Rev. int. métodos numér. cálc. diseño ing. 2016;xxx(xx):xxx–xxx

Revista Internacional de Métodos Numéricos para Cálculo y Diseño en Ingeniería

www.elsevier.es/rimni

# A modified fractional step method for fluid–structure interaction problems

P. Ryzhakov

*Centre Internacional de Mètodes Numèrics en Enginyeria (CIMNE), Spain*[1]

ARTICLE INFO

*Article history:*
Received 2 September 2015
Accepted 16 September 2015
Available online xxx

*Keywords:*
Fluid–structure interaction
Free surface flows
Lagrangian
Quasi-incompressible
Added mass

ABSTRACT

We propose a Lagrangian fluid formulation particularly suitable for fluid–structure interaction (FSI) simulation involving free-surface flows and light-weight structures. The technique combines the features of fractional step and quasi-incompressible approaches. The fractional momentum equation is modified so as to include an approximation for the current-step pressure using the assumption of quasi-incompressibility. The volumetric term in the tangent matrix is approximated allowing for the element-wise pressure condensation in the prediction step. The modified fractional momentum equation can be readily coupled with a structural code in a partitioned or monolithic fashion. The use of the quasi-incompressible prediction ensures convergent fluid–structure solution even for challenging cases when the densities of the fluid and the structure are similar. Once the prediction was obtained, the pressure Poisson equation and momentum correction equation are solved leading to a truly incompressible solution in the fluid domain except for the boundary where essential pressure boundary condition is prescribed. The paper concludes with two benchmark cases, highlighting the advantages of the method and comparing it with similar approaches proposed formerly.



## 1. Introduction

Fluid–structure interaction (FSI) problems involving incompressible fluid flows and flexible structures are found in many civil and mechanical engineering applications. Active research has been carried out in the field of FSI over past two decades and multiple numerical models were developed (a review can be found, e.g. in [1]). For the problems involving light-weight structures interacting with free-surface flows quasi-incompressible Lagrangian fluid formulations coupled to the standard structural formulations proved to be very advantageous. The evolution of the free-surfaces and FSI interfaces could be tracked since it was naturally defined by the position of the moving Lagrangian mesh. On the other hand, quasi-incompressible formulations circumvent the added mass effect [2] typically encountered when standard truly incompressible fluid formulations were used [3]. This benefit was achieved due to the relaxation of the incompressibility constraint introduced by the assumption of slight compressibility. Quasi-incompressible fluid formulations have been widely used for FSI simulation both in the finite element method (FEM) [4–7] and the smooth particle hydrodynamics (SPH) contexts [8–11].

For designing monolithic FSI solvers (i.e. the ones that rely on the solution of the coupled problem using a single discrete system) quasi-incompressible formulations are particularly benefitial as they allow for pressure condensation in the fluid domain while maintaining the velocity/pressure coupling. This results in (a) better monolithic system conditioning due to elimination of the different variables scales (b) simplicity of coupling with the structure when both sub-domains are described using the same primary variable (displacement or velocity). In such case elements of the fluid and the structure simply share the same degrees of freedom at a contact node. Thus, a fluid–structure problem can be solved very similarly to a single-material one. Of course, in such approaches the use of fitting interface meshes is obligatory.

Under the assumption of quasi-incompressibility the pressure is related to the kinematic field (velocity or displacement) via a constitutive equation involving the compressibility constant, also called the *bulk modulus*. For high values of the bulk modulus quasi-incompressible formulations provide an acceptable approximation of the incompressible behavior. The bulk modulus must be sufficiently high to conserve mass in a satisfactory way and introduce the sound propagation speed at least 1 order of magnitude higher than the expected velocity of the bulk flow. For FSI problems the one can update the fluid pressure in the coupling step using the constitutive relation ensuring that the pressure

*E-mail address:* pryzhakov@cimne.upc.edu
[1] http://www.cimne.com

http://dx.doi.org/10.1016/j.rimni.2015.09.002
0213-1315/

accounts for the motion of the structure. As we shall see further, this pressure update does not involve linear system solution and is therefore computationally cheap.

Quasi-incompressible fluid formulation based on linear velocity-constant pressure finite elements was proposed in [4]. Element-wise constant pressure formulation facilitated pressure condensation at an elemental level, i.e. prior to assembly. This facilitated solving the entire fluid–solid problem using a unified approach with the velocity being the only primary variable. However, the drawback of the formulation was the volumetric locking phenomenon (well-known in constant-pressure elements) that manifested already at moderately high values of the bulk modulus. On the other hand, low values of bulk modulus led to poor approximations of the incompressible behavior. In [6,12] an alternative based on linear pressure interpolations was proposed. The formulation exhibited superior behavior in terms of volumetric locking. Nonetheless, the computational cost of the solver increased due to the impossibility of condensing pressure elementally when using linear pressure approximation. The global pressure condensation procedure had to be introduced. Moreover, when approaching incompressibility limit the pressure instability problems (inf-sup instability [13]) due to using equal order velocity-pressure interpolations manifested. In general, the ambiguity of the quasi-incompressible or penalty approaches can be expressed as follows: the compressibility constant must be large enough to approximate the incompressibility accurately, but at the same time it must be small enough not to lead to "stiff" governing systems. An improvement with respect to modeling the incompressible behavior can be found in [14], where an idea of combining the above-mentioned quasi-incompressible approaches with the fractional step strategy was proposed. The method consisted in using the momentum equation of a quasi-incompressible fluid as a prediction (fractional momentum equation). The subsequent solution of the pressure Poisson's equation and the momentum equation correction led to the truly incompressible solution. The method allowed for using relatively low values of bulk modulus in the quasi-incompressible momentum equation, since the truly incompressible solution was recovered at the correction step. The necessity of the computationally expensive global pressure condensation inherited from [6] due to the use of linear pressure interpolations defined the main drawback of the methodology.

In the present work we propose one further improvement of the methods' family developed in [4,6,14]. Following the idea of combining the quasi-incompressible prediction with the fractional step method, we propose to use the approximation of the volumetric term in the tangent matrix that allows for computationally efficient elemental pressure condensation, defining a major advantage in comparison with [14]. We also introduce the fluid–structure interaction coupling strategy where the modified fractional momentum equation is solved together with the momentum equation of the structure monolithically, while the subsequent "incompressible correction" steps are carried out in the fluid domain exclusively.

The paper is organized as follows. We first introduce the modified fractional momentum equation using the quasi-incompressibility assumption. An approximate linearization of the volumetric term is introduced. Correction steps ensuring truly incompressible solution are specified next. Then the solution procedure for the FSI problems is outlined. The paper concludes with two challenging FSI benchmark examples.

## 2. Numerical model

### 2.1. Governing equations for the fluid

Let us consider a fluid domain $\Omega$ with the fixed boundary $\Gamma_d$. We shall consider viscous incompressible Newtonian fluids being the most common in the majority of the engineering applications. The governing system are therefore the Navier–Stokes equations equipped with the incompressibility condition. These can be written as:

$$\rho\frac{\partial \mathbf{v}}{\partial t} + \nabla p + \rho\mathbf{v}\cdot\nabla\mathbf{v} - \mu\nabla\cdot(2\epsilon(\mathbf{v})) = \rho\mathbf{g} \tag{1}$$

$$\nabla\cdot\mathbf{v} = 0 \tag{2}$$

where $\mathbf{v}$ is the velocity vector, $p$ the pressure, $t$ the time, $\mathbf{g}$ the body force, $\rho$ the density, $\mu$ the dynamic viscosity and $\epsilon = (\nabla\mathbf{v} + \nabla^T\mathbf{v})/2$ – the deviatoric strain rate.

At the fixed wall $\Gamma_d$, homogeneous Dirichlet boundary conditions are prescribed:

$$\mathbf{v} = 0 \qquad \text{at } \Gamma_d \tag{3}$$

#### 2.1.1. Finite element formulation

The equal order linear velocity/pressure interpolations over 3-noded triangles (2D) or 4-noded tetrahedra (3D) are used here for the space discretization of the governing equations Eqs. (1) and (2). We assume Backward Euler time integration scheme exclusively for the sake of simplicity. All the arguments presented in the paper are valid for any implicit time integration scheme. In the implementation carried out in this work the second order Newmark–Bossak scheme is used [6]. Being standard, the space and time discretization are not discussed here (see e.g. [15,16]). Pressure stabilization term is added due to the use of the equal order velocity-pressure formulation (Algebraic Sub-Grid Scales (ASGS) stabilization [17] is implemented here). Lagrangian description of the fluid is considered.

Given $\bar{\mathbf{v}}_n$ and $\bar{p}_n$ at $t_n$, the time discrete problem consists in finding $\bar{\mathbf{v}}_{n+1}$ and $\bar{p}_{n+1}$ at $t_{n+1}$ as the solution of

$$\mathbf{M}\frac{\bar{\mathbf{v}}_{n+1} - \bar{\mathbf{v}}_n}{\Delta t} + \mu\mathbf{L}\bar{\mathbf{v}}_{n+1} + \mathbf{G}\bar{p}_{n+1} = \bar{\mathbf{F}} \tag{4}$$

$$\mathbf{D}\bar{\mathbf{v}}_{n+1} + \mathbf{S}\bar{p}_{n+1} = 0 \tag{5}$$

where $\mathbf{M}$, $\mathbf{L}$, $\mathbf{G}$ and $\mathbf{S}$ are the mass, the Laplacian, the gradient and the stabilization matrices, respectively. $\bar{\mathbf{v}}$ and $\bar{p}$ are the velocity and pressure, respectively, and $\bar{\mathbf{F}}$ is the body force vector. Subindices indicate the time step. Note the absence of the convective term due to the use of the Lagrangian kinematic framework.

The matrices and vectors are assembled from the elemental contributions defined as

$$\mathbf{M} = \rho\int_{\Omega_e}\mathrm{N}\mathrm{N}^T\mathrm{d}\Omega \tag{6}$$

$$\mathbf{L} = \int_{\Omega_e}\nabla\mathbf{N}\nabla\mathbf{N}^T\mathrm{d}\Omega \tag{7}$$

$$\mathbf{G} = -\int_{\Omega_e}\nabla\mathbf{N}\mathbf{N}\mathrm{d}\Omega \tag{8}$$

$$\bar{\mathbf{F}} = \int_{\Omega_e}\mathbf{N}\rho\mathbf{g}\mathrm{d}\Omega \tag{9}$$

$$\mathbf{S} = \int_{\Omega_e}(\nabla\mathbf{N})\,\tau\left(\frac{\rho}{\Delta t}\mathbf{N}\right)\mathrm{d}\Omega \tag{10}$$

$$\mathbf{D} = -\mathbf{G}^T \tag{11}$$

$\mathbf{N}$ stands for the vector of standard linear FE shape functions, $\Omega_e$ is the element integration domain, $\tau$ is an algorithmic stabilization coefficient defined as $\tau = (((2||\bar{\mathbf{v}}||)/h) + (4\nu/h^2))^{-1}$, where $h$ is the element size. Note also that the discrete operators given by Eqs. (6)–(11) correspond to the unknown current configuration $\mathbf{X}_{n+1}$

Please cite this article in press as: P. Ryzhakov, A modified fractional step method for fluid–structure interaction problems, Rev. int. métodos numér. cálc. diseño ing. 2016. http://dx.doi.org/10.1016/j.rimni.2015.09.002