Journal of Computational Physics 313 (2016) 13–30

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

# A penalty-projection algorithm for a monolithic fluid-structure interaction solver

D. Cerroni, S. Manservisi *

*DIN, University of Bologna, Montecuccolino Laboratory, Via dei Colli 16, 40136 Bologna, Italy*

ARTICLE INFO

*Article history:*
Received 8 July 2015
Received in revised form 17 February 2016
Accepted 17 February 2016
Available online 20 February 2016

*Keywords:*
Fluid structure interaction
Penalty-projection method
Monolithic solver

ABSTRACT

In this paper we propose a new iterative penalty-projection algorithm for a monolithic fluid-structure interaction solver. Projection methods, that split the computation of the velocity from the pressure, are very popular in fluid dynamics since the boundary errors generated by the projection method are localized mainly near the boundary layers where the incorrect pressure boundary conditions are imposed. However, when solid regions are taken into account, the pressure projected field cannot satisfy fully coupled boundary constraints imposed on external solid surfaces such as stress-free conditions, and, due to the rigidity of the medium, the boundary errors propagate deeply in the interior. In order to reduce the projection errors we propose a one-step penalty-projection method in the fluid domain and an iterative penalty-projection method in the solid region. This technique decouples the pressure-velocity degrees of freedom and, as a consequence, the computational cost. In order to verify the accuracy and robustness of the proposed method we compare the results between this splitting velocity-pressure algorithm and the coupled one. These numerical results show stability and robustness of the proposed algorithm with a strong reduction of the computational effort.



## 1. Introduction

In the last decades the number of Fluid-Structure applications has been increasing in various fields of engineering. Some examples are the studies of the stability response of aircraft wings, the blood flow through arteries, the response of bridges and tall buildings to winds, the vibration of compressor blades and the studies of oscillations in heat exchangers (e.g., see [1–5,33] and citations therein). A fluid structure interaction problem solves the conservative set of equations over both fluid and solid domain.

The movement of the mesh can be taken into account with an Eulerian or Lagrangian approach. In the first case the solid domain can undergo deformations that may change the topology of the computational grid. In this case an immersed boundary method can be used [50,49]. For example, in [48], the motion of a flexible membrane is analyzed using a fluid solver on a fixed Cartesian grid. Such a large deformation cannot be handled by using a Lagrangian approach where the grid should follow the body domain. The Lagrangian approach, which does not show spurious numerical currents on the interface, is preferred when the change of topology does not cause overlapping in the mesh structure. Clearly, imposing the boundary conditions by using immersed boundary methods is not trivial. Issues on the accuracy of boundary treatment and conservation properties of the numerical scheme are not still open questions. The matching between the grid lines and the

* Corresponding author.
*E-mail addresses:* daniele.cerroni2@unibo.it (D. Cerroni), sandro.manservisi@unibo.it (S. Manservisi).

http://dx.doi.org/10.1016/j.jcp.2016.02.041
0021-9991/

body surface in standard moving grid approach allows better control of the grid resolution in the vicinity of the body. This is a crucial aspect with the increase of grid size and Reynolds number [50].

The most common solution strategy implemented in software packages is the so-called partitioned approach, which decouples the problem into two separate sub-problems and uses dedicated software for each different region. According to this method, the coupling is achieved by enforcing continuity conditions along the fluid-solid interface. For details the interested reader can see [5–9]. However, when large displacements may occur, one can see that explicit partitioned algorithms show instabilities due to the poor fluid-solid coupling matching where the solid-fluid power remains unbalanced at the interface.

In order to overcome this limit one could implicitly enforce coupling conditions that solve simultaneously for the fluid and structure unknowns, so that the solid and fluid regions are treated as a single continuum. These algorithms are called monolithic and usually exhibit good stability properties even with similar fluid and solid density [10–12,14,15]. Monolithic algorithms are stable and robust but they are also CPU-time expensive [16–21]. In a monolithic approach for numerical simulations of incompressible fluid-structure bodies the velocity and the pressure fields are coupled. This issue is enhanced by the saddle-point character of the incompressible Lagrange multiplier formulation and by multi-dimensional geometries where one has to deal with a high number of degrees of freedom [22,17,16,23]. The reduction of the computational cost for such problems motivate many papers published in recent years [4,24–26,31,32]. In order to reduce such a computational cost it is important to mention monolithic iterative solvers which use velocity-pressure splitting preconditioners, keep the original number of degrees of freedom and preserve exact boundary conditions, see for example [47,46,28–30] and references therein. In [47,46] the global system is solved by computing iteratively small systems defined by groups of finite element triangulations. In this case pressure projection and pressure-velocity coupled algorithms can be used independently over different sub-regions. In [28], corrections, implemented in the framework of finite element method, are investigated to improve several coupling strategies and, in [29], an efficient projection preconditioner method is used to solve the coupled system. Finally, recently in [30], the uncoupled velocity and pressure fields are related through the use of Lagrangian multipliers.

Projection methods, which split the velocity from the pressure degrees of freedom, are very popular in fluids since time dependent solutions can be obtained within a single iterative step and the boundary error can be easily controlled near boundary layers. In regions where the parameters, that characterized the medium, are larger over several orders of magnitude, the projection error propagates quickly in the interior and these errors may not be solved with a few iterative steps [28,34]. The most attractive feature of projection methods is that, at each time step, one only needs to solve a sequence of decoupled advective and elliptic equations for the velocity and the pressure field, making it very efficient for large scale time-dependent numerical simulations [25]. It is well known that standard one-step projection methods for fluids produce pressure solutions that are not exact near the domain boundary where velocity fields should be imposed and they must be completed with iterative algorithms able to correct the projection boundary conditions and the corresponding incompressible constraint. In a fluid domain the error is generated on the external walls where the velocity is prescribed while for a solid the main error can be found on the stress-free external walls.

In this paper we propose a new iterative penalty-projection algorithm for a generic monolithic fluid-structure interaction solver based on unstructured computational grids. In particular our strategy is to use a one-step penalty-projection method in the fluid domain and a few step iterative penalty-projection method in the solid region. The iterative algorithm considered here is the iterative penalty method which can be proved to be convergent under simple conditions [35]. The combination of the penalty and the projection method in the fluid and solid region at each time step leads to good performance as shown in the numerical section of the paper. The paper is organized as it follows: in the next section we give the mathematical description of the algorithm, then we test the stability and the accuracy of different cases taken from literature [45,5,11].

## 2. Mathematical model

### 2.1. Notation

In this section we introduce mathematical notation used in the paper. We denote by $H^s(\mathcal{O})$, $s \in \Re$, the standard Sobolev space of order $s$ with respect to the set $\mathcal{O}$, which is either the flow domain $\Omega$, or its boundary $\Gamma$, or part of its boundary. Whenever $m$ is a non-negative integer, the inner product over $H^m(\mathcal{O})$ is denoted by $(f,g)_m$ and $(f,g)$ denotes the inner product over $H^0(\mathcal{O}) = L^2(\mathcal{O})$. Hence, we associate with $H^m(\mathcal{O})$ its natural norm $\|f\|_{m,\mathcal{O}} = \sqrt{(f,f)_m}$. For $1 \le p < \infty$ the Sobolev space $W^{m,p}(\mathcal{O})$ is defined as the closure of $C^\infty(\mathcal{O})$ in the norm

$$\|f\|^p_{W^{m,p}(\mathcal{O})} = \sum_{|\alpha|\le m} \int_{\mathcal{O}} |\Big(\frac{\partial}{\partial x}\Big)^\alpha f(x)|^p \, dx \,.$$

The closure of $C_0^\infty(\mathcal{O})$ under the norm $\|\cdot\|_{W^{m,p}(\mathcal{O})}$ will be denoted by $W_0^{m,p}(\mathcal{O})$. Whenever possible, we will neglect the domain label in the norm. For vector-valued functions and spaces, we use boldface notation. For example, $\mathbf{H}^s(\Omega) = [H^s(\Omega)]^n$ denotes the space of $\Re^n$-valued functions such that each component belongs to $H^s(\Omega)$. Of special interest is the space

$$\mathbf{H}^1(\Omega) = \left\{ v_j \in L^2(\Omega) \;\middle|\; \frac{\partial v_j}{\partial x_k} \in L^2(\Omega) \quad \text{for } j,k = 1,2 \right\}$$