On the role of (weak) compressibility for fluid-structure interaction solvers

Andrea La Spina | Christiane Förster | Martin Kronbichler | Wolfgang A. Wall

Institute for Computational Mechanics,
Technical University of Munich,
Garching, Germany

Correspondence
Andrea La Spina, Institute for
Computational Mechanics, Technische
Universität München, Boltzmannstraße
15, 85748 Garching, Germany.
Email: laspina@lnm.mw.tum.de

Funding information
European Education, Audiovisual and
Culture Executive Agency (EACEA)
under the Erasmus Mundus Joint
Doctorate Simulation in Engineering and
Entrepreneurship Development (SEED),
Grant/Award Number: FPA 2013-0043

Summary
In the present study, a weakly compressible formulation of the Navier-Stokes equations is developed and examined for the solution of fluid-structure interaction (FSI) problems. Newtonian viscous fluids under isothermal conditions are considered, and the Murnaghan-Tait equation of state is employed for the evaluation of mass density changes with pressure. A pressure-based approach is adopted to handle the low Mach number regime, i.e., the pressure is chosen as primary variable, and the divergence-free condition of the velocity field for incompressible flows is replaced by the continuity equation for compressible flows. The approach is then embedded into a partitioned FSI solver based on a Dirichlet-Neumann coupling scheme. It is analytically demonstrated how this formulation alleviates the constraints of the instability condition of the artificial added mass effect, due to the reduction of the maximal eigenvalue of the so-called added mass operator. The numerical performance is examined on a selection of benchmark problems. In comparison to a fully incompressible solver, a significant reduction of the coupling iterations and the computational time and a notable increase in the relaxation parameter evaluated according to Aitken’s $\Delta^2$ method are observed.

KEYWORDS
artificial added mass effect, Dirichlet-Neumann partitioning, finite element method, fluid-structure interaction, weakly compressible flow

1 | INTRODUCTION

The interaction of fluid flow with deformable structures is a classical problem in engineering science. The solution of fluid-structure interaction (FSI) problems is required in different fields such as aerospace, civil, and biomedical engineering. The characteristics of the fluid and the structure involved in this type of problems can vary a lot: in aerodynamics, the coupling may involve a light compressible fluid and a stiff structure like an aircraft wing, or it may involve a light incompressible fluid and an extremely flexible structure such as a parachute. In hemodynamics on the other hand, incompressible fluids and flexible structures with similar densities are usually coupled. Considering the variety of problems encountered when dealing with FSI, it is obvious that there cannot be a unique solver that fits all the needs. For the sake of simplicity, with the expression “compressible” or “incompressible” fluids, we refer to fluids that behave compressible or incompressible in the regime of interest.

The magnitude of the compressibility of the fluid plays an important role on the choice of the FSI solver. Compressibility effects arise from the variation of density, which in general depends on pressure and temperature. The influence
of compressibility on a flow field is also described by the Mach number, which is defined as the ratio of the fluid velocity and the speed of sound. The incompressible limit of a compressible flow is approached when the Mach number tends to zero. From a physical point of view, under these circumstances, the speed of sound is much larger than the fluid velocity, and fast pressure waves lead to a rapid pressure equalization. As a consequence the fluid field becomes incompressible, since no density variations due to compression can take place.

In this contribution, we focus on what are classically called incompressible flows, and we are interested in the interaction of such flows with flexible and mostly light structures. Such scenarios pose particular challenges to the development of efficient and robust coupled solvers. Among others, this can easily be seen from the fact that the so-called loosely coupled or staggered solution schemes work well for many aeroelastic problems (see, for example, the work of Farhat) but do not work at all for many examples of this category. The main reason for this is the so-called artificial added mass effect. The present analysis reveals that this effect would also be present when a compressible FSI solver is driven to the incompressible limit (but is usually not observed because the time step restrictions of the compressible flow solvers close to the incompressible limit make it very unlikely for such instabilities to show up in a real simulation). In addition, there are good reasons why incompressible flow solvers have to be and are used for low Mach number flows, as usually compressible flow solvers driven to the incompressible limit—and hence also FSI solvers based on them—are not the most efficient or appropriate approaches. While the use of incompressible flow solvers does not represent a problem for pure flow simulations, it adds additional challenges when using coupling schemes. In addition, in many cases, one does not even need to model fully incompressible flows, as this might also not be the most appropriate physical model but just a common approximation. Hence, in some situations, one has to pay a price for using a model that is not even the most appropriate one. This can be due to overall weak compressibility effects in some cases or local compressibility effects when strong pressure differences appear locally, eg, in scenarios of fluid-structure-fracture interaction. In such cases, including weak compressibility in the fluid formulation could both be a better physical approximation and might also be beneficial with regards to the coupled solver performance. This is why we propose such a formulation in this paper and study its influence on widely used coupling schemes.

In the last decades, other researchers also used (artificial) compressibility to solve FSI problems. One example that made this necessity especially evident was the analysis of an incompressible fluid enclosed in an elastic container with inlets only. Considering the walls at rest at the beginning of the simulation, the fluid is subjected to zero velocity boundary conditions at the interface, and because of its inability to compress, the continuity equation cannot be satisfied since there is a net flux into the container. One way to solve this would be to introduce a constraint equation that satisfies fluid mass conservation in the structural solution step, such as that proposed by Kättler et al. In the work of Van Opstal et al, it is shown that the Lagrange multiplier associated with the constraint equation can be interpreted as an excess pressure in the enclosed fluid volume, which serves to adapt the liquid volume to the inflows and outflows and the deformation. Another approach to solve this issue is to use compressibility, at least during the coupling iterations, as presented by Riemslagh et al, where a compressibility term is added only in the continuity equation of the boundary nodes at the flexible structure. In another case, presented by Råback et al, the method of artificial compressibility is employed, which allows the fluid to imitate the elastic response of the structure. The compressibility is continuously updated using the current pressure and its effects vanish as soon as the coupled iteration converges. The automated scheme proposed by Råback et al requires a user defined functional behavior to consider a spatially varying artificial compressibility, and although it may be useful for specific applications, it is not suitable for general problems of engineering interest. In a later work, the artificial compressibility is combined with a quasi-Newton method to account for the missing interface sensitivities when solving strongly coupled, fully/quasi-enclosed FSI problems. However, although the artificial compressibility has a physical basis, it violates the general continuity equation of compressible flows because it neglects the space derivatives of density. Moreover, it is not clear whether the artificial compressibility method works robustly independently of the examples.

The numerical approaches to handle low Mach number flows can be divided into two categories: density-based and pressure-based schemes. The methods developed for compressible flows usually use density as primary variable and extract pressure from an equation of state. These schemes cannot be used for incompressible or low Mach number flows because, under these circumstances, the density variations are small, and the pressure-density coupling becomes very weak. In this contribution, a pressure-based scheme is adopted, and therefore, the pressure is chosen as the main independent variable. This method is more robust and flexible compared to the density-based scheme because the pressure changes in the fluid domain are always finite, irrespective of the flow Mach number. The main idea has been introduced by Karki and Patankar, where a compressible extension of the SIMPLE method, originally developed for incompressible flows, has been proposed. According to this approach, the divergence-free constraint for incompressible flows is replaced...
by the continuity equation of compressible flows. The density is then rewritten in terms of the pressure via the equation of state to obtain a pressure correction equation. In the case of a temperature-dependent density (not addressed in this paper), an additional conservation equation for energy has to be taken into account within the iteration procedure.17

With regards to the coupling of fluids with deformable structures, two classes of solution procedures can be formulated. Monolithic schemes18,19 simultaneously solve the fluid and the structure equations within one global system. This approach is typically more robust and stable but requires an ad hoc implementation, in the sense that already existing and well-established codes for the single fields cannot be directly employed, and the introduction of efficient preconditioners to make it accessible to large-scale computations. Partitioned schemes20,21 instead require a sequence of solutions for each field and the exchange of information between the fluid and the structure. Such approaches are easier to implement and ease the use of existing codes for the solution of each field. On the other hand, one must deal with stability or more pronounced convergence issues when using a partitioned approach. In this contribution, we build upon the predominant partitioned approach for FSI solvers, the so-called Dirichlet-Neumann scheme, which has been extensively investigated in the literature. In this paper, the effects of the weakly compressible formulation for FSI problems will be explored with regards to partitioned schemes in the sense of iterative fixed-point methods and two relaxation techniques: the case of a fixed relaxation parameter for all time steps and the case of a dynamic relaxation parameter based on Aitken’s Δ2 method.20,22 In addition, we will also have a brief look into staggered partitioned schemes. As is well known and mentioned above, staggered partitioned approaches are affected by instabilities due to the so-called artificial added mass effect. This phenomenon was carefully analyzed by Causin et al3 and Förster et al4 in the context of incompressible flows interacting with flexible structures, and this instability has been shown to be relaxed for compressible fluids by van Brummelen.5 While the analysis of van Brummelen5 is based on the particular setting of a two-dimensional (2D) panel-model problem, in this contribution, the derivation is more general and not constraint by any specific geometrical feature. Moreover, the theoretical results are here supported by numerical tests on some popular FSI benchmarks, and the advantages of the weakly compressible formulation for the solution of the coupled problem is also shown in terms of computational time.

The present article is organized as follows. In Section 2, the governing equations for weakly compressible flows and flexible structures according to nonlinear elastodynamics, as well as the coupling conditions and their discretization, are expounded. In Section 3, we recall the main steps of the Dirichlet-Neumann coupling scheme. To ease the analysis and insight, a novel artificial added mass operator is derived in Section 4 in the context of weakly compressible flows. We present the numerical results of the presented approach for some benchmark test problems in Section 5. Finally, in Section 6, the main conclusions are drawn.

2 | PROBLEM DEFINITION

In this section, we introduce the governing equations for the fluid, the structure and the mesh field in the arbitrary Lagrangian–Eulerian (ALE) framework. In the following, the superscripts (·)F, (·)S, and (·)G denote the fluid field, the structure field, and the fluid grid, respectively.

2.1 | Fluid field

The fluid field is considered to be governed by the unsteady, weakly compressible Navier-Stokes equations for a Newtonian fluid under isothermal conditions. The conservation equations for mass and momentum can be written as

\[
\begin{align*}
\frac{\partial \rho^F}{\partial t} + \nabla \cdot (\rho^F \mathbf{u}^F) &= 0 \quad \text{in} \quad \Omega^F \times (0, T), \\
\frac{\partial \rho^F \mathbf{u}^F}{\partial t} + \nabla \cdot (\rho^F \mathbf{u}^F \otimes \mathbf{u}^F) - 2\mu^F \nabla \cdot \left( \epsilon(\mathbf{u}^F) - \frac{1}{3}(\nabla \cdot \mathbf{u}^F) \mathbf{I} \right) + \nabla p^F &= \rho^F \mathbf{b}^F \quad \text{in} \quad \Omega^F \times (0, T),
\end{align*}
\]

where the fluid velocity \( \mathbf{u}^F \) and the pressure \( p^F \) are the unknowns of the fluid problem. Here, \( \mathbf{b}^F \) denotes a prescribed body force, \( \epsilon(\mathbf{u}^F) = \frac{1}{2} \left( \nabla \mathbf{u}^F + (\nabla \mathbf{u}^F)^T \right) \) is the strain rate tensor of the Newtonian fluid, and \( \mathbf{I} \) is the second-order identity tensor. The dynamic viscosity \( \mu^F \) is assumed to be constant. For relating the fluid density \( \rho^F \) to the pressure, we use the
well-established Murnaghan-Tait equation of state\textsuperscript{23}

\[
\rho^F = \rho^F_0 \left( n^F \frac{p^F - p^F_0}{K^F_0} + 1 \right)^{\frac{1}{n^F}},
\]  

(3)

where \( \rho^F_0 \) and \( p^F_0 \) denote the reference values of density and pressure, respectively; \( K^F_0 \) is defined as the reference bulk modulus; and \( n^F \) is a material parameter dependent on the specific fluid considered. Strictly speaking, Equation (3) only applies to isentropic changes, but it can be applied in general with reasonable accuracy since \( n^F \) is independent of entropy and \( K^F_0 \) and \( \rho^F_0 \) are only slowly varying functions of entropy.\textsuperscript{24}

The deformation of the fluid domain \( \Omega^F \) is defined by a unique, arbitrary mapping \( \varphi \) given by

\[
d^G(x, t) = \varphi \left( d^G_{\Gamma}, x, t \right) \text{ in } \Omega^F \times (0, T),
\]  

(4)

where \( d^G_{\Gamma} \) represents the mesh interface displacement. The grid velocity \( u^G \) is then defined by

\[
u^G(x, t) = \frac{\partial d^G(x, t)}{\partial t} \text{ in } \Omega^F \times (0, T).
\]  

(5)

Equation (5) allows for the definition of the ALE convective velocity \( c = u^F - u^G \) representing the fluid velocity \( u^F \) relative to the arbitrarily moving fluid domain. The momentum Equation (2) can be rewritten in convective form according to the ALE formulation, assuming mass conservation (1) being exactly fulfilled:

\[
\rho^F \frac{\partial u^F}{\partial t} + \rho^F c \cdot \nabla u^F - 2 \mu^F \nabla \cdot \epsilon^F(u^F) + \nabla p^F = \rho^F b^F \text{ in } \Omega^F \times (0, T),
\]  

(6)

where \( \epsilon^F(u^F) = \epsilon(u^F) - \frac{1}{3} (\nabla \cdot u^F) I \) denotes the augmented strain rate tensor. Using the equation of state (3), mass conservation (1) may be reformulated as

\[
\nabla \cdot u^F = - \frac{1}{K^F_0 + n^F \left( p^F - p^F_0 \right)} \left( \frac{\partial p^F}{\partial t} + c \cdot \nabla p^F \right) \text{ in } \Omega^F \times (0, T).
\]  

(7)

The partial differential equations are subject to the boundary conditions

\[
u^F = \bar{u}^F \text{ on } \Gamma^F_{D_u} \times (0, T),
\]  

(8)

\[
p^F = \bar{p}^F \text{ on } \Gamma^F_{D_p} \times (0, T),
\]  

(9)

\[
\sigma^F \cdot n^F = \bar{h}^F \text{ on } \Gamma^F_N \times (0, T),
\]  

(10)

where \( \Gamma^F_{D_u} \) and \( \Gamma^F_N \) denote the Dirichlet and the Neumann partitions of the fluid boundary, respectively; \( n^F \) represents the outward-pointing normal vector; and \( \bar{h}^F \) is the boundary traction. The Cauchy stress tensor \( \sigma^F \) for the weakly compressible Newtonian fluid is given by

\[
\sigma^F = -p^F I + 2\mu^F \epsilon^F(u^F).
\]  

(11)

To complete the fluid problem, the initial conditions

\[
u^F = u^F_{(0)} \text{ in } \Omega^F \text{ at } t = 0,
\]  

(12)

\[
p^F = p^F_{(0)} \text{ in } \Omega^F \text{ at } t = 0,
\]  

(13)

are specified.
2.2 | Structure field

The structure field is governed by the equations of nonlinear elastodynamics. The balance of linear momentum reads

$$\rho^S \frac{d^2 d^S}{dt^2} = \nabla \cdot (F^S S^S) + \rho^S b^S \text{ in } \Omega^S \times (0, T).$$

(14)

The equation is solved for the structural displacement $d^S$. Here, $b^S$ denotes a prescribed body force, $F^S = \nabla d^S + I$ is the deformation gradient, and $S^S$ is the second Piola-Kirchhoff stress tensor. $\rho^S$ is the structural density, which is considered to be constant. Assuming a hyperelastic material behavior, $S^S$ is defined as $S^S = 2 \partial_\psi^S / \partial C^S$, where $C^S = F^S T F^S$ is the right Cauchy-Green tensor and $\psi^S$ is the strain energy function.

The partial differential equation is subject to the boundary conditions

$$d^S = \bar{d}^S \text{ on } \Gamma^S_D \times (0, T),$$

(15)

$$(F^S S^S) \cdot n^S = \bar{h}^S \text{ on } \Gamma^S_N \times (0, T),$$

(16)

where $\Gamma^S_D$ and $\Gamma^S_N$ denote the Dirichlet and the Neumann portions of the structural boundary, respectively, $n^S$ represents the outward-pointing normal vector, and $\bar{h}^S$ is the boundary traction. To complete the problem, the initial conditions

$$d^S = d^S_0 \text{ in } \Omega^S \text{ at } t = 0,$$

(17)

$$\frac{dd^S}{dt} = u^S_0 \text{ in } \Omega^S \text{ at } t = 0,$$

(18)

are specified.

2.3 | Grid motion

Several techniques may be employed to define the motion of the fluid mesh, such that the grid displacements and velocities can be computed purely based on prescribed boundary information as given by (4). The grid motion strategy is an artificial problem and does not affect the physics of the coupled problem; its sole purpose is to generate a proper mesh for the solution of the fluid problem. A detailed and comprehensive overview of classical ALE methods can be found by Donéa et al.\textsuperscript{25}

In this contribution, both the spring model and the elasticity approach have been employed in the numerical examples presented in Section 5. The spring model was introduced by Batina,\textsuperscript{26} and it interprets the finite element mesh as a network of linear springs. Another approach for mesh motion is the elasticity model, according to which the interior mesh displacements are obtained by solving the following equilibrium equation:

$$\nabla \cdot \sigma^G = 0 \text{ in } \Omega^F \times (0, T),$$

(19)

with

$$\sigma^G = \lambda^G \text{tr}(\epsilon^G) I + 2\mu^G \epsilon^G,$$

(20)

$$\epsilon^G = \frac{1}{2} \left( \nabla d^G + (\nabla d^G)^T \right),$$

(21)

where $\lambda^G$ and $\mu^G$ are the parameters of the pseudo material. Newly developed techniques based on a volumetric coupling scheme (see for example the work of Kronbichler et al.\textsuperscript{27}) allow to separately define the mesh for the fluid degrees of freedom and the grid on which the deformation is calculated, improving the robustness and the efficiency in comparison to classical ALE mesh moving techniques.
2.4 | Fluid-structure interface

At the fluid-structure interface $\Gamma_{\text{FSI}}$ kinematic and dynamic constraints have to be fulfilled:

$$\frac{\partial \mathbf{d}^S}{\partial t} = \mathbf{u}^F_{\text{on}} \quad \text{on} \quad \Gamma_{\text{FSI}} \times (0, T),$$

(22)

$$\mathbf{h}^S_{\text{on}} = -\mathbf{h}^F_{\text{on}} \quad \text{on} \quad \Gamma_{\text{FSI}} \times (0, T).$$

(23)

Equation (22) prohibits a mass flow across and a relative tangential movement of fluid and structure at the interface $\Gamma_{\text{FSI}}$, while Equation (23) states the balance of the surface tractions $\mathbf{h}^S_{\text{on}}$ and $\mathbf{h}^F_{\text{on}}$.

2.5 | Discretization

As one option, the spatial discretization of the FSI problem is performed by means of the finite element method in this paper, while both insights and conclusions of this study are independent of the specific discretization scheme employed. In particular, we use residual-based stabilized finite elements for the fluid domain and displacement-based or mixed/hybrid finite elements in the structural domain. For integration in time the generalized-$\alpha$ method is used both for the fluid and for the structural field. The sets of discretized nonlinear equations are denoted as $\mathbf{F}$, for the fluid field, $\mathbf{S}$, for the structural field, and $\mathbf{G}$, for the grid motion and are given as

$$\mathbf{F} \left( \mathbf{u}^{F,n+1}, P^{F,n+1}, \mathbf{d}^{G,n+1} \right) = \mathbf{h}^F, \quad \mathbf{S} \left( \mathbf{d}^{S,n+1} \right) = \mathbf{h}^S, \quad \mathbf{G} \left( \mathbf{d}^{G,n+1} \right) = 0.$$  

(24)

3 | DIRICHLET-NEUMANN COUPLING SCHEME

As mentioned in the introduction, both monolithic or partitioned solvers can be employed to solve the set of nonlinear equations 24. In this contribution, a partitioned solution algorithm based on a Dirichlet-Neumann coupling scheme is used. This approach sets the fluid field as the Dirichlet partition with prescribed interface velocities $\mathbf{u}^F_{\text{on}}$ and the structure field as the Neumann partition loaded with interface tractions $\mathbf{h}^F_{\text{on}}$. To clarify the scheme, the interior degrees of freedom are denoted with the subscript $(\cdot)_I$, while the degrees of freedom at the fluid-structure interface are denoted with the subscript $(\cdot)_{\Gamma}$. Moreover, the pressure degrees of freedom are subsumed in the variable $\mathbf{u}^F$ in the following. With this splitting, the fluid operator reads

$$\begin{bmatrix} \mathbf{F}_{II} & \mathbf{F}_{IT} \\ \mathbf{F}_{TI} & \mathbf{F}_{TT} \end{bmatrix} \begin{bmatrix} \mathbf{u}^F_I \\ \mathbf{u}^F_T \end{bmatrix} = \begin{bmatrix} \mathbf{h}^F_I \\ \mathbf{h}^F_T \end{bmatrix},$$

(25)

the structure operator reads

$$\begin{bmatrix} \mathbf{S}_{II} & \mathbf{S}_{IT} \\ \mathbf{S}_{TI} & \mathbf{S}_{TT} \end{bmatrix} \begin{bmatrix} \mathbf{d}^S_I \\ \mathbf{d}^S_T \end{bmatrix} = \begin{bmatrix} \mathbf{h}^S_I \\ \mathbf{h}^S_T \end{bmatrix},$$

(26)

and the grid operator reads

$$\begin{bmatrix} \mathbf{G}_{II} & \mathbf{G}_{IT} \\ \mathbf{G}_{TI} & \mathbf{G}_{TT} \end{bmatrix} \begin{bmatrix} \mathbf{d}^G_I \\ \mathbf{d}^G_T \end{bmatrix} = 0.$$  

(27)

With these definitions, following the scheme presented by Küttler and Wall,20 the solver coupling at each time step becomes as follows:

1. Start with predicted structural interface displacements $\mathbf{d}^{S,n+1}_T$.
2. Solve the grid equation

$$\mathbf{G}_{II} \mathbf{d}^{G,n+1}_I = -\mathbf{G}_{IT} \mathbf{d}^{G,n+1}_T.$$  

(28)
with interface displacements $d_i^{G,n+1} = d_i^{S,n+1}$ as Dirichlet condition. Then, evaluate the grid velocity (see also $^{28}$)

$$u_{G,n+1} = d_{G,n+1},$$

(29)

as well as the fluid interface velocity

$$u_{F,n+1} = d_{F,n+1}^{G,n+1}.$$  

(30)

Here, $d_{G,n+1}$ and $d_{F,n+1}^{G,n+1}$ are suitably defined time derivatives of the displacement field consistent with the time integration scheme, i.e., deduced from the respective variables available in the generalized-$\alpha$ method.

3. Solve the fluid equation

$$F_{II} u_{F,n+1} = h_{F,n} - F_{II} u_{F,n+1}^{F,n+1}$$

(31)

with interface fluid velocity $u_{F,n+1}^{F,n+1}$ as Dirichlet condition. We would like to point out that the equation of state, which represents the density variation with pressure, introduces an additional nonlinearity in the fluid problem aside from the usual convective term in the momentum equation. Once the fluid equation is solved, the coupling forces can be evaluated as

$$h_{F,n}^{F,n+1} = F_{II} u_{F,n}^{E,n+1} + F_{II} u_{F,n}^{E,n+1}.$$  

(32)

4. Solve the structure equation

$$\begin{bmatrix} S_{II} & S_{II} \\ S_{II} & S_{II} \end{bmatrix} \begin{bmatrix} d_{S,n+1}^{S,n+1} \\ d_{S,n+1}^{S,n+1} \end{bmatrix} = \begin{bmatrix} h_i^S \\ h_i^S - (1 - \alpha_f^S) h_i^{E,n+1} - \alpha_f^S h_i^{E,n+1} \end{bmatrix}$$

(33)

with fluid interface traction $h_i^F$ as Neumann condition, with $\alpha_f^S$ denoting the generalized-$\alpha$ method parameter for the time integration of the structural problem.

5. Check for convergence: continue with next time step if the algorithm is converged; otherwise, return to step 2. To define a stopping criterion, the increment of the structural interface displacements is introduced:

$$r_{S,n+1}^{i,n+1} = d_{S,n+1}^{S,n+1} - d_{S,n+1}^{S,n+1}.$$  

(34)

The length scaled norm of this increment is used as convergence criterion

$$\frac{\parallel r_{S,n+1}^{i,n+1} \parallel}{\sqrt{n_{eq}}} < \varepsilon.$$  

(35)

To ensure and/or accelerate the convergence of the fixed-point scheme, after each cycle, a relaxation step is introduced:

$$d_{S,n+1}^{S,n+1} = \omega d_{S,n+1}^{S,n+1} + (1 - \omega) d_{S,n+1}^{S,n+1}.$$  

(36)

Several techniques have been developed to identify an appropriate relaxation parameter. The simplest method is to choose a constant parameter $\omega$ for all time steps. The relaxation parameter has to fulfill two demands simultaneously: it has to be small enough to prevent the divergence of the iteration, and at the same time, it has to be as large as possible to reduce the number of FSI iterations. The optimal value of the relaxation parameter is not known a priori, and it strongly depends on the specific problem at hand. The only way to determine it is therefore by a trial and error approach. A more efficient and easy to implement technique to accelerate the convergence of the fixed-point algorithm for the FSI problem is a different relaxation parameter for every iterative step based on the so-called Aitken $\Delta^2$ method. $^{20}$ The key idea of this
method is to use the results of two previous iterations to improve the current solution. Following the work of Irons and Tuck,22 the relaxation parameter can be evaluated at each step of the nonlinear iteration according to

\[
\omega_{i+1} = -\omega_i \frac{r_{S,i+1}}{\left\| r_{S,i+2} - r_{S,i+1} \right\|}.
\]

(37)

In Section 5, it is shown how the weakly compressible formulation for the fluid dramatically reduces the required number of FSI iterations when using a fixed relaxation parameter. In the case of a dynamic adaptation of the relaxation parameter, the number of FSI iterations also decreases, while at the same time, the value of the relaxation parameter increases proportionally to the compressibility of the fluid introduced. As already stated, the main purpose of the relaxation parameter is to help the convergence of the FSI problem. The relaxation parameter is a single scalar value (at each iteration), and therefore, although its use is effective for very simple academic examples, it might be not sufficiently flexible for more complex problems of engineering interest, where the characteristics of the fluid flow and the structure may differ a lot from one part of the domain to another. The introduction of a weak compressibility allows the density of the fluid to adapt locally depending on the magnitude of the pressure and therefore helps to alleviate the constraints that the incompressibility poses on the interaction between the fluid and the structure field. As a consequence, significant improvements in the performance of the solution of the FSI problem can be achieved by making use of a weakly compressible formulation of the fluid motion.

## 4 | ARTIFICIAL ADDED MASS EFFECT

The artificial added mass effect is a well-known phenomenon that causes instabilities in sequentially staggered coupling schemes. It has been first observed and analyzed by Causin et al3 and Förster et al4 in the context of incompressible flows interacting with flexible structures. It has been shown that the stability is governed by the ratio of fluid and structural densities \( \rho_F / \rho_S \), and it depends also on the geometry of the problem. At the same time, the higher the temporal accuracy of the scheme adopted, the more severe the instability condition, and a reduction of the time step size does not have a beneficial effect. We will now analyze how this changes when considering weakly compressible flows.

### 4.1 | Weakly compressible added mass operator

The coupling force vector at the FSI interface can be symbolically written as

\[
f_I = m^F \mathcal{M}_A \hat{u}_I,
\]

(38)

where \( m^F \) denotes a characteristic fluid mass and \( \mathcal{M}_A \) represents the added mass operator, which maps a dimensionless interface acceleration onto a force vector at the FSI interface. It has been shown by Förster et al4 that, for small time step sizes and considering a lumped mass matrix, the added mass operator for incompressible fluids takes the following form:

\[
\mathcal{M}_{A,\text{incompr.}} = I_I + 2G_I (G_{I}^T G_I)^{-1} G_I^T,
\]

(39)

where \( G_I \) and \( G_I^T \) denote the discrete gradient operator related to the interior and the interface degrees of freedom, respectively. The first term in (39) represents the forces required to accelerate the fluid masses, which are directly adjacent to the FSI interface, while the second term yields forces generated by the effect of incompressibility. The sequentially staggered scheme is unconditionally unstable if

\[
\frac{m^F}{m_I^S} \max_i \mu_i > C_{\text{inst}},
\]

(40)

where \( \mu_i \) denotes the \( i \)th eigenvalue of the added mass operator while the constant \( C_{\text{inst}} \) depends on the time integration method, and it is smaller for higher temporal accuracy of the scheme. In the work of Förster et al,4 it has further been shown that there is no possibility to have such a standard sequentially staggered scheme, which would be stable regardless of the mass ratio. Thus, sequentially staggered schemes for incompressible flows may work in the case that the fluid is much lighter than the structural part and if the maximal eigenvalue is not much larger than one. However, the maximal
eigenvalue of the incompressibility operator $G_t(G_t^T G_t)^{-1} G_t^T$ tends to be rather large. In the case, mentioned above, of an incompressible flow entirely enclosed by Dirichlet boundary conditions and a coupling boundary $\Gamma$ (which is also treated as Dirichlet boundary), the maximal eigenvalue of this incompressibility operator is even infinite.

To derive the added mass operator for weakly compressible fluids, it is convenient to rewrite Equations (6) and (7) by means of material time derivatives:

$$\rho^F \frac{D u^F}{D t} - 2 \mu^F \nabla \cdot \epsilon'(u^F) + \nabla p^F = \rho^F b^F \text{ in } \Omega^F \times (0, T)$$  \hspace{1cm} (41)

$$\frac{1}{K_0^F + n^F (p^F - p_0^F)} \frac{D p^F}{D t} + \nabla \cdot u^F = 0 \text{ in } \Omega^F \times (0, T).$$  \hspace{1cm} (42)

Since we are interested in the behavior of the flow in the vicinity of the incompressible limit where density variations in time are small, it is justified to linearize the system such that Equations (41)-(42) are solved using the mass density of the previous time step while the density is updated subsequently by means of (3). Upon the finite element discretization, the discrete system of equations for the fluid field reads

$$M \dot{u} + G p = f$$  \hspace{1cm} (43)

$$a M^p \dot{p} - G^T u = 0,$$  \hspace{1cm} (44)

where $M$ denotes the usual fluid mass matrix, while $M^p$ is a mass matrix associated to the pressure degrees of freedom. The viscous term has been dropped because it exhibits a much lower impact than the mass terms when small time steps are used. The compressibility factor $a$ is given by

$$a = \frac{1}{K_0^F + n^F (p^F - p_0^F)}$$  \hspace{1cm} (45)

and tends to zero in the incompressible limit, i.e., when the reference bulk modulus approaches infinity. The superscript $(\cdot)^F$, denoting the variables related to the fluid field, is omitted in this section to ease the notation. For a general time discretization, the solution $x$ at the new time level $n + 1$ can be expressed as

$$x^{n+1} = \beta \Delta t x^{n+1} + \tilde{x},$$  \hspace{1cm} (46)

where $\tilde{x}$ denotes a linear combination of previous values and time derivatives, while $\beta$ depends on the particular time integration scheme. All one-step time discretization schemes for first order differential equations can be written in the form (46). Explicit integrators exhibit $\beta = 0$, while $\beta \neq 0$ will yield an implicit method. Using (46) for the velocity and the pressure allows to write

$$u = \beta_\nu \Delta t \dot{u} + \ddot{u}$$  \hspace{1cm} (47)

$$p = \beta_\nu \Delta t \dot{p} + \ddot{p},$$  \hspace{1cm} (48)

where the superscripts of the time level have been omitted for simplicity of notation. Introducing (47) and (48), Equations (43) and (44) read

$$M \dot{u} + \beta_\nu \Delta t G \dot{p} + G \ddot{p} = f$$  \hspace{1cm} (49)

$$a M^p \dot{p} - \beta_\nu \Delta t G^T \ddot{u} - G^T \dot{u} = 0,$$  \hspace{1cm} (50)

leading to the following system:

$$\begin{bmatrix}
M & \beta_\nu \Delta t G \\
-\beta_\nu \Delta t G^T & a M^p
\end{bmatrix}
\begin{bmatrix}
\dot{u} \\
\dot{p}
\end{bmatrix}
= \begin{bmatrix}
f - G \ddot{p} \\
G^T \ddot{u}
\end{bmatrix}.$$  \hspace{1cm} (51)
To derive the fluid added mass operator, the velocity vector is split into interior and interface degrees of freedom, following the notation in Section 3. Hence, the fluid operator can be written as

\[
\begin{bmatrix}
    M_{II} & M_{I\Gamma} & \rho_p \Delta t G_I \\
    M_{I\Gamma} & M_{\Gamma\Gamma} & \rho_p \Delta t G_{\Gamma} \\
    -\beta_u \Delta t G^T_I & -\beta_u \Delta t G^T_{\Gamma} & \alpha M^p
\end{bmatrix}
\begin{bmatrix}
    \ddot{u}_I \\
    \ddot{u}_\Gamma \\
    \ddot{p}
\end{bmatrix} =
\begin{bmatrix}
    -G_{I\Gamma} \ddot{p} \\
    -G_{\Gamma\Gamma} \ddot{p} + f_{I\Gamma} \\
    G^T \ddot{u}
\end{bmatrix}.
\] (52)

The solution of the system in terms of the interface acceleration leads to this expression of the coupling force vector:

\[
f_{I\Gamma} = \left( M_{I\Gamma} M_{\Gamma\Gamma}^{-1} G_I - G_{\Gamma\Gamma} \right) \left( \frac{\alpha}{\beta_u \rho_p \Delta t^2} M^p + G^T_{I\Gamma} M_{\Gamma\Gamma}^{-1} G_I \right)^{-1} \left( G^T_{I\Gamma} M_{\Gamma\Gamma}^{-1} M_{\Gamma\Gamma} - G_{\Gamma\Gamma} \right) - M_{I\Gamma} M_{\Gamma\Gamma}^{-1} M_{\Gamma\Gamma} + M_{I\Gamma} \right) \ddot{u}_\Gamma
\]

\[
- \left( M_{I\Gamma} M_{\Gamma\Gamma}^{-1} G_I - G_{\Gamma\Gamma} \right) \left( \frac{\alpha}{\beta_u \rho_p \Delta t^2} M^p + G^T_{I\Gamma} M_{\Gamma\Gamma}^{-1} G_I \right)^{-1} G^T_{I\Gamma} \frac{1}{\beta_u \Delta t} \ddot{u}
\]

\[
+ \left( M_{I\Gamma} M_{\Gamma\Gamma}^{-1} G_I - G_{\Gamma\Gamma} \right) \left( \frac{\alpha}{\beta_u \rho_p \Delta t^2} M^p + G^T_{I\Gamma} M_{\Gamma\Gamma}^{-1} G_I \right)^{-1} G^T_{I\Gamma} M_{\Gamma\Gamma}^{-1} G_I - M_{I\Gamma} M_{\Gamma\Gamma}^{-1} G_I + G_{\Gamma} \right) \ddot{p}.
\] (53)

The first line of Equation (53) shows the dependency of the interface forces on the interface acceleration and the term in square brackets can therefore be interpreted as a mass, while the second and third lines depend upon the history of velocity and pressure, respectively. By employing a lumped mass matrix, the off-diagonal terms \( M_{I\Gamma} \) and \( M_{I\Gamma} \) vanish and the following simplified expression of the interface force can be obtained:

\[
f_{I\Gamma, I} = \left[ 2m^F G_{I\Gamma} \left( \frac{2m^F \alpha}{\beta_u \rho_p \Delta t^2} M^p + G^T_{I\Gamma} G_I \right)^{-1} G^T_{I\Gamma} + m^F I_{\Gamma\Gamma} \right] \ddot{u}_\Gamma
\]

\[
+ \left[ 2m^F G_{I\Gamma} \left( \frac{2m^F \alpha}{\beta_u \rho_p \Delta t^2} M^p + G^T_{I\Gamma} G_I \right)^{-1} G^T_{I\Gamma} \frac{1}{\beta_u \Delta t} \ddot{u}
\]

\[
- \left[ G_{I\Gamma} \left( \frac{2m^F \alpha}{\beta_u \rho_p \Delta t^2} M^p + G^T_{I\Gamma} G_I \right)^{-1} G^T_{I\Gamma} G_I - G_{I\Gamma} \right] \ddot{p}.
\] (54)

As previously specified, \( m^F \) denotes a characteristic fluid mass, while the term \( m^F I_{\Gamma\Gamma} \) symbolically stands for the diagonal mass matrix related to the interface degrees of freedom. In the incompressible case when the coefficient \( \alpha \) tends to zero, the coupling force vector reduces to

\[
\lim_{\alpha \to 0} f_{I\Gamma, I} = \left[ 2m^F G_{I\Gamma} \left( G^T_{I\Gamma} G_I \right)^{-1} G^T_{I\Gamma} + m^F I_{\Gamma\Gamma} \right] \ddot{u}_\Gamma
\] (55)

which, by means of the definition (38), leads to the added mass operator for incompressible flows (39). On the other hand, as long as even a small compressibility is taken into account \((\alpha \neq 0), \text{ i.e., } K_w^p \neq \infty\), the interface force takes the general form (54) which, as the time step size approaches zero, reduces to

\[
\lim_{\Delta t \to 0} f_{I\Gamma, I} = m^F I_{\Gamma\Gamma} \ddot{u}_\Gamma + G_{I\Gamma} \ddot{p}
\] (56)

regardless of the actual compressibility. For the zero time step limit, Equation (48) implies the following equality:

\[
\ddot{p} = \ddot{p} = \text{const.}
\] (57)

Moreover, the second term in Equation (56) is balanced by external loads on the structure, and it does not influence the stability analysis. Therefore, the added mass operator for weakly compressible fluids is

\[
\mathcal{M}_{A, I}^{\text{weak compr.}} = I_{\Gamma\Gamma}.
\] (58)
Discussion on incompressible versus (weakly) compressible added mass effect

To understand the stability properties of sequentially staggered FSI solvers involving weakly compressible fluids in contrast to incompressible fluids, we have to compare the added mass operator given by (39) and its weakly compressible counterpart (58). The maximal eigenvalue of the added mass operator determines the onset of an unconditional instability according to (40). As the second term in the incompressible added mass operator (39) tends to have very large eigenvalues, the weakly compressible case behaves much more gently in the small time step limit.

On the other hand, the compressible formulation in the primitive variables, velocity, and pressure is able to recover the incompressible limit. However, from (54), it can be observed that the incompressible limit is approached only if $\alpha/\Delta t^2$ goes to zero. If for any fixed $\alpha$ the time step size is reduced, the influence of the incompressibility operator is diminished, explaining why FSI simulations with compressible flow solvers do not show the artificial added mass effect. However, from the instability condition (40) and the weakly compressible artificial added mass operator (58), we can deduce that also the compressible case will get unstable if

$$\frac{m^F}{m^S} > C_{\text{inst}},$$  \hspace{1cm} (59)

where the instability constant $C_{\text{inst}}$ depends upon the particular time discretization items and does not exceed one for schemes of practical interest. Consequently, also in the weakly compressible case, there will be a mass ratio at which the system gets unconditionally unstable. However, this limit mass ratio is much more permissive than that of the incompressible case. From the analysis, it is also clear that if the limit mass ratio given by (59) is not reached by a particular problem, then time step size reduction will eventually stabilize the staggered scheme. It is therefore obvious why potential stability problems in the compressible case frequently can be handled in the standard way, ie, by time step size reduction—an approach that has shown\textsuperscript{4} not to work in the incompressible case. A similar observation has been made by van Brummelen\textsuperscript{5} where it has been shown for a 2D model problem that the added mass in the compressible case is proportional to the length of the time step. This beneficial property constitutes an important motivation for the introduction of a weakly compressible formulation for the solution of FSI problems.

5 | NUMERICAL EXAMPLES

In this section, we present numerical results that show the computational advantages and the robustness of the weakly compressible formulation for the solution of FSI problems.

5.1 | Driven cavity with flexible bottom

The first example is a 2D driven cavity with flexible bottom that has been used for several numerical studies. The cavity is a unit square, and the structure on the bottom has a thickness of 0.002 m. The structure is modeled as a Saint-Venant–Kirchhoff material with Young’s modulus $E^S = 250 \text{ N/m}^2$ and Poisson’s ratio $\nu^S = 0$. The main feature of this example is that both the structural configuration and the material of the very flexible structure are chosen such that the structure’s main resistance to fluid pressure derives from its density. This allows to set several test cases with increasing difficulty for the coupling algorithm, by considering different structural densities. For this example, three values of the structural density are considered: $\rho^S = 50 \text{ kg/m}^3$, $\rho^S = 500 \text{ kg/m}^3$, and $\rho^S = 5000 \text{ kg/m}^3$. To make a comparison and to evaluate the advantages of the weakly compressible framework with respect to the classical incompressible formulation, first the fluid is modeled as an incompressible Newtonian flow field with density $\rho^F = 1 \text{ kg/m}^3$ and dynamic viscosity $\mu^F = 0.01 \text{ kg/(m \cdot s)}$ and in the following set of experiments, it is assumed to be governed by the weakly compressible formulation of the Navier-Stokes equations, with $\rho^F = 1 \text{ kg/m}^3$, $p^F = 0 \text{ N/m}^2$ and $\mu^F = 0.01 \text{ kg/(m \cdot s)}$ and considering the following values of the reference bulk moduli: $K_0^F = 10 \text{ N/m}^2$, $K_0^S = 100 \text{ N/m}^2$ and $K_0^V = 1000 \text{ N/m}^2$. For the sake of simplicity, the relation between density and pressure is considered to be linear ($n^V = 1$). At the top of the cavity, the velocity in $x$-direction is prescribed according to $\bar{u}_x(t) = 1 - \cos(2\pi \frac{t}{T}) \text{ m/s}$ with $T = 5 \text{ s}$, while the velocity in $y$-direction is kept equal to zero. On the lateral walls of the fluid domain, no-slip boundary conditions are imposed. The structural domain is clamped on its left and right edges. In the left panel of Figure 1, the geometry and the boundary conditions of the driven cavity problem with flexible structure are sketched.
The fluid unknowns, ie, both velocity and pressure, are discretized with 32 × 32 stabilized first-order quadrilateral elements, while the structure is discretized with 32 × 1 first-order quadrilateral elements. At each side of the cavity, there are two unconstrained nodes that allow free inflow and outflow of fluid. The time step is $\Delta t = 0.1$ s, and the final time of the simulation is set to 15 s. The temporal discretization is performed with the generalized-$\alpha$ time integration for both the fluid and the structure field. The parameters for the temporal integration are $\alpha_F^f = 1.0$, $\alpha_F^m = 1.0$ and $\gamma_F = 1.0$ for the fluid and $\alpha_S^f = 0.0$, $\alpha_S^m = 0.0$, $\rho_S = 0.5$ and $\gamma_S = 1.0$ for the structure. With regards to the relaxation techniques, both the case of a fixed relaxation parameter $\omega = 0.825$, determined by a trial and error approach, and the case of a dynamic relaxation according to Aitken’s $\Delta^2$ method are considered.

In Figure 2, we summarize the average number of FSI iterations required at each time step as a function of the reference bulk modulus ($K_0^F = \infty$ denotes the results obtained with the incompressible formulation) considering the three values of the structural density and the two relaxation techniques discussed in Section 3. We can first notice that the most challenging example, where $\rho_S = 50$ kg/m$^3$ and where the relaxation parameter is constant during the simulation, converges neither for the weakly compressible case nor for the incompressible case, as known from literature. From the data, a general trend can be observed: the weakly compressible formulation accelerates the convergence of the iterative solver that resolves the FSI coupling and the higher the compressibility, the more iterations can be saved. For example, the average number of FSI iterations per time step in the case of $\rho_S = 500$ kg/m$^3$ and $\omega = 0.825$ drops from 12.2 with the incompressible formulation to 2.7 with the weakly compressible formulation with $K_0^F = 10$ N/m$^2$, a saving of about 78% of FSI iterations. The evolution of the number of FSI iterations over the course of the simulation for this case is plotted in the left panel of Figure 3. From Figure 2, it can also be noticed that the number of FSI iterations decreases also with a dynamic adaptation of the relaxation parameter, but in this case, the advantages are less accentuated (the reasons for which are explained above, ie, at the end of Section 3). Considering the same set of data as for the fixed relaxation parameter, the FSI iterations decrease by about 35%, while the maximum reduction in iterations obtained with Aitken’s method.
(48%) is observed in the most challenging case with $\rho_s = 50 \, \text{kg/m}^3$. In the right panel of Figure 3, the data for this case is monitored in more detail.

As anticipated in Section 3, the introduction of a weak compressibility in the fluid leads not only to a reduction of the number of FSI iterations, but in the case of Aitken’s $\Delta^2$ method, it also comes along with a significant increase in the relaxation parameter during the simulation. The plots in Figure 4 show the evolution of the relaxation parameter over the time steps considering the different levels of compressibility and the three values of structural density defined before. To homogeneously compare the results, the value of $\omega$ for each time step has been evaluated as the arithmetic mean of the relaxation parameters obtained in all the FSI iterations inside the considered step. It can be immediately observed how the relaxation parameter decreases with the difficulty of the coupled problem, but at the same time, it increases with the compressibility of the fluid. Moreover when decreasing the strength of the FSI coupling, the relaxation parameter tends to have a less oscillating behavior. Figure 5 summarizes the data of the previous three graphs, plotting the average of the relaxation parameter over the whole simulation as a function of the reference bulk modulus, making the considerations just exposed more clear. The advantages in terms of relaxation parameter are again more accentuated in the most challenging FSI cases, but they become almost negligible for trivial situations. In fact the average relaxation parameter increases for the weakly compressible formulation by about 72% for $\rho_s = 50 \, \text{kg/m}^3$, by 35% for $\rho_s = 500 \, \text{kg/m}^3$ while it increases by just 5% for $\rho_s = 5000 \, \text{kg/m}^3$ (also demonstrating again the usefulness of the Aitken method).

5.2 | Pressure wave in flexible tube

The second example is a three-dimensional flow in a flexible tube according to Fernández and Moubachir and Gerbeau and Vidrascu. This FSI problem arises in the modeling of blood flow in large arteries and meanwhile constitutes a classic benchmark in hemodynamics and especially for examining different coupling strategies in a challenging situation. The tube has a length of $l = 5 \, \text{cm}$, an outer radius of $r_o = 0.6 \, \text{cm}$, and an inner radius of $r_i = 0.5 \, \text{cm}$. The solid is described by the linear elasticity equations (Saint-Venant–Kirchhoff material) with Young’s modulus $E = 3 \cdot 10^9 \, \text{g/(cm}^2\text{)}$ and Poisson’s ratio $\nu = 0.3$, while the structural density is $\rho^s = 1.2 \, \text{g/cm}^3$. The fluid is first considered incompressible with density $\rho^F = 1.0 \, \text{g/cm}^3$ and dynamic viscosity $\mu^F = 0.03 \, \text{g/(cm} \cdot \text{s})$. Then, the weakly compressible formulation of the Navier-Stokes equations is adopted with $\rho_0^F = 1.0 \, \text{g/cm}^3$, $\rho_0^F = 0 \, \text{g/(cm} \cdot \text{s}^2\text{)}$, and $\mu^F = 0.03 \, \text{g/(cm} \cdot \text{s})$. Like in the previous example, the material parameter is $n^F = 1$, while four different orders of magnitude of the reference bulk modulus are considered: $K_0^F = 10^5 \, \text{g/(cm} \cdot \text{s}^2\text{)}$, $K_0^F = 10^6 \, \text{g/(cm} \cdot \text{s}^2\text{)}$, $K_0^F = 10^7 \, \text{g/(cm} \cdot \text{s}^2\text{)}$, and $K_0^F = 2.2 \cdot 10^{10} \, \text{g/(cm} \cdot \text{s}^2\text{)}$, with the latter representing the reference bulk modulus of water. The first three parameters are in the range where the solution starts to globally alter the physical behavior compared to the incompressible formulation in this setup (see also Figure 10 below), whereas the water case serves as a reference for a real material. The time step used in the simulations is $\Delta t = 0.0001 \, \text{s}$. The solid tube is clamped at both ends, and the fluid is initially at rest. It is loaded with a surface traction of $\bar{h}_F = 1.3332 \cdot 10^4 \, \text{g/(cm} \cdot \text{s}^2\text{)}$ at the front of the cylinder for the duration of $3 \cdot 10^{-3} \, \text{s}$. The velocity of the fluid at the end
of the cylinder is prescribed to zero, simulating a condition in which the tube is closed at that end. As a consequence of these boundary conditions, a pressure wave travels along the longitudinal axis of the tube. The geometry of the problem is sketched in the left panel of Figure 6.

The fluid unknowns are discretized with 2480 stabilized trilinear hexahedral elements, while the structure is discretized with 1600 trilinear hexahedral elements. The temporal discretization is performed with the generalized-\(\alpha\) time integration for both the fluid and the structure field with the same parameters as in the previous example. The relaxation of the fixed-point method for the solution of the FSI problem is performed according to Aitken’s \(\Delta^2\) method.

In the left panel of Figure 7 the evolution of the number of FSI iterations at each time step is plotted for the different levels of compressibility considered. In the more challenging pressure wave example, the decrease in iterations is even more dramatic than in the driven cavity example. The average number of FSI iterations drops down from 27.8 for the
FIGURE 5  Simulation-average relaxation parameter with Aitken's $\Delta^2$ method for the driven cavity with flexible bottom [Colour figure can be viewed at wileyonlinelibrary.com]

FIGURE 6  (Left) Geometry and (Right) snapshot at $t = 0.0055$ s of the pressure wave example [Colour figure can be viewed at wileyonlinelibrary.com]

FIGURE 7  (Left) Number of fluid-structure interaction iterations and (Right) computational time in each time step for the pressure wave example [Colour figure can be viewed at wileyonlinelibrary.com]

incompressible case to 15.7, 3.4, 1.7 and 1.0 when using the weakly compressible formulation with the reference bulk moduli equal to $K_0^F = 2.2 \cdot 10^{10}$ g/(cm $\cdot$ s$^2$), $K_0^F = 10^7$ g/(cm $\cdot$ s$^2$), $K_0^F = 10^6$ g/(cm $\cdot$ s$^2$), and $K_0^F = 10^5$ g/(cm $\cdot$ s$^2$), respectively. The corresponding saving of FSI iterations is equal to 44%, 88%, 94%, and 96% and the total saving in terms of computational time is equal to 48%, 74%, 81%, and 83%, respectively. These numbers clearly show that the increase of the computational cost of the fluid solver due to the (weak) compressibility has only a marginal effect compared to the decrease of the computational time induced by the reduction of the number of coupling iterations. The right panel of Figure 7 details the computational time spent in all the time steps.
With regards to the relaxation parameter obtained with Aitken’s method, Figure 8 clearly shows how its value increases with the increase in the compressibility in the fluid. The average relaxation parameter rises from 0.11 in the incompressible case to 0.87 for the most compressible case, leading to an increase in a factor of 8.

For this numerical example also, the effect of the time step size is investigated with respect to the average number of FSI iterations and the total computational time. The temporal domain is therefore discretized with a variable number of time steps, ranging from 10 to 150. The left panel of Figure 9 shows how the average number of FSI iterations does not decrease when reducing the time step size in the incompressible case, while in the weakly compressible case, this number decreases progressively for sufficiently small time step sizes. From the right panel of Figure 9 it can be observed that the computational time required to solve the problem in the incompressible and the weakly compressible case is approximately the same when a very large time step size is used, while the gap in computational time increases significantly when smaller time step sizes are considered. Moreover, this gap is larger for higher compressibility.

From the analysis in Section 4.1, it follows that, in some cases, the incompressible solver might fail, while, if the instability condition (59) is not satisfied, the weakly compressible solver might fail for a certain $\Delta t$ and be stable for sufficiently small values of the time step size. If for example this pressure wave problem is tried to be solved without relaxation (ie, the relaxation parameter is constant end equal to 1), the incompressible solver is never able to converge, regardless of the time step size. On the other hand, considering the weakly compressible case with $K_F^0 = 10^6 \text{g/(cm} \cdot \text{s}^2)$, the solver fails if the temporal domain is discretized with 8, 16, 32, and 64 time steps, but it is stable if the time steps are 128, 256, and 512, exhibiting a progressively reduced number of coupling iterations.
Obviously, the weakly compressible formulation slightly changes the behavior of the dynamics of the problem, and these changes are inversely proportional to the magnitude of the reference bulk modulus. In Figure 10, the structure radial displacement at point \((2.5, 0.6, 0.0)\) and the fluid pressure at point \((2.5, 0.0, 0.0)\) are depicted. For this simple example, it is clear how the compressibility of the fluid induces a decrease in the pressure wave speed, which results in a shift of the peaks of the curves to the right in the graphs in Figure 10. In particular, it can be observed that the solution obtained with the weakly compressible formulation using the reference bulk modulus of water nicely matches the incompressible solution. Moreover, the results obtained with \(K_F^0 = 10^7 \text{ g/(cm} \cdot \text{s}^2)\) and \(K_F^0 = 10^6 \text{ g/(cm} \cdot \text{s}^2)\) still accurately reproduce the physical behavior of the system, while for smaller values of the reference bulk modulus the mismatch with the incompressible solution becomes excessive. The value of \(K_F^0\) plays therefore an important role in the FSI problem. We can qualitatively state that the value of the reference bulk modulus has to be large enough to accurately reproduce the physical behavior of the system, but at the same time, it should be chosen as small as possible to benefit from the advantages of the weakly compressible formulation. It is also worth recalling that the incompressibility is a mathematical assumption which simplifies the solution of a big variety of problems, but each fluid exhibits a certain level of compressibility that, under the conditions discussed in Section 1, can be taken into account through Equation (3). Hence, in strict terms, the incompressible solution cannot be taken as reference to evaluate the fidelity of the simulation. A better reference can therefore be obtained by estimating the parameters in Equation (3) from experimental data, of which the literature is abundant. From the numerical experiments presented here, it can be observed that a good trade-off between the accuracy of the simulations and the computational advantages is achieved when \(K_F^0\) is chosen such that

\[
\frac{p_{\text{max}}^F - p_0^F}{K_0^F} = \varepsilon
\]

with \(\varepsilon = 10^{-3} - 10^{-2}\) and where \(p_{\text{max}}^F\) denotes the maximum pressure in the fluid field.

### 6 CONCLUSION

A weakly compressible formulation of the instationary Navier-Stokes equations for Newtonian fluids has been proposed in the context of FSI problems. A pressure-based approach has been adopted to handle low Mach number flows, choosing the pressure as primary variable and replacing the divergence-free condition of the velocity field for incompressible fluids by the continuity equation of compressible flows. For this purpose, an equation of state by the Murnaghan-Tait model has been used, defining a one-to-one relation between density and pressure. The approach has been embedded into a fixed-point FSI solver, and two techniques for the evaluation of the relaxation parameter have been considered.
The introduction of a weak compressibility sometimes allows to more accurately reproduce the physics of the problem at hand by choosing a suitable reference bulk modulus for the considered flow. Moreover, this formulation has been demonstrated to have several beneficial effects for the coupling of the different fields. The advantages of the formulation against potential instabilities caused by the artificial added mass effect have been demonstrated analytically, while the numerical test cases presented have shown a significant decrease of the average number of FSI iterations and the computational time and an increase of the average relaxation parameter evaluated according to Aitken’s $\Delta^2$ method, even in those simple academic examples that do not at all explore the full potential of the discussed approach of introducing such a weak compressibility. The advantages are more accentuated in the most challenging FSI problems and when increasing the compressibility of the fluid. Furthermore, the weakly compressible formulation allows the density of the fluid to adapt itself locally, depending on the magnitude of the pressure, which helps to alleviate the constraints that the incompressibility poses on the coupling between the fluid and the structure field. Overall, we believe that this is a very promising approach to efficiently exploit more complex problems of engineering interest, where the characteristics of the fields may differ a lot throughout the domain and where conventional techniques that are usually introduced to accelerate convergence are insufficient.

ACKNOWLEDGEMENTS

This work was supported by the European Education, Audiovisual and Culture Executive Agency (EACEA) under the Erasmus Mundus Joint Doctorate Simulation in Engineering and Entrepreneurship Development (SEED), FPA 2013-0043. Open access funding enabled and organized by Projekt DEAL. [Correction added on 31 October 2020, after first online publication: Projekt Deal funding statement has been added.]

ORCID

Martin Kronbichler https://orcid.org/0000-0001-8406-835X

REFERENCES

1. Klainerman S, Majda A. Compressible and incompressible fluids. *Commun Pure Appl Math*. 1982;35(5):629-651.
2. Farhat C. *CFD-Based Nonlinear Computational Aeroelasticity*. New York, NY: John Wiley & Sons, Ltd; 2004.
3. Causin P, Gerbeau JF, Nobile F. Added-mass effect in the design of partitioned algorithms for fluid–structure problems. *Comput Methods Appl Mech Eng*. 2005;194(42):4506-4527.
4. Förster C, Wall WA, Ramm E. Artificial added mass instabilities in sequential staggered coupling of nonlinear structures and incompressible viscous flows. *Comput Methods Appl Mech Eng*. 2007;196(7):1278-1293.
5. van Brummelen E. Added mass effects of compressible and incompressible flows in fluid-structure interaction. *J Appl Mech*. 2009;76(2):021206-021212.
6. van Brummelen E. Partitioned iterative solution methods for fluid-structure interaction. *Int J Numer Methods Fluids*. 2011;65(3-4):180-201.
7. Fehn N, Wall W, Kronbichler M. A matrix-free high-order discontinuous Galerkin compressible Navier–Stokes solver: a performance comparison of compressible and incompressible formulations for turbulent incompressible flows. *Int J Numer Methods Fluids*. 2019;89(3):71-102.
8. Sudhakar Y, Wall WA. A strongly coupled partitioned approach for fluid-structure-fracture interaction. *Int J Numer Methods Fluids*. 2017;87(2):90-108.
9. Råback P, Ruokolainen J, Lyly M, Järvinen E. Fluid-structure interaction boundary conditions by artificial compressibility. Paper presented at: ECCOMAS Computational Fluid Dynamics Conference; 2001; Swansea, UK.
10. Järvinen E, Råback P, Lyly M, Salenius I-P. A method for partitioned fluid–structure interaction computation of flow in arteries. *Med Eng Phys*. 2008;30(7):917-923.
11. Degroote J, Swillens A, Bruggeman P, Haelterman R, Segers P, Vierendeels J. Simulation of fluid–structure interaction with the interface artificial compressibility method. *Int J Numer Methods Biomed Eng*. 2010;26(3-4):276-289.
12. Bogaers AEJ, Kok S, Reddy B, Franz T. Extending the robustness and efficiency of artificial compressibility for partitioned fluid–structure interactions. *Comput Methods Appl Mech Eng*. 2015;283:1278-1295.
13. Küttler U, Förster C, Wall WA. A solution for the incompressibility dilemma in partitioned fluid-structure interaction with pure Dirichlet fluid domains. *Computational Mechanics*. 2006;38(4):417-429.
14. van Opstal T, van Brummelen E, van Zwieten GJ. A finite-element/boundary-element method for three-dimensional, large-displacement fluid–structure-interaction. *Comput Methods Appl Mech Eng*. 2015;284:637-663. Isogeometric Analysis Special Issue.
15. Riemsdijk K, Vierendeels J, Dick E. A simple but efficient coupling procedure for flexible wall fluid-structure interaction. In: Proceedings of the AIAA Fluids 2000 Conference; 2000; Denver, CO.
16. Karki KC, Patankar SV. Pressure based calculation procedure for viscous flows at all speeds in arbitrary configurations. *AIAA Journal*. 1989;27(9):1167-1174.
17. Rhie CM. Pressure-based Navier-Stokes solver using the multigrid method. *AIAA Journal*. 1989;27(8):1017-1018.
18. Gee MW, Küttler U, Wall WA. Truly monolithic algebraic multigrid for fluid-structure interaction. *Int J Numer Methods Eng.* 2011;85(8):987-1016.
19. Heil M. An efficient solver for the fully coupled solution of large-displacement fluid–structure interaction problems. *Comput Methods Appl Mech Eng.* 2004;193(1–2):1-23.
20. Küttler U, Wall WA. Fixed-point fluid–structure interaction solvers with dynamic relaxation. *Computational Mechanics.* 2008;43(1):61-72.
21. Fernández MA, Gerbeau J-F, Grandmont C. A projection semi-implicit scheme for the coupling of an elastic structure with an incompressible fluid. *Int J Numer Methods Eng.* 2007;69(4):794-821.
22. Irons BM, Tuck RC. A version of the Aitken accelerator for computer iteration. *Int J Numer Methods Eng.* 1969;1(3):275-277.
23. Macdonald JR. Some simple isothermal equations of state. *Rev Mod Phys.* 1966;38:669-679.
24. Brujan EA. A first-order model for bubble dynamics in a compressible viscoelastic liquid. *J Non-Newton Fluid Mech.* 1999;84(1):83-103.
25. Donéa J, Huerta A, Ponthot JP, Rodríguez-Ferran A. *Arbitrary Lagrangian–Eulerian Methods.* New York, NY: John Wiley & Sons, Ltd; 2004.
26. Batina JT. Unsteady Euler algorithm with unstructured dynamic mesh for complex-aircraft aerodynamic analysis. *AIAA Journal.* 1991;29(3):327-333.
27. La Spina A, Kronbichler M, Vuong A-T, Farah P, Wall WA. A robust, efficient and easy to implement moving grid approach to fluid-structure interaction based on a volumetric coupling scheme. In press.
28. Förster C, Wall WA, Ramm E. On the geometric conservation law in transient flow calculations on deforming domains. *Int J Numer Methods Fluids.* 2006;50(12):1369-1379.
29. Fernández MA, Moubachir M. A Newton method using exact Jacobians for solving fluid–structure coupling. *Comput Struct.* 2005;83(2-3):127-142.
30. Gerbeau J-F, Vidrascu M. A quasi-Newton algorithm based on a reduced model for fluid-structure interaction problems in blood flow. *Math Model Numer Anal.* 2003;37(4):631-647.

**How to cite this article:** La Spina A, Förster C, Kronbichler M, Wall WA. On the role of (weak) compressibility for fluid-structure interaction solvers. *Int J Numer Meth Fluids.* 2020;92:129–147. [https://doi.org/10.1002/fld.4776](https://doi.org/10.1002/fld.4776)