Methods and applications of computational fluid-structure interaction for nvh problems - a review

Methods and applications of computational fluid-structure interaction for nvh problems - a review

D. Fabbri, F. Bruzzone, C. Rosso, R. Arina
Politecnico di Torino, Department of Mechanical and Aerospace Engineering,
Corso Duca degli Abruzzi, 24, Torino, 10129

Abstract
The extensive application field of the interactions between fluids and structures makes studying these phenomena crucial in the multiphysics domains. The numerical simulations are the primary solution for investigating the fundamental physics involved in complex interactions. This review delves into several methods employed to solve fluid-structure problems. A fundamental aspect discussed is the distinction between monolithic and partitioned approaches. While the monolithic approach involves formulating a unified set of system equations for the problem, the partitioned approach treats the fluid and solid domains separately. In addition, the two approaches differ regarding discretization of problem domains, solution strategies and computational costs. Through a comprehensive analysis of numerical methods and their applications, this review is supposed to be a starting point for those who want to approach the topic of multiphysics interactions

1 Introduction

Fluid and structure interaction (FSI) is one of the most fascinating topics in the multiphysics field, but at the same time is broad, complex, not fully comprehended. In recent years some studies have been conducted on several FSI problems, *Hessenthalera et Al.* [1] present 16 different analytical solutions, considering transient and static scenarios as well as linear or hyperelastic solids. Experimental tests are also limited to the difficulty of having a faithful test set up.

Therefore, to solve a fluid-structure interaction problem usually a numerical method is used. As different levels of interaction are possible, it is important to understand the applicability of each method according to the specific case study, this review delves into the differences between each technique.

In the numerical methods, the solid can be considered as a rigid body, otherwise a matrix stiffness is defined and the body deformation it is not negligible. The fluid can be categorized as liquid or gas, for liquids-solid interaction the assumption of incompressible flow is typically applied. However, when gases interact with solids the incompressible flow assumption is justified only when the Mach number of the flow is below 0.3.

Fluid-structure interaction can also be classified according to the approach used to find solutions. The two main approaches to an FSI problem are the monolithic and the partitioned. The first treats fluid and solid as a single integrated system, the result is a nonlinear system of equations that is solved simultaneously to the mesh movement, instead the partitioned considered the fluid and solid as separate systems linked through an interface.

In [2] and [3] *Felippa and Park* explained the advantages of partitioned solution procedures, focusing also on their stability, explaining that the main source of instability was the delayed feedback of fluid radiation damping energy from the fluid equation to the structural equation. The stabilized form was developed by tailoring the equations of motion adding a homogeneous damping term. To improve the convergence and stability of the partial solver an additional iteration loop can be used, this leads to a strongly coupled partitioned solution [4]. The partitioned approach presents two potentially significant advantages, it requires relatively few modifications to single-field analyzers, this modularity is valuable given the current costs associated

with software development, modification, and maintenance. The computational efficiency is also improved as the cost per time step of the staggered solution is approximately the same as processing the component fields separately. This makes the staggered approach economically attractive when time step size restrictions are not a concern. In case of strong interaction, a monolithic approach ensures convergence and stability of the coupled solution, it consists in simultaneous procedures that solve in a single iteration loop the entire coupled system. The unique system of equations usually is solved using Newton's method [5], however, consolidating multiple equations into a single system can result in ill-conditioned system matrices, sometimes containing zero entries on the diagonal. Consequently, it is challenging to find appropriate preconditioners that can efficiently solve these large, sparse, linearized systems on parallel computing platforms. Therefore developing, implementing, and validating an entirely new solver becomes necessary.

In [6] an efficient simultaneous solution of a nonlinear elastic structure interacting with incompressible viscous fluid is analyzed, in the same iteration loop both the system of non linear equations and the fluid mesh movements are solved. The space-time finite element method provides a consistent discretization of both time and space domains, this allows the natural description of moving fluid domains, since isoparametric space–time elements are adaptable in the time direction. The work done by *Heil et Al.* [7] compares the two approaches in different test cases, it turns out that monolithic solvers are competitive, in CPU time, even in the cases with weak interaction. Instead for strongly coupled systems, the monolithic approach must be used as the partitioned tends to diverge. Nevertheless, in the case of large-scale problems, especially in three dimensions, maximizing the utility of monolithic solvers demands the availability of efficient preconditioners. These tests involve interactions between flows with finite Reynolds numbers and beam or shell structures, as well as solids with finite thickness. The feasibility and potential of a monolithic approach in a fluid-structure interaction are proved also in [8] where applications in bio-mechanics are analyzed, in which the focus is on the possibility of using different mesh for fluid, solid and interface.

In a FSI problem, the treatment of the interface between the two physical domains can be done in different ways. A conforming mesh method treats interface conditions as physical boundary conditions, considering them as part of the solution. Meshes must conform to the interface as the solid structure moves or deforms, re-meshing (or mesh-updating) becomes necessary as the solution progresses. A non-conforming mesh method imposes boundary conditions as constraints on the model equations, allowing independent solutions of fluid and solid equations on their respective grids without the need for re-meshing.

The interface is not the only place where focus the attention, a large problem is also the discretization of the entire problem. As the knowledge obtained in the use of finite element method (FEM) in solid domains [9], it was studied how to apply it to fluids by *Belytschko et Al* in [10] and [11] and then by *Donea et Al.* [12]. In these works, a Lagrangian approach has been used for the kinematic description and this causes some problems in coping with strong deformation that characterizes fluid flows. To solve it and obtain a unique mesh for the entire problem the Arbitrary Lagrange Eulerian (ALE) method can be used. It is a three domains method, in which in addition to the material and spatial domain there is a reference domain that can arbitrarily move. The movement of this reference domain is obtained by using a mesh moving algorithm, this allows to fix the problem related to the movement of the solid inside a fluid domain. *Noh* [13] was the first to design a code for calculating the solution of two space dimensional Eulerian hydrodynamics problems within regions, featuring arbitrary polygonal approximating meshes with moving fluid boundaries.Subsequently, many efforts have been made to improve the reliability of this method [14],[15],[16],[17] and it is widely used in various engineering and scientific applications as: aerospace, automotive, and biomechanics.

To avoid the computational effort of re-meshing an immersed method can be used. In this type of approach, a force-equivalent term is introduced into fluid equations to capture interactions without the need for mesh updates during numerical computation. This FSI force derives from the structural configuration and then is used to compute fluid velocity. The immersed boundary method can be used if the structure occupies a small fraction of volume, otherwise, if a bulk body is inside a large fluid domain, an immersed volume domain is used. The mathematical structure of the method was developed by *Peskin et Al.* [18], improvements and diversification have been made over the years [19] , [20] and an extensive review of all these methods was done by *Mittal and Al* [21] in 2005.

Different reviews on fluid-structure interaction can be found: *Hou et Al* (2012) [22] focused on partitioned approach-based conforming mesh methods and the immersed non-conforming mesh methods. *Morab and*

Sharma [23] explored different numerical methods testing their stability in several computational FSI application. In [24] special attention has been given to the immersed boundary and non conforming mesh methods.

The studies in literature referred to the different fields of application as aeroelasticity [25] in which a complex flow solvers is used and the structure is simplified. In biomechanics, *Crosetto et Al.* [26] focus on developing and verifying novel algorithms for accurately predicting hemodynamics in major arteries. These algorithms are based on finite element (FE) simulations, which model the interaction between blood flow and the deformation of arterial walls, particularly in a healthy aorta. Typically, Noise Vibration Harshness (NVH) problems are understood as the interaction between the vehicle's body and the fluid volume within its interior. In [27] an analysis of different methods used to solve the FE model has been carried out, methods taken into account are: Direct, Automated MultiLevel Substructuring (AMLS) and Lanczos, the parameters used to compare them were the mesh refinement and the frequency of analysis investigation. Other FSI applications are: Battery Thermal Management, [28], hearth valve function [29], particle motion in an incompressible fluid [30] and magneto-hydrodynamics [31].

In this paper, the mathematical formulation of FSI problems is summarised in section 2. Differences between the monolithic and partitioned approach is described in section 3. Section 4 presents FSI applications and finally conclusions are drawn.

2 Mathematical formulation

The domain of interest of a generic fluid-structure interaction is divided in two: the solid Ω_s and the fluid domain Ω_f, the site in which the interaction takes place is the structure surface Γ.

2.1 Fluid domain

The fluid motion is described by the Navier–Stokes equations, a system of non-linear partial differential equations valid only for the Newtonian fluids, in which the stress sensor is a linear function of strain. These equations explain the connection between the change in the momentum of an element of fluid and the forces applied to it. The conservation of momentum in a control volume is expressed in the form:

$$\rho \frac{Dv^f}{Dt} = \rho F + \nabla \sigma \tag{1}$$

Where $\rho(x,t)$ is the density, v(x,t) the velocity vector, F(x,t) is the body force vector (e.g. gravity) and $\boldsymbol{\sigma}$ the stress tensor of the fluid at a point described by vector x at time t. Recalling that the material derived of a general vector A is the sum of the derivative over time of A and the product between the velocity and the spatial gradient of vector A

$$\frac{DA}{Dt} = \frac{\partial A}{\partial t} + (v\,\nabla)A \tag{2}$$

The stress tensor is linked linearly with the rate of strain:

$$\sigma_{i,j} = -p\delta_{i,j} + \lambda \frac{\partial v_k}{\partial x_k}\delta_{i,j} + \mu\left(\frac{\partial v_k}{\partial x_j} + \frac{\partial v_j}{\partial x_k}\right) \tag{3}$$

Where λ is the second coefficient of viscosity and μ is the coefficient of viscosity of the fluid, they depend on the thermodynamic properties of the fluid, so they may vary in space and time. Generally, it is assumed that they are uniform in space and constant in time. In a control volume is also imposed the conservation of mass:

$$\frac{1}{\rho}\frac{D\rho}{Dt} + \nabla v^f = 0 \tag{4}$$

When the fluid is incompressible equation (4) changes in:

$$\nabla v^f = 0 \tag{5}$$

The last equation needed to study a fluid is the energy conservation:

$$\frac{\partial(\rho s)}{\partial t} + \nabla(\rho s v^f) = \frac{1}{T}(-\nabla q + \Phi + Q) \tag{6}$$

where: s is the entropy per unit mass, T is Temperature, Φ is is the viscous dissipation function, q is the local heat flux and Q a possible heat source. In many fluid-structure interaction problems the constitutive equations for the fluid are two: Stoke's expression relates the bulk viscosity λ with μ (7) and the Fourier heat conduction law (8)

$$\lambda + \frac{2}{3}\mu = 0 \tag{7}$$

$$q = -k\nabla T \tag{8}$$

The Navier-Stokes equations describe the motion of fluid, including air, that is essential for comprehending the generation and propagation of sound in a fluid medium. Acoustics focuses on the transport and propagation of minor perturbations, which can be significantly smaller than the background conditions [32], [33].In most practical cases, the acoustic problem can be assumed linear, as solving the complete governing equations is often impractical because it demands extremely high numerical precision. Since the purpose of this review is not to dwell on the study of small perturbation inside a stationary background mean field, the Helmholtz equation it is directly presented (9), it comes from the linearization of the formulae seen above using the general scalar wave equation and several simplifications. The mathematical problem is reduced to solving an equation with one dependent variable: the pressure.

$$-\frac{1}{\rho_0}\frac{\omega^2}{c_s^2}p_1 + \nabla[-\frac{1}{\rho_0}(\nabla p_1 - \mathbf{F}_1)] = -i\omega(\frac{1}{\rho_0}M_1 + \frac{\alpha_p}{(\rho_0 c_p)}Q_1) \tag{9}$$

With: ρ_0 density of background flow, c_s the isentropic speed of sound , p_1 the pressure of the perturbation, F_1 volume force source term, M_1 possible mass source term, Q_1 the heat source term, α_p the isobaric coefficient of thermal expansion, and, finally, c_p the fluid heat capacity at constant pressure.

2.2 Solid domain

Several different formulations for the structure dynamics concerning the fluid-structure interaction problems are present in the literature. In this review, the solid is usually described as elastic, homogeneous and isotropic, which interact with the fluid at the interface Γ. As described in [22] following the D'Alembert-Lagrange principle it is possible to obtain the solid equation of motion:

$$\rho\frac{Dv^s}{Dt} - \frac{\partial\sigma^s_{i,j}}{\partial x_j} + f^s_i = 0 \tag{10}$$

Since the displacement field is given in a Lagrangian description the material derivative is equal to the time derivative of the velocity, also the structural stress tensor $\sigma_{i.j}$ is related to strain $e_{i,j}$ with the linear Hook's law:

$$\sigma_{i.j} = \lambda\delta_{i,j}\epsilon_{l,l} + 2G\epsilon_{i,j} \tag{11}$$

with :

$$\lambda = \frac{E\nu}{(1+\nu)(1-2\nu)} \tag{12}$$

$$\epsilon_{i,j} = \frac{1}{2}(u_{i,j} + u_{j,i}) \tag{13}$$

$$G = \frac{E}{2(1+v)} \tag{14}$$

where E is the Young's modulus, ν the Poisson's ratio and λ the Lamé's costant. The fluid stresses the solid by applying its force, which causes a movement of the structure that depends on its characteristics described by the stiffness matrix; this displacement can cause a process of re-meshing according to the numerical method used. The force is composed of the first part, which describes the pressure of the fluid and another part that refers to the viscous forces (15).

$$F_i = -n(-p + \mu((\frac{\partial u_i^f}{\partial x_j})_i + (\frac{\partial u_i^f}{\partial x_j})_i^T) - \frac{2}{3}\mu\frac{\partial u_i^f}{\partial x_i}) \tag{15}$$

with n is the normal vector to the boundary and μ the viscosity of the fluid.

2.3 Coupling conditions at the interface

In order to complete the mathematical treatment of the problem, the coupling conditions at the interface Γ must be taken into account. The first possible condition is the no-slip at the interface [34], which is a kinematic condition on fluid and solid velocities:

$$v_i^s = v_i^f \tag{16}$$

This is a boundary condition that ensures that a viscous fluid has the same velocities as the solid at the interface. If the numerical method includes mesh updating, it is possible to use this condition, derive it in time, and use it for the interface profile. Another possible condition is the stress continuity constrain:

$$\sigma_{i,j}^s n_i = \sigma_{i,j}^f n_i \tag{17}$$

A Neumann condition which imposes equal the external force on the solid and the tension exerted by the fluid at the interface. The application of these conditions demands a meticulous approach, tailored to the specific test case under examination. For instance, in the scenario of fluid loading on a structure, the pressure and viscous forces in the fluid impact the structure, while the structure itself does not influence the fluid flow. Conversely, if the structural velocity serves as a boundary condition for the fluid, the fluid's influence on the structure is negligible. In the fully coupled case, both effects are considered, necessitating a precise understanding and application of the coupling conditions.

3 Partitioned and monolithic approach

3.1 Partitioned approach

Two primary approaches have been developed to accurately simulate the interactions between solid and fluid: the monolithic and partitioned methods. This chapter will explain the differences between the two approaches, both in terms of discretization and solution algorithm. These approaches differ mainly in how they solve fluid and structure governing equations. In the monolithic approach, the complete system of nonlinear equations, which derives from the coupled discretization of the equations seen above, is solved as a whole. In contrast, the partitioned approach solves the fluid and structure equations separately, exchanging information between the two domains at each time step. This method offers greater flexibility, allowing the use of specialized and optimized solvers for each domain. However, this separation can introduce challenges related to stability and accuracy, especially in cases of strong fluid-structure coupling.

The fluid usually applies a force (15) to the solid interface and the structure, if it is not considered rigid, can respond with a displacement, causing the change of the pressure field. It is intuitive that modelling these phenomena can be complicated, for this reason rather than create a single field analysis, it is possible to use separate analyzers for each domain and coupled it only at the interface. This is the work done by partitioned approaches; partitioning a large complex field is not a singularity of fluid-structure interaction problem but is also used in other areas of mechanics, such as dynamic substructuring.

A particular partitioned solution approach called staggered is composed of two phases: the solution phase and the predictor phase, in figure 1 it is shown a temporal flow diagram of the approach. The generic field X (fluid or solid) is solved at the generic time t, and the solution S is given as an input to solve the other field Y(solid or fluid), which is solved by a different analyzer that gives as output P, so called the predictor variable used to solve X at the next time $t + h$ where h is the time step. It is quite clear that the time step must be chosen wisely, as this could be one of the instability sources of the time integration procedure as the fluid and solid time constants are very different. This is a very easy example of how a partitioned approach works.

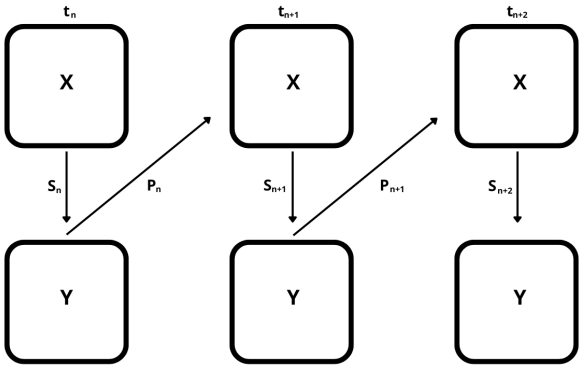

Figure 1: Staggered approach temporal flow diagram

3.2 Monolithic approach

A monolithic approach refers to a numerical technique in which the entire system of equations governing a physical problem is solved simultaneously in a single unified framework. In the context of fluid-structure interaction simulations, a monolithic approach involves solving the equations describing the fluid dynamics and structural mechanics together as a coupled system. This approach differs from partitioned, where the fluid and solid domains are described independently and then coupled through interface conditions.

Different types of monolithic approaches have been studied in the literature. However, this review focuses on the use of the finite element method and also considers the possibility of re-meshing at each time step. In figure 2 it is shown the temporal flow diagram of the monolithic approach. In the monolithic approach, the description of the whole system becomes complex, for the solid part, the natural view is the Lagrangian description instead, for the fluid is the spatial Eulerian, thus a possible solution for the FSI interaction problem is to use the Arbitrary Lagrangian-Eulerian (ALE) kinematic representation.

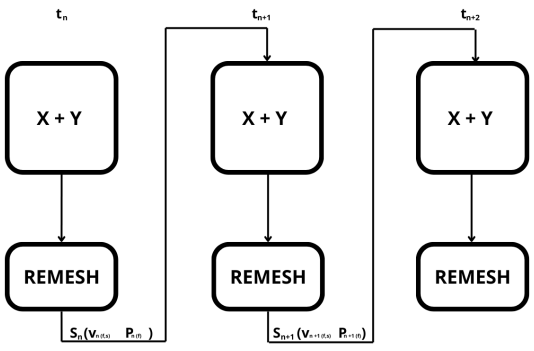

Figure 2: Monolithic approach temporal flow diagram

3.2.1 Arbitrary Lagrangian Eulerian description

The ALE description is based on a reference grid which has an arbitrary velocity $\mathbf{v}^z$ totally independent from both fluid and solid domains, if $\mathbf{v}^z = 0$ the method becomes Eulerian with the reference frame fixed in the space, instead if $\mathbf{v}^z = \mathbf{v}^f$ the problem is described in a Lagrangian point of view.

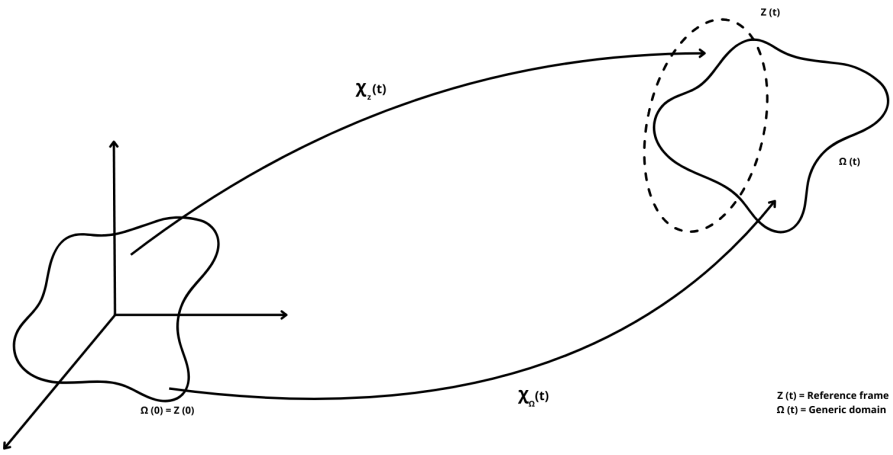

Figure 3: Initial and configuration at time t of a generic domain Ω and ALE computational mesh

In order to have a clear idea of how this method works, the figure 3 refers to only one domain (fluid or solid). Following the mathematical notation of *Hron and Turek* in [8], let Ω be the stress-free (or initial) configuration of a given body and Ω_t the configuration at time t; to describe the motion of the body a smooth function χ_Ω is used:

$$\chi_\Omega : \Omega \; x \; [0, T] \rightarrow \mathbb{R}^3 \tag{18}$$

The vector X describes coordinate of the material point at the initial configuration, thus the position of the same point at time t is given by

$$x = \chi_\Omega(X, t) \tag{19}$$

The mathematical quantities are described in a standard way: the displacement field, the velocity field, the deformation gradient and its determinant are:

$$u(X, t) = \chi_\Omega(X, t) - X \qquad v = \frac{\partial \chi}{\partial t} \qquad D = \frac{\partial \chi}{\partial X} \qquad J = detD \tag{20}$$

The principle behind the ALE description is to use as a computational mesh a region $\boldsymbol{Z}$, which moves independently from body and this requires a new mathematical formulation. The function which describe the region $\boldsymbol{Z}$ law of motion:

$$\chi_z : \boldsymbol{Z} \; x \; [0, T] \rightarrow \mathbb{R}^3 \tag{21}$$

The corresponding velocity, deformation gradient and its determinant are:

$$v^z = \frac{\partial \chi_z}{\partial t} \qquad D_z = \frac{\partial \chi_z}{\partial X} \qquad J_z = detD_z \tag{22}$$

The ALE description means mapping the initial configuration of the continuum into the current configuration of the reference frame. The Jacobian provides a link between the coordinates in the reference frame and material coordinates. Using this type of description, the equation of motion must be modified to "translate' the law, previously expressed in spatial (Eulerian or Lagrangian) variables, into an equivalent one expressed in mixed variables. To have a clear idea on how the equations change refer to: [8], [12], [14], [15].

To illustrate the coupling procedure along fluid-solid interfaces in an ALE formulation, let consider a structural member in contact with a fluid which is allowed to slide along one face of the structure.

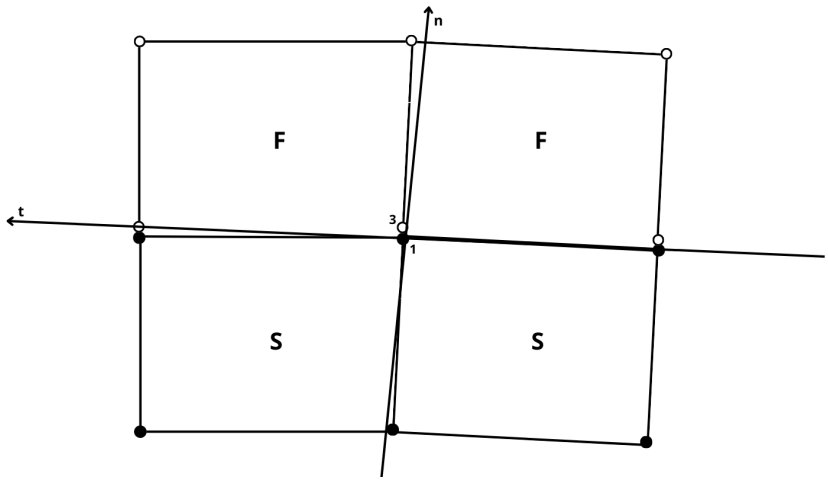

Figure 4: Nodes at fluid structure interface ALE method

Figure 4 presents a simplified problem, since the movement of the fluid mesh is completely independent of the movement of the fluid itself, it is possible to constrain the fluid nodes to remain contiguous to the structural nodes so that all nodes on the interface remain permanently aligned. It is evident that a permanent alignment of nodes along the interface significantly enhances the flow of information. The conditions to impose at each point of the fluid solid interface are three:

- the grid velocity of the fluid v^z must be equal to the velocity of the solid v^s,
- the normal velocity of the fluid v_n^f coincides with the normal velocity of the solid v_n^s,
- the tangential velocity of the fluid v_t^f is unconstrained.

Referring to the numbering in Figure 4, the condition on the solid and grid velocity means $v_3^z = v_1^s$. This condition will be used in each time step as a boundary condition for an automatic fluid mesh displacement algorithm. By imposing the other two conditions on the fluid and structure velocity and following the mathematical formulation described in [12] it is possible to find a relation between the acceleration of the nodes at the interface and the internal nodal forces. In conclusion, the problem of fluid-structure coupling is simplified, as fluid nodes on the structure must stay attached to it throughout the calculation, leaving unconstrained the fluid's tangential velocity.

3.2.2 Solution strategy of the system of equations

The entire system of equations has to be solved to obtain the coupling solution. Obviously, this partial differential equations system must be discretized in time and space. Time discretization can be done using explicit methods that are simple and efficient for non-stiff problems with small time steps. Implicit and semi-implicit methods are better suited for stiff problems where stability is a concern, like the fluid structure interactions. The space discretization is carried out using the finite element method, the same element is used for both fluid and solid. The result of the discretization is a non linear system of equations, let write the system as:

$$\boldsymbol{F}(\boldsymbol{X}) = 0 \tag{23}$$

Where $\boldsymbol{X} = (v_i^{s\ n+1}, v_i^{f\ n+1}, p_i^{n+1})$ is the solution vector to find in each time step. As described by *Hron and Turek* in [8], the system (23) can be solved by using the damped Newton method with line search.

Instead, in order to avoid the evaluation of the Jacobian matrix *Hubner et Al.* in [6], decide to use a Picard iteration, leading to a fixed point time solution procedure. The generic i th iteration step of the unknowns $\boldsymbol{x}_n$ at time n is:

$$\boldsymbol{A}(\boldsymbol{x}_n^{i-1})\ \boldsymbol{x}_n^i = \boldsymbol{b}(\boldsymbol{x}_{n-1}^i, \boldsymbol{x}_n^{i-1}) \tag{24}$$

The components of the unknown vector are structural displacements, convective velocities and fluid mesh coordinates. At time n the solution from the end of the previous time $n-1$ is used as the estimate for the velocity field in the current time slab. For structural displacements and fluid mesh coordinates, a linear extrapolation is performed. The linearized model equations for the fluid, structure, and coupling conditions are then solved simultaneously as a single system of equations, excluding the fluid pressure variables.

4 FSI application cases

The last section of the papaer focuses on two different application cases. The first case demonstrates how fluid flow can deform structures and how to solve for the flow in a continuously changing geometry using the arbitrary Lagrangian-Eulerian (ALE) method. This first example is used to test the monolithic and partitioned approaches on a simple application case. The second example shows the interaction of sound with a structure by providing the case of a cylinder in water, the structure is considered both rigid and elastic. Models come from the COMSOL Multiphysics library and are well-known and studied in the literature [35].

4.1 Obstacle in a horizontal flow channel

The model geometry consists of a horizontal flow channel with an obstacle and a narrow vertical structure in the middle. The fluid flows from left to right, but the obstacle forces it into a narrow path in the upper part of the channel. Figure 5 shows the model geometry at a certain point during the simulation.

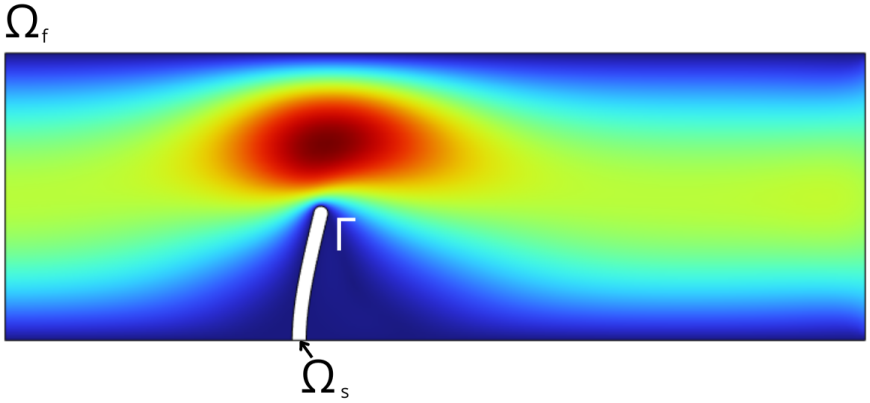

Figure 5: Model geometry

The fluid flow exerts a force on the structure's walls due to viscous drag and fluid pressure. Since the structure is made of a deformable material, it bends under the applied load. Consequently, the fluid flow adjusts to a new path, making incorrect solutions based on the original geometry. The ALE method manages the dynamics of the deforming geometry and moving boundaries with a moving grid. At the entrance of the channel on the left, the flow has fully developed laminar characteristics with a parabolic velocity profile, though its amplitude varies over time. Initially, the flow increases rapidly, peaking at $0.215\,s$, then gradually decreases to a steady-state value of $5\,e(-02)\,m/s$. The geometric properties of the channel and the fluid are summarised in table 1.

The structural mechanics follow the equation described in section 2; the structure is loaded at the boundary with the force 15, and this causes a large deformation, as figure 5 shows. The Navier-Stokes equations are solved in the reference frame described by the ALE method; this allows the fluid's moving mesh to follow the structure's deformation. In Figure 6 is shown the equivalent von Mises stress at the end of the simulation ($t = 4s$).

Table 1: Domain data

Quantity	Value	Description
L_f	300 μm	$Length\ of\ the\ flow\ channel$
H_f	100 μm	$Heigth\ of\ the\ flow\ channel$
L_s	5 μm	$Length\ of\ the\ structure$
H_s	50 μm	$Heigth\ of\ the\ structure$
ρ_f	1000 kg/m^3	$Fluid\ density$
μ	0.001 $P_a\,s$	$Fluid\ dynamic\ viscosity$
ρ_s	7850 kg/m^3	$Structure\ density$
E	2000 kPa	$Structure\ Young's\ module$

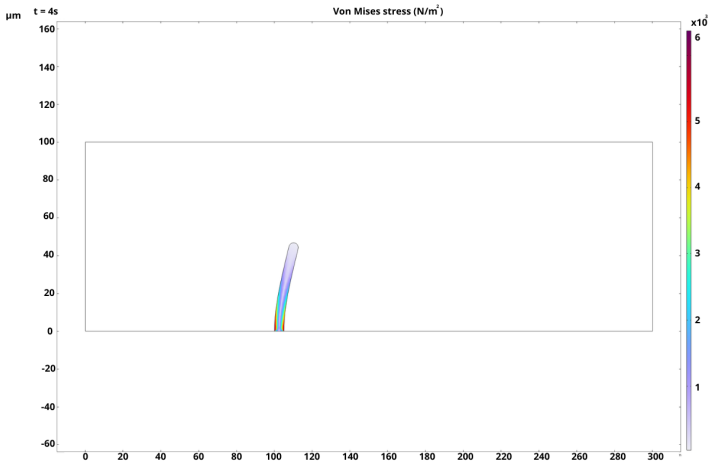

Figure 6: Von Mises stress

Other interesting considerations about the mesh movement and the optimal shape of the body are presented in [35]. In this case, the focus is on the approaches to solving the problem. The results and calculation time are compared using both monolithic and partitioned approaches. For the monolithic approach, the Jacobian updating has been modified in various ways that are allowed by COMSOL Multiphysics: Update minimum (J1), update once per time step (J2), and update every iteration (J3). The table 2 below displays the computational time for each approach and the standard deviation of the differences between the structural stress of each monolithic and the result of the partitioned approach.

Table 2: Differences between partitioned and monolithic approaches

	Partitioned	J1	J2	J3
$computational\ time\ t_c$	79 s	44 s	47 s	86 s
$standard\ deviation\ \sigma$	//	2.14 $e(-0.4)$	0.0016	0.0016

These results show a major difference in the computational time between the use of partitioned and monolithic, at least until the update of the Jacobian does not happen in each iteration. The results suggest the efficiency of the monolithic solver, as well as great precision in the results compared with the partitioned approach. However, the case under examination is a very simple case, the total number of degrees of freedom are 19000, for future can be interesting test the different solver for larger case. A possible parameter to take into account to decide which type of solver to use can be the difference between the time constant of the two domains. The convective and diffusion constants for the fluid and the time constant for the mechanical part are described in the equation:

$$\tau_{conv} = \frac{L}{U} \qquad \tau_{diff} = \frac{L}{D^2} \qquad \tau_s = \frac{1}{\omega_n} \tag{25}$$

Where L is the characteristic length of the problem, U characteristic speed of fluids, D term thermal or mass diffusivity and ω_n the structure natural frequency of interest.It would be interesting in future work to determine which approach to use based on how the time constants of the systems differ. Choosing a monolithic approach when the time constants are close and partitioned if the time constants differ significantly.

4.2 Acoustic structure interaction

In this case, the interaction is between the cylinder and the sound propagating through the water, the structure responds, and this induces new sound waves in the air; it is a classic example of two-way coupling between the acoustic medium and the cylinder. The cylinder is immersed in a water domain with an incident acoustic wave inside. The walls are stressed by the acoustic pressure. The model determines the frequency response of the solid and then sends this data back to the acoustics domain to analyze the resulting wave pattern. The Acoustic–Solid Interaction, Frequency Domain multiphysics interface is used to model this phenomenon properly. The structure is considered as linear and elastic, the material used is from the COMSOL database Aluminum 3003-H18; instead, for modelling the pressure acoustic of waves inside the water domain, the Helmholtz equation (9) is employed, with $M_1 = Q_1 = F_1 = 0$ and the following acoustic data are used:

Table 3: Acoustic domain data

Quantity	Value	Description
ρ_c	997 kg/m^3	$Density$
c_c	1500 m/s	$speed\ of\ sound$
$f = \omega/2\pi$	60000 Hz	$Wave\ frequency$

The cylinder is 2 cm in height and has a diameter of 1 cm. The water acoustic domain is considered as a sphere with a reasonably large diameter. Figure 7 shows the acoustic and structure domain.

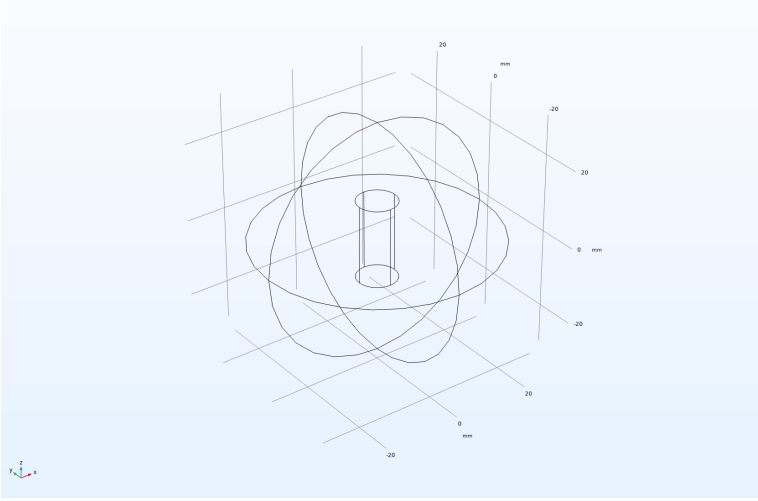

Figure 7: Acoustic and structure domain

To represent an incoming sound wave, an incident plane wave is specified at the boundary of the sphere, in table 4 the radiation boundary conditions are specified:

The force that stresses the cylinder at the boundary is described by (15) without the part related to the viscous forces, with p the acoustic pressure. At the same time, on the fluid, the normal acceleration is defined equal to the normal acceleration of the solid. The case of a rigid body has been studied to understand how strict

Table 4: Radiation boundary condition

Quantity	Value	Description
θ	$4\pi/6\ rad$	$Wave\ direction\ angle$
ϕ	$-\pi/6\ rad$	$Wave\ direction\ angle$
k	$(sin\theta\ cos\phi),\ (sin\theta\ sin\phi),\ (cos\theta)$	$Incident\ wave\ direction\ vector$
p_1	$1\ Pa$	$Pressure\ amplitude$

is the bidirectional coupling. Figure 8 compares the hard-wall example and the linear elastic solid model. Near the cylinder wall, the plot indicates that the sound pressure level (SPL) is higher on the upstream side for the hard-wall case than the aluminum model. This is also true on the downstream side, where the SPL is higher for the hard-wall model than for the aluminum model. This demonstrates that the hard wall reflects more sound than the linear elastic model.

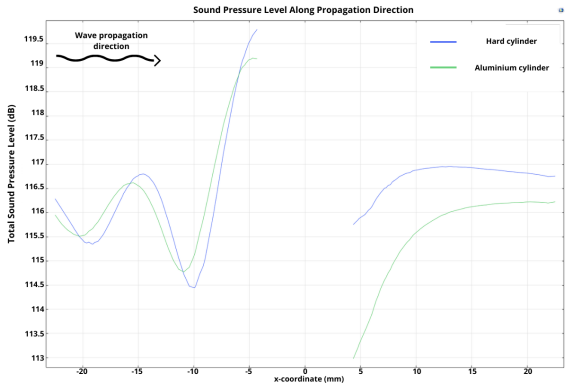

Figure 8: Sound pressure level near the cylinder, rigid and deformable structure

Figure 9 displays the structural response and the sound pressure of the acoustic waves. Even if the values of the deformation are minimal, the acceleration is large enough to affect the sound waves. To represent acceleration a visual plot is used with the help of arrows since their length is proportional to the surface acceleration.

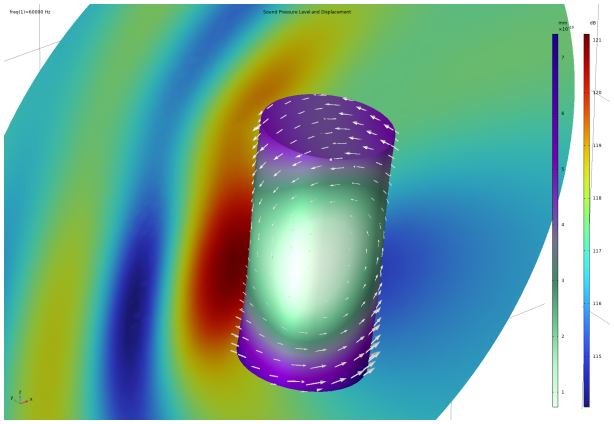

Figure 9: Structural response and sound-pressure plot (dB) of the acoustic waves.

5 Conclusions

In recent decades, many numerical methods have been developed for simulating FSI. This review is intended to be a starting point for the study of the fluid-structure interaction problem to understand how the commer-

cial software works behind the graphic interface. The first chapter presents a general discussion about the differences between numerical methods, while the domain equations are explained in chapter two. The third chapter focuses on the solution procedure of the two main approaches to the problem: partitioned and monolithic. Finally, practical examples of fluid-structure interaction have been presented in the fourth chapter; the first example highlights the differences between the monolithic and partitioned approaches. The second one shows the results of an acoustic structure interaction, considering the rigid and elastic structure. Given the improved computing power and reduced CPU time requirements from FSI solvers, it would be interesting to imagine how to apply fluid-structure interaction analysis to scientific fields that have not yet been explored from this multiphysics point of view, such as the NVH.

Acknowledgements

This publication is part of the project PNRR-NGEU which has received funding from the MUR – DM 117/2023

References

[1] Hessenthalera, Balmusb, Röhrlea, Davd, and Nordslettenc, "A class of analytic solutions for verification and convergence analysis of linear and nonlinear fluid-structure interaction algorithms," *Comput. Methods Appl. Mech. Engrg.*, vol. 362, pp. 1–24, 2020.

[2] C. Felippa and K. Park, "Staggered transient analysis procedures for coupled mechanical systems: Formulation,," *Computer Methods in Applied Mechanics and Engineering,*, vol. 24, pp. 61–111, 1980.

[3] C. Felippa, K. Park, and J. DeRuntz, "Stabilization of staggered solution procedures for fluid–structure interaction analysis," *Computational Methods for Fluid-Structure Interactions Problems*, vol. 26, 1977.

[4] P. L. Tallec, J.-F. Gerbeau, P. Hauret, and M. Vidrascu, "Fluid structure interaction problems in large deformation,," *Comptes Rendus Mécanique,*, vol. 33, pp. 910–922, 2005.

[5] A. Galantai, "The theory of newton's method," *Journal of Computational and Applied Mathematics*, vol. 124, pp. 25–44, 2000.

[6] B. Hubner, E. W., and D. D., "A monolithic approach to fluid–structure interaction using space–time finite elements," *Comput. Methods Applied Mechanics Engineering*, vol. 193, pp. 2087–2104, 2004.

[7] M. Heil, A. Hazel, and J. Boyle, "Solvers for large-displacement fluid–structure interaction problems: segregated versus monolithic approaches," *Computational Mechanics*, vol. 43, pp. 91–101, 2008.

[8] J. Hron and S. Turek, "A monolithic fem/multigrid solver for ale formulation of fluid-structure interaction with application in biomechanics," 2007.

[9] R. H. Macneal, *Finite Elements: Their Design and Performance*. 270, Madison Avenue, New York: DEKKER, 1994.

[10] T. Belytschko and J. M. Kennedy, "Computer models for subassembly simulation," *Nuclear Engineering and Design*, vol. 49, pp. 17–38, 1978.

[11] T. Belytschko, J. M. Kennedy, and D. F. Schoeberle, "Quasi-eulerian finite element formulation for fluid-structure interaction," *Journal of Pressure Vessel Technology-transactions of The Asme*, vol. 102, pp. 62–69, 1980.

[12] J. Donea, S. Giuliani, and J. Halleux, "An arbitrary lagrangian-eulerian finite element method for transient dynamic fluid-structure interactions," *Computer Methods in Applied Mechanics and Engineering*, vol. 33, no. 1, pp. 689–723, 1982.

[13] W. F. Noh, *CEL: A Time-Dependent, Two-Space-Dimensional, Coupled Eulerian-Lagrange Code*. Livermore, California: Lawrence Radiation Laboratory, 1963.

[14] P. Kjellgren and J. Hyvärinen, "An arbitrary lagrangian-eulerian finite element method," *Computational Mechanics*, vol. 21, no. 1, pp. 689–723, 1998.

[15] M. Souli, A. Ouahsine, and L. Lewin, "Ale formulation for fluid–structure interaction problems," *Computer Methods in Applied Mechanics and Engineering*, vol. 190, no. 5-7, pp. 659–675, 2000.

[16] T. Belytschko and J. M. Kennedy, "Quasi-eulerian finite element formulation for fluid-structure interaction," *nuclear Engineering and Design,*, vol. 68, pp. 129–146, 1982.

[17] C. Hirt, A. Amsden, and J. Cook, "Ale formulation for fluid–structure interaction problems," *Journal of Computational Physics*, vol. 14, no. 3, pp. 227–253, 1974.

[18] C. S. Peskin, "The immersed boundary method," *Acta Numerica*, vol. 11, p. 479–517, 2002.

[19] J. Sethian and A. Wiegmann, "Structural boundary design via level set and immersed interface methods," *Journal of Computational Physics*, vol. 163, no. 2, pp. 489–528, 2000.

[20] J. Kim, D. Kim, and H. Choi, "An immersed-boundary finite-volume method for simulations of flow in complex geometries," *Journal of Computational Physics*, vol. 171, no. 1, pp. 132–150, 2001.

[21] R. Mittal and G. Iaccarino, "Immersed boundary methods," *Annual Review of Fluid Mechanics*, vol. 37, pp. 239–261, 2005.

[22] W. J. Hou G and L. A., "Numerical methods for fluid-structure interaction," *Communications in Computational Physics 2012;12(2):337-377.*, pp. 337–377, 2012.

[23] S. R. Morab and A. Sharma, "An overview of computational fluid structure interaction: Methods and applications," 2020.

[24] O. Zienkiewicz, R. Taylor, and P. Nithiarasu, "Chapter 13 - fluid–structure interaction, the finite element method for fluid dynamic," pp. 423–449, 2014.

[25] Kamakoti and Shyy, "Fluid–structure interaction for aeroelastic applications," *Progress in Aerospace Sciences*, vol. 40, no. 8, pp. 535–558, 2004.

[26] Crosetto, Reymon, Deparis, Kontaxakis, Stergiopulos, and Quarteroni, "Fluid–structure interaction simulation of aortic blood flow," *Computers Fluids*, vol. 43, no. 1, pp. 46–57, 2011.

[27] Kropp and Heiserer, "Efficient broadband vibro-acoustic analysis of passenger car bodies using an fe-based component mode synthesis approach," *Journal of Computational Acoustics*, vol. 11, no. 2, pp. 139–157, 2003.

[28] Y. Liao, H. Xu, S. X. Huang, Taishuo, and Z. Hu, "Modeling of fluid-structure interaction to enhance battery thermal management in electric vehicles," pp. 14–29, 2023.

[29] W. Sun, W. Mao, and B. E. Griffith, "Computer modeling and simulation of heart valve function and intervention," *Principles of Heart Valve Engineering*, pp. 177–211, 2019.

[30] S. Koshizuka and Y. Oka, "Moving-particle semi-implicit method for fragmentation of incompressible fluid," *Nuclear Science and Engineering*, vol. 123, no. 3, pp. 421–434, 1996.

[31] B. MÜCK, C. GÜNTHER, U. MÜLLER, and L. BÜHLER, "Three-dimensional mhd flows in rectangular ducts with internal obstacles," *Journal of Fluid Mechanics*, vol. 418, pp. 265–295, 2000.

[32] D. T. Blackstock, *Fundamentals of Physical Acoustics*. Wiley-Interscience, 2000.

[33] A. D. Pierce, *Acoustic, An Introduction to Its Physical Principles and Applications*. Springer Cham, 2019.

[34] T. Kruger, H. Kusumaatmaja, A. Kuzmin, O. Shardt, G. Silva, and E. M. Viggen, "Boundary conditions for fluid-structure interaction," in *The Lattice Boltzmann Method: Principles and Practice*. Springer International Publishing Cham, 2017, pp. 433–491.

[35] K. S. Rao, K. G. Sravani, G. Yugandhar, G. V. Rao, and V. Mani, "Design and analysis of fluid structure interaction in a horizontal micro channel," *Procedia Materials Science*, vol. 10, pp. 768–788, 2015.