Advances in Engineering Software 000 (2016) 1–12

Contents lists available at ScienceDirect

Advances in Engineering Software

journal homepage: www.elsevier.com/locate/advengsoft

Numerical simulation of fluid-structure interactions with stabilized finite element method

Petr Sváček

Czech Technical University in Prague, Faculty of Mechanical Engineering, Department of Technical Mathematics, Karlovo nám. 13, 121 35 Praha 2, Czech Republic

ARTICLE INFO

Article history:
Received 23 March 2016
Accepted 31 August 2016
Available online xxx

Keywords:
Finite element method
Aeroelasticity
ALE method

ABSTRACT

In this paper the problem of numerical approximation of an interaction of the incompressible viscous flow with a vibrating airfoil is considered. The attention is paid to the practical implementation of the finite element based solver. The underlying mathematical model is based on the incompressible Navier–Stokes equations coupled with the continuity equation. The arbitrary Lagrangian–Eulerian method is used to treat the time dependent domain. The weak formulation of the Navier–Stokes equation is introduced and several important issues are addressed. The numerical method based on the fully stabilized finite element method is described in details. The numerical results are shown. The reliability of the method is discussed.

1. Introduction

Numerical simulation of problems in science and technology has become important in many scientific and engineering applications as e.g. turbomachinery, aerospace engineering, biomechanics, etc. In the last decades the numerical methods are being used also for approximation of fluid-structure interaction (FSI). FSIs play a central role in aerospace engineering and many other fields like civil, mechanical and biomedical engineering, see e.g. [12,25,39].

In technical practice usually the prediction of the aeroelastic instability as flutter is important in the design of aircraft, see [32,48]. Typically the aeroelastic stability is investigated with the aid of a linearized approach (asymptotic stability), see [12]. However, the asymptotic stability is just necessary condition to guarantee safety. The transient growth induced by external excitation (as gust loads) can lead to structural failure eventhough the system is aeroelastically stable, see [26,49]. In [3] the combined aeroelastic behaviour and gust response of a flexible aerofoil is explored theoretically. In [3,44] the FSI problem of gust response of a flexible typical section is investigated in term of both high- and low-fidelity simulations. The use of accurate aeroelastic simulations is particularly attractive because it reduces the development risk, the number of experiments and the possibile design modifications. However, the computational effort associated with high fidelity aeroelastic models currently precludes their direct use in industry, see [50]. Acceleration of time-accurate high fidelity aeroelastic simulation algorithms has therefore become an active area of research.

The most common solution strategy is the so-called partitioned approach, which de-couples the original FSI problem and uses specialized solvers for each sub-problem (see [7,25]). The coupling is then enforced at the fluid-solid interface by suitable interface conditions. In [7] an efficient loosely-coupled partitioned scheme for fluid-structure interaction is presented. The algorithm is tested on a proposed benchmark problem of an incompressible, viscous fluid interacting with a structure composed of two layers. An energy estimate associated with unconditional stability is derived for the fully nonlinear FSI problem defined on moving domains. In [29] the kinematic splitting algorithm for fluid-structure interaction classified as loosely-coupled scheme is considered. However, the explicit partitioned algorithms can exhibit instabilities due to the poor fluid-solid coupling particularly in the case of large displacements, see [20].

One possibility how to overcome this instability is to use monolithic approach, which treat the solid and fluid regions as a single continuum. Although these algorithms exhibit good stability properties, its applicability is limited due to the fact that they are also CPU-time expensive, see [18,35]. In order to reduce such a computational cost several strategies can be used. Let us mention the monolithic iterative solvers with the velocity-pressure splitting preconditioners, see [1,23,30]. The other possibility is to use partitioned strongly-coupled approach, which is usually more expensive but many times unavoidable since loosely coupled algorithms yield unacceptable accuracy. This motivates the development of more efficient partitioned strong coupling algorithms. Paper [4]

E-mail address: petr.svacek@fs.cvut.cz, psvacek@gmail.com

http://dx.doi.org/10.1016/j.advengsoft.2016.08.012

Please cite this article as: P. Sváček, Numerical simulation of fluid-structure interactions with stabilized finite element method, Advances in Engineering Software (2016), http://dx.doi.org/10.1016/j.advengsoft.2016.08.012

describe partitioned procedures for fluid-structure interaction problems, based on Robin-type transmission conditions.

Another approach, how to efficiently solve FSI problems is shown in [43] based on the use of super-computers and a parallel algorithm for solution of the flow problem. Here, the massively separated airflow in human vocal folds during phonation is approximated. The understanding of the air flow is fundamental for the understanding and modeling of the aeroacoustic processes involved in human voice production. In this case it requires CFD simulations on large 3D dynamic meshes. The 3D flow model past vibrating vocal folds is solved by cell-centered finite volume method on large dynamic meshes with geometries modeling convergent and divergent glottis.

Let us also mention the FSI solvers based on the projection method, see [9,37]. The main drawback of such a schemes is that the projection method generates boundary errors, which can badly influence the solution of FSI problems, see [10,23]. The projection methods however can be very efficient for large scale time-dependent numerical simulations (see [24]). On the other hand the application of projection method can be difficult in the case of the finite element solution with an advanced stabilization and on anisotropically refined meshes resolving e.g. the boundary layers.

The finite element method applied for numerical simulation of incompressible flow problem must overcome several sources of instabilities. One instability is caused by the incompatibility of the pressure and velocity pairs of finite elements, cf. [22,42]. The other instability is due to the dominating convection terms, see [40]. There is another less well-studied instability source in the Galerkin discretization method related to a possible poor resolution of pressure, see [21,27,28]. Paper [38] revisits the definition of the stabilization parameter in the finite element approximation of the convection-diffusion-reaction equation in the case of distorted meshes.

For the case of high Reynolds numbers the turbulence effects should be also taken into account. Several strategies can be used for its approximation: except the Direct Numerical Simulations (DNS) approach, the Large Eddy Simulation (LES) can be applied, which simulates the coarser scales and models the finer, see [41]. The already mention finite element stabilization methods canbe related to the turbulence modelling, see [2]. See also [36], where the grad-div stabilization for the incompressible Navier–Stokes finite element approximations is discussed as either variational multiscale approach or as a stabilization procedure of least-square type. Both DNS and LES are still too demanding in engineering practice, where the mostly applied method is the Reynolds Averaged (RANS) approach, see [51]. Although the turbulence in FSI problems can be addressed also by LES models (see e.g. [31]), still the RANS approach is more typical (see [13,16]).

The other thing is that for approximation of FSI problem the applied numerical method should be able to treat the moving domain/meshes. The most popular method is the arbitrary Lagrangian–Eulerian(ALE) method, see [34]. There are several possibilities how to apply the finite element method, see [46]. Usually in order to provide higher order accuracy or stability some additional assumptions on the ALE are required, see [8,19]. Particularly the so-called geometrical conservation law is important, see [14].

In this paper the numerical analysis of an interaction of the incompressible viscous flow with a vibrating airfoil is considered, see e.g. [45,46] or [15]. The attention is paid to an approximation on moving meshes with the aid of suitable arbitrary Lagrangian–Eulerian (ALE) method. For the case of high Reynolds numbers anisotropically refined mesh is used and the choice of the stabilizing parameters is discussed. Two finite element pairs are considered for the numerical approximation and the stabilization procedure for both of them is presented based on the so-called residual based stabilization, cf. [21]. A benchmark problem of mutual interaction of the airflow and a flexibly supported airfoil with two degrees of freedom (2DOF) (the bending and the torsion modes) is considered. The problem is solved using a partitioned scheme with strongly coupled/loosely-coupled scheme. The problem of the solution of the non-linear flow problem per-each time step is discussed. The paper is organized as follows: first the mathematical model is presented, then the weak form suitable for application of the finite element method is introduced, then the numerical approximation is presented and the numerical results are shown.

2. Mathematical model

The flow of the incompressible viscous fluid in the computational domain $\Omega_t \subset \mathbb{R}^2$ is governed by the Navier–Stokes system of equations

$$\frac{\partial \boldsymbol{u}}{\partial t} + \nabla \cdot (\boldsymbol{u} \otimes \boldsymbol{u}) = \nabla \cdot \boldsymbol{\sigma}, \qquad \nabla \cdot \boldsymbol{u} = 0,$$

where $\boldsymbol{u} = \boldsymbol{u}(x,t)$ denotes the velocity vector, $\boldsymbol{u} = (u_1, u_2)$, $\boldsymbol{\sigma}$ denotes the Cauchy stress tensor with the components σ_{ij} given by $\boldsymbol{\sigma} = -p\mathbb{I} + 2\nu \boldsymbol{S}(\boldsymbol{u})$ and $\boldsymbol{S}(\boldsymbol{u}) = \frac{1}{2}(\nabla \boldsymbol{u} + \nabla^T \boldsymbol{u})$. Here $p = p(x,t)$ denotes the kinematic pressure and ν is the constant kinematic viscosity (i.e. the viscosity divided by the constant fluid density ρ). System of Eq. (1) is equipped with the initial condition

$$\mathbf{u}(x,0) = \mathbf{u}^0(x) \qquad x \in \Omega, \tag{1}$$

and with boundary conditions

$$\begin{aligned} &\text{a)}\ \boldsymbol{u} = \boldsymbol{u}_D && \text{on } \Gamma_I,\\ &\text{b)}\ \boldsymbol{u} = \boldsymbol{w}_D && \text{on } \Gamma_{Wt},\\ &\text{c)}\ -\boldsymbol{\sigma}\cdot\boldsymbol{n} + \frac{1}{2}(\boldsymbol{u}\cdot\boldsymbol{n})^-\boldsymbol{u} = 0 && \text{on } \Gamma_O, \end{aligned} \tag{2}$$

where $\mathbf{n}$ denotes the unit outward normal vector to the Lipschitz continuous boundary $\partial\Omega$ and $(\alpha)^- = \min(0,\alpha)$. The boundary condition (2c) is a modification of the so-called 'do-nothing' boundary condition, cf. [6] or [5]. In order to discretize Eq. (1), the so-called arbitrary Lagrangian–Eulerian (ALE) method is applied, cf. [34].

2.1. ALE method

The ALE method is usually introduced with the aid of the (ALE) mapping $\mathcal{A} = \mathcal{A}(\xi,t) = \mathcal{A}_t(\xi)$ of a reference configuration Ω_0 onto the current configuration Ω_t. For what follows the following assumption (see also [33]) about ALE mapping is made: the mapping $\mathcal{A}$ has continuous second order partial derivatives *and moreover* it is continuously differentiable mapping of Ω_0 onto Ω_t with the continuous bounded Jacobian $\mathcal{J}$

$$\mathcal{J}(x,t) = \hat{\mathcal{J}}(\xi,t) = \det \frac{\partial x}{\partial \xi}, \text{ where } x = \mathcal{A}_t(\xi).$$

Using the assumptions the *domain velocity* $\boldsymbol{w}_D$ is defined both on the reference domain Ω_0 as well as on Ω_t by

$$\widehat{\mathbf{w}}_D(\xi,t) = \frac{\partial \mathcal{A}}{\partial t}(\xi,t), \qquad \boldsymbol{w}_D(x,t) = \widehat{\mathbf{w}}_D(\xi,t), \qquad \forall x = \mathcal{A}(\xi,t) \in \Omega_t.$$

Similary any function $f(x,t)$ defined for all $x \in \Omega_t$ and $t \in (0,T)$ can be transformed on Ω_0 as the function $\hat{f}(\xi,t)$ defined for any $\xi \in \Omega_0$ and $t \in (0,T)$ by

$$\hat{f}(\xi,t) = f(\mathcal{A}(\xi,t),t).$$

Further, the ALE derivative denoted by the symbol $D^{\mathcal{A}}/Dt$ is then the time derivative with respect to the reference configuration, i.e.

$$\frac{D^{\mathcal{A}} f}{Dt}(x,t) = \frac{\partial \hat{f}}{\partial t}(\xi,t), \text{ where } x = \mathcal{A}_t(\xi).$$

Please cite this article as: P. Sváček, Numerical simulation of fluid-structure interactions with stabilized finite element method, Advances in Engineering Software (2016), http://dx.doi.org/10.1016/j.advengsoft.2016.08.012