathbf{x}, t) \rightarrow h(x_1, x_2) + x_3 = 0$. This free surface $\eta$ is the new fundamental unknown of the problem. Two boundary conditions are required:
- The kinematic condition:
$$\frac{DF}{Dt} = \frac{D\eta}{Dt} - \frac{Dx_3}{Dt} = \partial_t \eta + \nabla \phi \cdot \nabla \eta - \partial_{x_3} \phi = 0 \tag{4}$$
- The dynamic condition: the normal stress at the free surface is given by the difference in pressure. Bernoulli's equation evaluated on the free surface gives:

$$\partial_t \phi + \frac{1}{2} \|\nabla \phi\|^2 + g\eta = 0 \tag{5}$$

The only condition to enforce at the bottom is the kinematic condition:

$$\nabla \phi \cdot \nabla h + \partial_{x_3} \phi = 0 \tag{6}$$

## 3.2 Boundary element method

As one can see, the equation that one need to fulfill in the domain $\Omega_\omega(t)$ is simple compare to the one at its boundaries. The main idea is to use Green's second identity transforming this equation into a boundary integral equation:

$$\alpha(\mathbf{x}_l)\phi(\mathbf{x}_l) = \int_{\partial \Omega_\omega(t)} [\partial_n \phi(\mathbf{x}) G(\mathbf{x}, \mathbf{x}_l) - \phi(\mathbf{x}) \partial_n G(\mathbf{x}, \mathbf{x}_l)] d\partial\Omega_\omega(t) \tag{7}$$

As seen in the equations above, the problem can thus be defined only on the boundary. The spatial discretization is defined by Boundary Element Method (BEM). The time integration is done by an explicit scheme (Fochesato, Grilli and Dias 2007). Some numerical aspects of the NWT are improved, such as the accurate computation of higher-order derivatives on the free surface and the implementation of the fast multipole algorithm for the spatial solver.

One can notice that the breaking of the wave is at the moment still an open problem, where BEM shows its limits. Some potentially successful strategies are based on weakly coupled BEM-VOF, are proposed in (Lachaume et al. 2003), or BEM-SPH.

## 4. Problem of structural vibrations

### 4.1 Structural model

The structural model is set in the Lagrangian framework. The governing equation for a structure describes the momentum conservation, which is also known as the Cauchy equations of motion:

$$\rho_s \partial_t^2 \mathbf{u} = \nabla \cdot \mathbf{S} + \rho_s \mathbf{b} \quad \text{in } \Omega_s \times [0, T] \tag{8}$$

where $\rho_s$ denotes the solid density, $\mathbf{u}$ the displacement field, $\partial_t^2 \mathbf{u}$ the acceleration, $\mathbf{b}$ the body force and $\mathbf{S}$ the second Piola-Kirchhoff stress tensor (that can be linked to the true or Cauchy stress tensor, denoted as $\boldsymbol{\sigma}$ in direct tensor

The Cauchy equation is solved with initial condition imposed to the displacement and velocity field on the whole domain $\Omega_s$ and boundary conditions of kind Dirichlet (or displacement): notation).

$$\mathbf{u} = \bar{\mathbf{u}} \quad \text{on } \Gamma_D \times [0, T] \tag{9}$$

and Neumann (or traction):

$$\mathbf{S} \cdot \mathbf{n} = \bar{\mathbf{t}} \quad \text{on } \Gamma_N \times [0, T] \tag{10}$$

To complete this set of partial differential equations one needs to link the displacements (or its derivatives field) and the stresses through the law for constitutive behavior. We will assume in the following the linear-elastic material model based on St.Venant-Kirchhoff constitutive equation, which are linking the second Piola-Kirchhoff tensor $\mathbf{S}$ and the Green-Lagrange strain tensor $\mathbf{E}$:

$$\mathbf{S} = C{:}\mathbf{E} \tag{11}$$

where $C$ denoting the (linear) constitutive tensor. The non-linearity of the problem come from the large displacement described by the following relation between Green- Lagrange tensor $E$ and the material deformation gradient $\mathbf{F} = \nabla \mathbf{u}$:

$$\mathbf{E} = \frac{1}{2}(\mathbf{F}^T \mathbf{F} - \mathbf{I}) \tag{12}$$

### 4.2 Finite Element Method

It is not the goal of the paper to give complete presentation for the Finite Element Method (FEM) implementation. For more details, we refer readers to (Ibrahimbegovic 2006,2009). We will only say that in the problem considered in Sec. 6, the solid are discretized on 9-Nodes plane stress elements for flexion dominated problems where the 4-node finite elements are known to have poor behavior. The time integration is carried out by a generalized-α Method and the Newton iterative scheme.

## 5. Partitioned strategy and its software implementation

### 5.1 Coupling algorithm

For fluid-structure interaction problems, and especially for aero-elastic problems, coupling algorithm often relies on weak coupling strategies. As said in Section 3, weakly coupling strategies are sometimes proposed to solve wave propagation near the coasts as BEM code becomes unavailable. Such strategies, where the results from one computation are given to the next one without any "come-back", present the advantages of simplicity and little cost, but even if the error propagation at each exchange of data can be estimate, it can lead to over- or under-estimate physical instabilities' phenomenon (Arnold and Gunther 2001, Matthies, Niekamp, and Steindorf 2006).
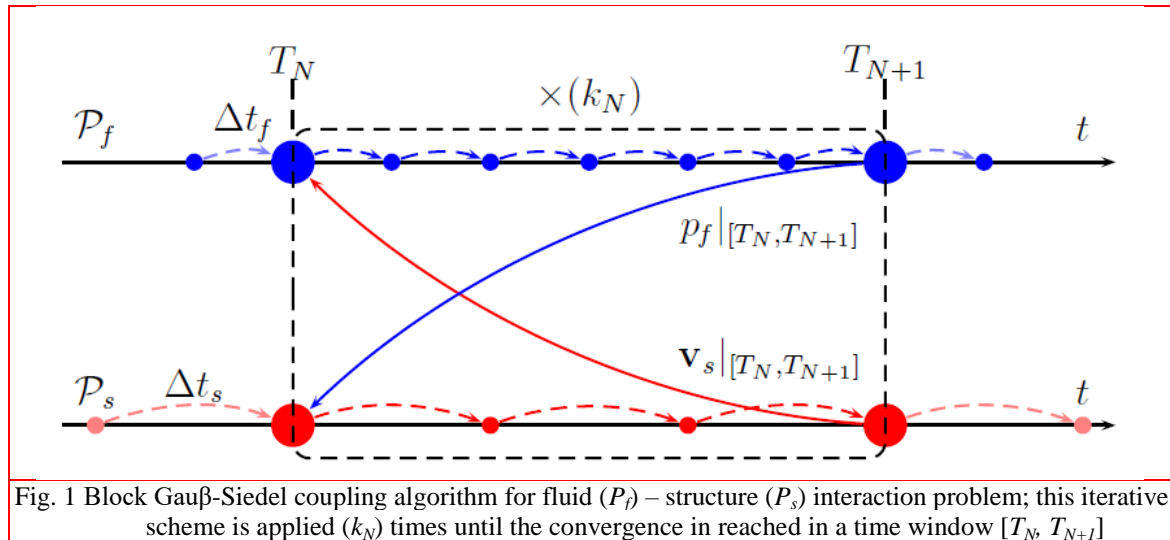


Fig. 1 Block Gauß-Siedel coupling algorithm for fluid ($P_f$) – structure ($P_s$) interaction problem; this iterative scheme is applied ($k_N$) times until the convergence in reached in a time window [$T_N$, $T_{N+1}$]

In our presentation we will compare both weak-coupling and the so-called block-Gauß-Seidel strong coupling algorithm. For each coupling, there remain the following questions:

- which data (physical quantities) exchange, and in which order? In Fig. 1 we present for the sake of simplicity the block-Gauß-Seidel algorithm for a two field (fluid and structure) problem.
- is the coupling algorithm stable. In fact, even if each sub-problem is computed by a stable and converging algorithm, some coupling algorithm can diverge; for a more detailed discussion, we refer to (Arnold and Gunther 2001, Matthies, Niekamp, and Steindorf 2006).

As one can see in Fig. 1 we consider independent time integration solvers in a window (i.e. $t \in$ [$T_n$, $T_{n+1}$]). For this reason not only the value at synchronization points $T_n$ or $T_{n+1}$, but the interpolated evolution of considered variables on the whole window as to be exchanged.

We emphasize that one of the big advantages of the proposed algorithm is the possibility to use different

time steps for the sub-problems considered. For instance, the integration scheme for the VFM based code is explicit, and in order to respect the Courant condition, small time steps are required. On the other hand, the structure is integrated with implicit schemes that allows bigger time step. We are able to couple simulations taking advantage of the natural time stepping arising from each particular problems and its discretization (see Sec. 6 for examples).

*5.2 Component Template Library framework*

The different parts of our problem as been solved by different research teams, and one of the goal of our work is to show the possibility to re-use existing codes in a multi-physics context.

- The NWT problem is solved by a Fortran code using C subroutines to solve the fast multipole algorithm [3]. The component based on this code is named conuwata.
- The VFM code used is OpenFoam[1], a very general C++ library to solve fluid problems (Weller, Tabora, Jasak, and Fureby 1998). From this code, a component named ofoam was developed.
- The mechanical part is solved by FEAP[2], a FEM code programmed in Fortran (Zienkiewicz and Taylor 2005). The mechanical component based on FEAP is coFeap.


Fig. 2: Coupled three components for the BEM/VOF/FEM problem.
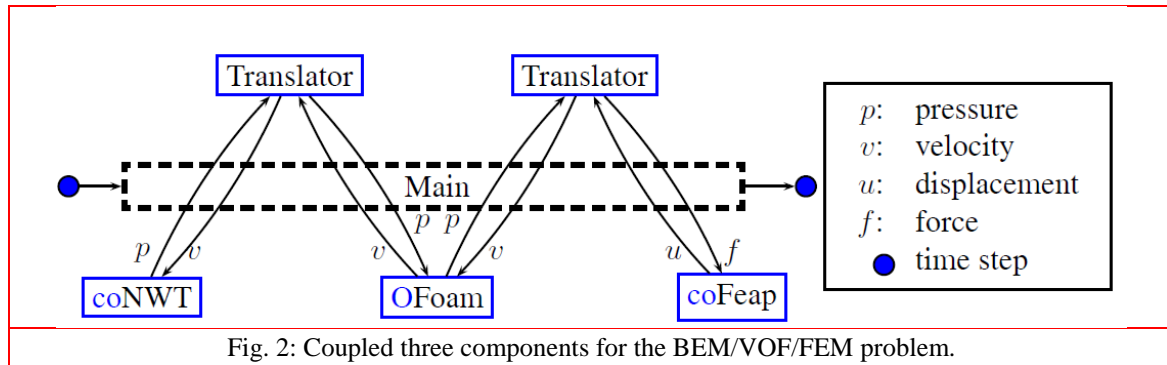
[1]http://www.opencfd.co.uk/openfoam
[2]http://www.ce.berkeley.edu/~rlt/feap
[3]http://www.wire.tu-bs.de/forschung/projekte/ctl/e_ctl.html

Each program is embedded in a component. The communication between the component above is insured by the Component Template Library (CTL)[3] developed at the Institute for scientific computing (Matthies, Niekamp, and Steindorf 2006, Srisupattarawanit, Niekamp, and Matthies. 2006), and each component can be executed on a single-processor personal computer, or on different processors on a cluster, or yet on different machines connected with the network communication. As illustrated in Fig 2, a translator is in charge to make the data from one component matching their target component. This is required in particular when one needs to change their nature (from pressures to forces for instance) or the space interpolation (for non-matching meshes).

# 6. Numerical examples

## 6.1 Oscillating flexible structure in a flow

### 6.1.1 Problem's description

In the first example, we will consider a benchmark fluid-structure interaction problem involving two components for a fluid solved by FVM, and a structure solved by FEM in order to validate our coupling strategy. This benchmark was first introduced by (Wall and Ramm 1998). The main goal (see illustration

in Fig. 3) is to solve the motion of a thin appendage behind a bluff body in a fluid flow at a Reynold number around 100.



Fig. 3: The benchmark used for FSI problems
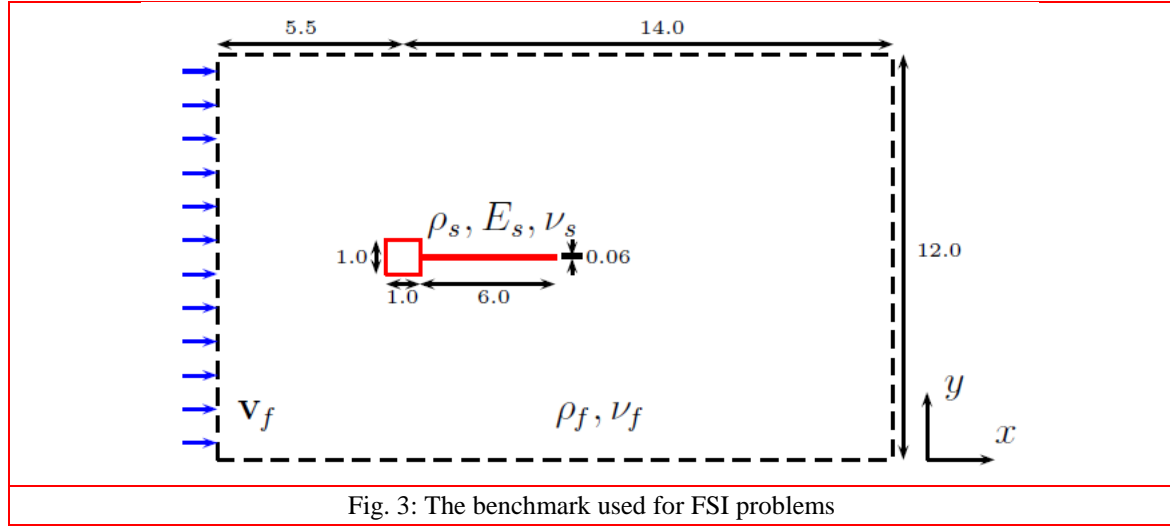
For the fluid, the incompressible Navier-Stokes equations are solved. The discrete model used for this computation uses around 6000 finite volume cells (Q1). For the constitutive behavior of solid, we consider the Saint-Venant-Kirchoff type material, based upon the finite deformation measure and the plane stress description. The discrete model for solid part is constructed with 40 9-node finite elements (Q9).
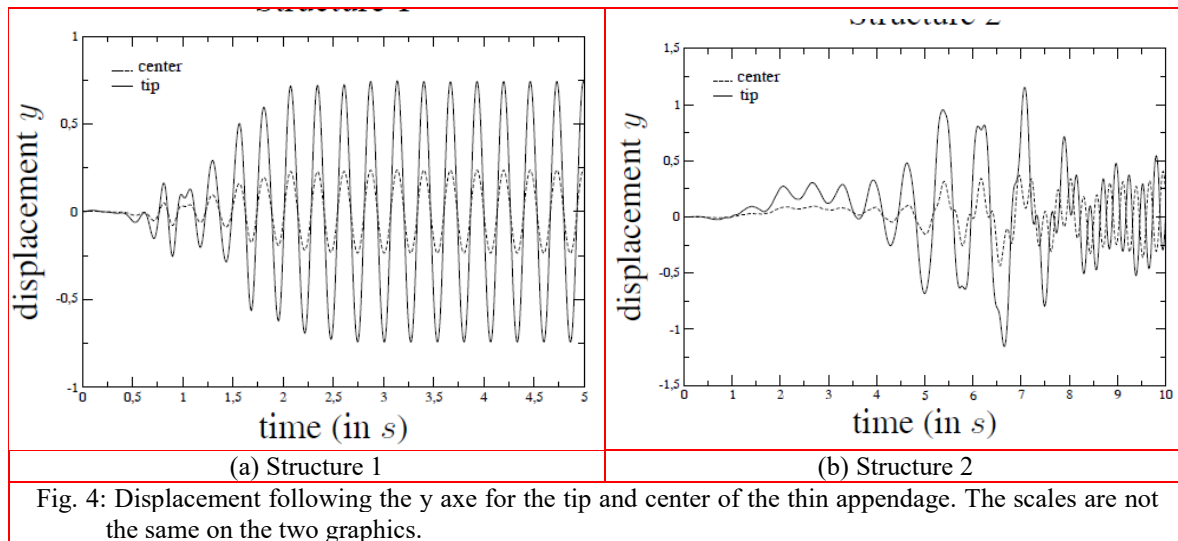
The *material* properties are the following:
- for the fluid: dynamic viscosity $v_f = 1.51 \cdot 10{-}14$ as $\rho_f = 1.18 \cdot 10{-}3$ (like air) and $\mu_f = 1.82 \cdot 10{-}4$ ($10\times$ more than air).
- for the solid: Two structure are considered. The first one, with Young's modulus $E_{s,1} = 2.5 \cdot 106$, Poisson's ratio $v_{s,1} = 0.35$ and density $\rho_{s,1} = 0.1$. The second one, with Young's modulus $E_{s,2} = 2.0 \cdot 106$, Poisson's ratio $v_{s,2} = 0.35$ and density $\rho_{s,2} = 2.0$.

The boundary conditions are imposed as follow:

- for the fluid: imposed velocity $\mathbf{v}_f = [51.3, 0, 0]^T$ in input, slip condition for the upper and lower part of the mesh, "do-nothing" output condition.
- for the solid: imposed null displacement at the origin of the appendage.
- for the interface: kinematic continuity ($\mathbf{v}_s = \mathbf{v}_f$) and dynamic equilibrium ($p_f = \rho\sigma_s\mathbf{n}$) are imposed

The initial condition are computed from a stationary flow regime with fixed structure for the fluid and are null for the solid. The fluid-structure interaction partitioned strategy allows to use different time step for the fluid and the structure. We consider here $\Delta t_f = 0.0002$ and $\Delta t_s = 0.005$ for the computation time interval [0, 5].

|                    |                    |
| ------------------ | ------------------ |
| (a) Structure 1    | (b) Structure 2    |

Fig. 4: Displacement following the y axe for the tip and center of the thin appendage. The scales are not the same on the two graphics.

In Fig. 4, we present the displacement of the thing appendage at two points pick at the center and the tip. As one can see the first structure is excited in the first eigenmode (first mode's oscillation period computed for a linear structure is 0.33) whereas the second one is excited in the second eigenmode (first mode's oscillation period computed for a linear structure is 1.65) when the system take pseudo-periodic oscillations.

[4]like in (Wall and Ramm 1998, 1999) the value are given without units. A careful reading can see that the unit for dimensions is $10^{-1}m$, the other units are the international system one (kg, s. . . )

To further illustrate the results obtained in this example, we propose flow snapshots for the first and second fluid-structure coupling problem in Fig. 5.

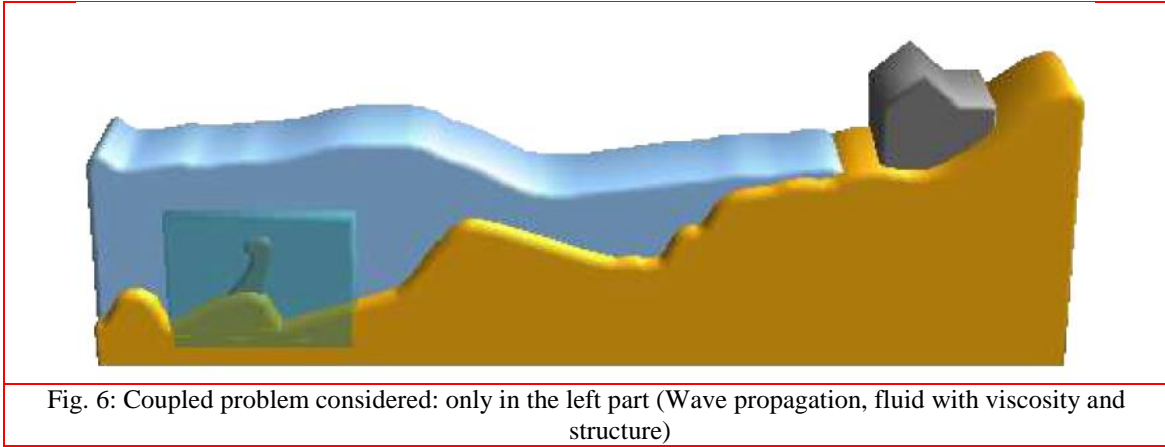Fig. 5: Pressure and velocity fields in the fluid around the two structures considered
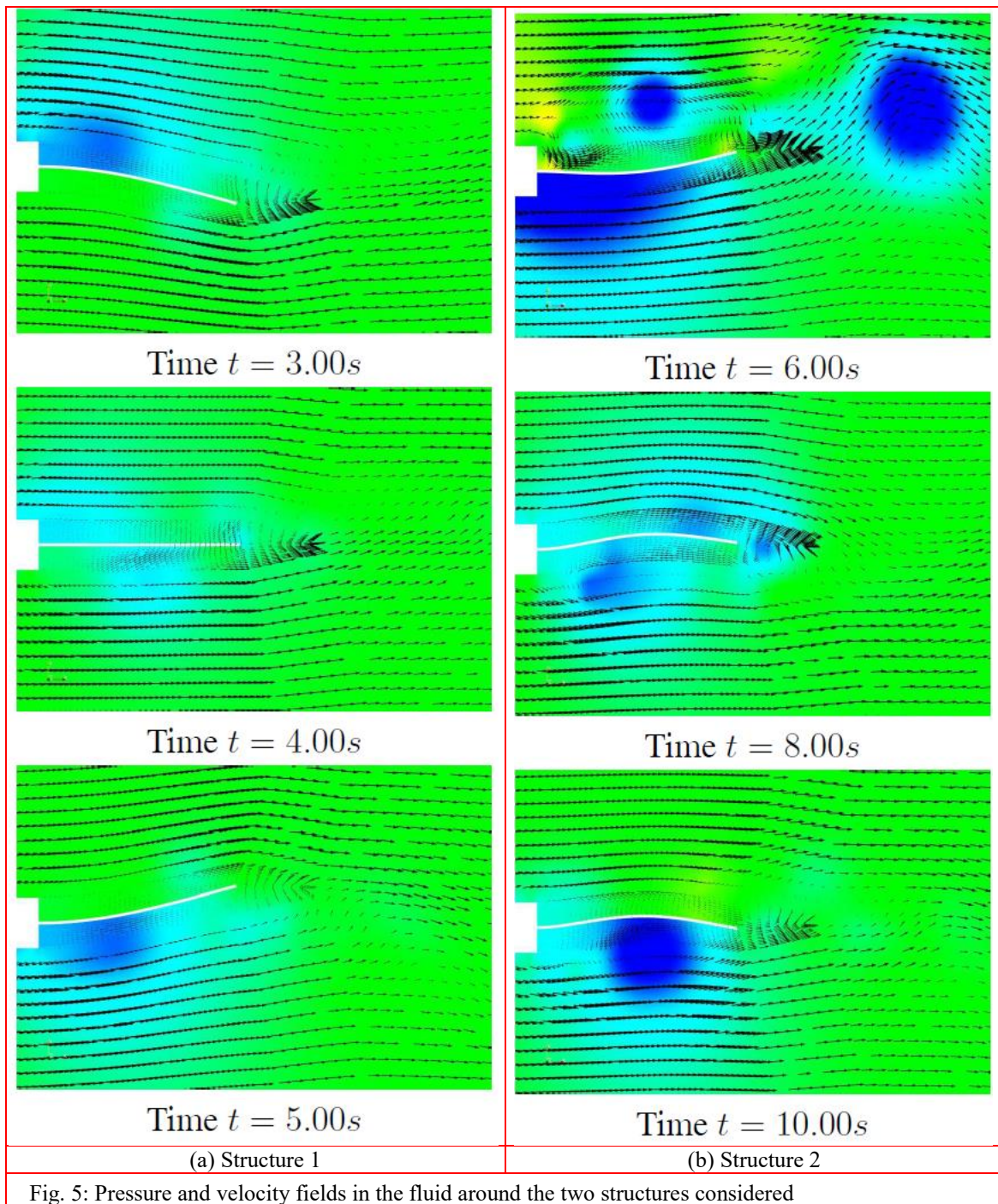
For the example considered, all computations are run with a Gauß-Seidel implicit coupling algorithm. This iterative schema necessitates around 4 or 5 iterations to converge at each time step. However, the computation coast is increased by a factor 4 to 5 compared to an explicit coupling scheme.

## 6.2 Submerged structure impacted by a Tsunami

As the second example, we focus on the modeling of the impact of tsunami waves on coastal protection (see Fig. 6); such problem is intrinsically a multi-physics one, and need the use of complex numerical
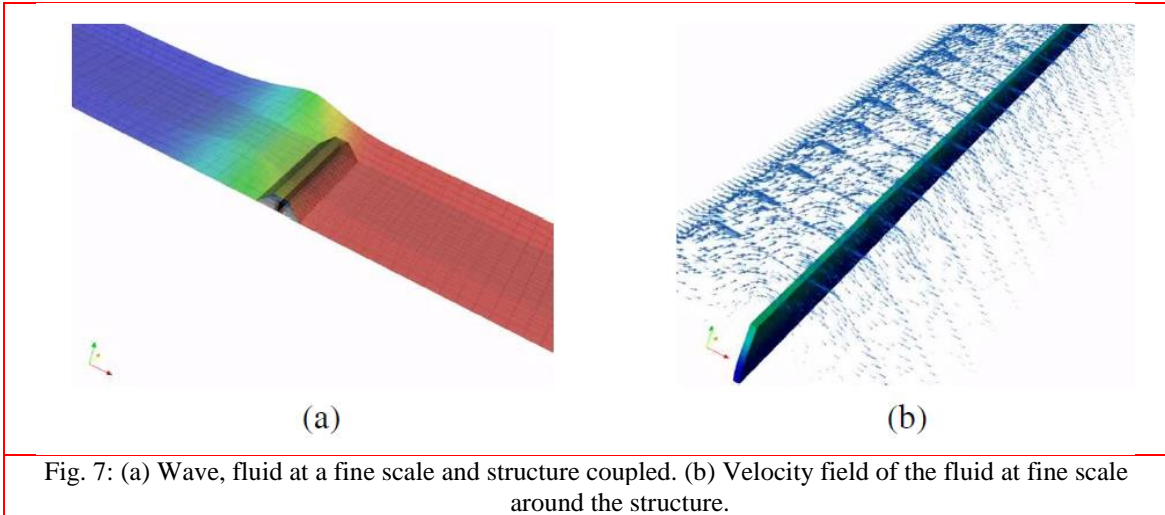
tools to solve:



Fig. 6: Coupled problem considered: only in the left part (Wave propagation, fluid with viscosity and structure)

- The propagation of tsunami wave, which is a fully non-linear problem is the domain considered (Section 3).
- The viscosity effect that cannot be neglected very close to the beach or near the coastal engineering protections (Section 2).
- The mechanical behavior of the structures: very complex non-linear laws that represent at the macro-scale the behavior of engineering materials are now be implemented on Finite Element Method (FEM) based codes (Section 4).

The coupling between the sub-problems is described in Section 5.

In Fig. 7 we present as preliminary results some snapshots from the solved coupled problem. One can notice that our first computations are based on explicit coupling in order to save computational time.



Fig. 7: (a) Wave, fluid at a fine scale and structure coupled. (b) Velocity field of the fluid at fine scale around the structure.

## 7. Conclusions

We proposed quite general coupling strategies for multi-physics coupling and interaction problems, illustrated in detail for fluid-structure interaction. The proposed framework of component technology allows to reuse existing codes developed by independent teams for a particular specialized application

and include them directly in a more general context of coupling different application domains. This feature was truly emphasized by the use of three components relying on different discretization methods (BEM/FVM/FEM). This lead to the possibility to couple non-trivial models for each sub-problem considered herein (e.g. surface waves represented with high-order polynomials functions, non-linear finite elasticity structural elements. . . ). Furthermore, the coupling algorithm proposed herein allows to preserve the time integration scheme independence. Another important feature is the possibility to use the different time steps naturally arising from the physical problem and its discretization adapted to each sub-problems in order to reduce the computational coast.

A useful direction for future research is a multi-scale modeling of tiny obstacles that can represent, for instance, the see flora that, as shown in recent tsunami impacts in South-Est Asia, have proved to be of great importance in slowing down the wave and reducing the resulting damage. However, for taking into account this kind of obstacle, a lot of parameters are not precisely known, and this uncertainties need to be model through the use of stochastic tools.

## Acknowledgments

## References

Arnold M. and M. Gunther (2001). Preconditioned dynamic iteration for coupled differential-algebraic systems. BIT Numer. Math., **41**:1–25

Fochesato S., S. Grilli, and F. Dias (2007). Numerical modeling of extreme rogue waves generated by directional energy focusing. Wave Motion, **44**:395–416

Ibrahimbegovic A. (2006). Mécanique non linéaire des solides déformables : Formulation théorique et résolution numérique par éléments finis. Hermès Sciences –Lavoisier, Paris

Ibrahimbegovic A. (2009), Nonlinear Solid Mechanics: Theoretical Formulation and Finite Element Solution Methods, Springer, Berlin

Jasak H. (1996). Error Analysis and Estimation for the Finite Volume Method with Applications to Fluid Flows. Ph.D. Dissertation, Department of Mechanical Engineering, Imperial College of Science, Technology and Medicine

Jasak H and Z. Tukovic (2007). Automatic mesh motion for the unstructured finite volume method. Transactions of *FAMENA*, **30**(2):1–18

Lachaume C., B. Biausser, S. T. Grilli, P. Fraunie, and S. Guignard (2003). Modeling of breaking and post-breaking waves on slopes by coupling of bem and vof methods, in 13th Offshore and Polar Engineering Conference (ISOPE), 353–359

Matthies H., R. Niekamp, and J. Steindorf (2006). Algorithms for strong coupling procedures. Computer Methods in Applied Mechanics and Engineering, **195**:2028–2049

Srisupattarawanit T, R. Niekamp, and H. G. Matthies (2006). Simulation of nonlinear random finite depth waves coupled with an elastic structure. Computer Methods in Applied Mechanics and Engineering, **195**:3072–3086

Wall W.A. and E. Ramm. Fluid-structure interaction based upon a stabilized (ALE) finite element method. Sonderforschungsbereich 404, Institute of Structural Mechanics, 1998

Wall W.A.. Fluid-Struktur Interaktion mit stabilisierten Finiten Elementen. Phd Thesis, Institut f¨ur Baustatik, Universit¨at Stuttgart, 1999

Weller H.G., G. Tabora, H. Jasak, and C. Fureby. A tensorial approach to computational continuum mechanics using object-oriented techniques. Computers in physics, 12(6):620–631, 1998.

Zienkiewicz O.C. and R. L. Taylor (2005). The Finite Element Method. Butterworth- Heinemann, Oxford, 5th ed.