COMPUTATIONAL MECHANICS 2018
34th conference with international participation
Srní
October 31 - November 2, 2018

Finite element method application for fluid structure interactions: Mathematical background and implementation

P. Sváček[a]

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

In this paper the mathematical modelling of fluid-structure interaction problems is addressed particularly with the interest paid to the biomechanics of human voice. The attention is paid to the precise approximation of the fluid flow, particularly in the glottal part, with the aid of the numerical approximation of the Navier–Stokes equations. This problem is even more complicated in the context of the voice creation process, e.g., by the glottal gap closing or by the presence of the contact problem. In this case one need to take into account not only a significant mesh deformation but also the influence of the prescribed artificial inlet/outlet boundary conditions.

We shall focus particularly on several implementation aspects of the finite element method used for the solution of the fluid-structure interaction (FSI) problem. The practical realization of the finite element method shall be discussed based on the variational formulation of the underlying problems. The FSI problem consists of the solution of the fluid flow, the structure deformation and the mesh displacement problems. For the fluid flow the moving mesh should be taken into account which leeds to the Navier-Stokes system of equations on the computational domain Ω_t^f written in the ALE form

$$\rho \frac{D^{\mathcal{A}} \boldsymbol{v}}{Dt} + \rho((\boldsymbol{v} - \boldsymbol{w}_D) \cdot \nabla)\boldsymbol{v} = \operatorname{div} \boldsymbol{\tau}^f, \qquad \nabla \cdot \boldsymbol{v} = 0, \tag{1}$$

where $\boldsymbol{v}$ is the fluid velocity vector, ρ is the constant fluid density, and $\boldsymbol{\tau}^f$ is the fluid stress tensor given by $\boldsymbol{\tau}^f = -p\mathbb{I} + 2\mu\mathbb{D}(\boldsymbol{v})$. Here, p is the pressure, $\mu > 0$ is the constant fluid viscosity and $\mathbb{D}(\boldsymbol{v}) = \frac{1}{2}(\nabla\boldsymbol{v} + (\nabla\boldsymbol{v})^T)$. This system of equations is equipped with initial and boundary conditions. The deformation of the domain is result of the mutual interaction with a structural model. For the structure deformation as well as the mesh deformation, e.g., the linear elastic model can be used.

In order to describe the details of the application of the finite element method for solution of (stationary) boundary value problems, we shall use a context of an abstract variational formulation of finding $u \in \mathcal{V}$ such that

$$a(u, v) = L(v) \qquad \text{for all } v \in \mathcal{V}, \tag{2}$$

where $\mathcal{V}$ is a Banach space and $a(u, v)$, $L(v)$ are in general forms on $\mathcal{V}$. This formulation represents a weak formulation of the above mentioned specific boundary value problems and contains the information about the solved equations as well as the applied boundary conditions. The problem (2) can be formally discretized by introducing a finite dimensional finite element space $\mathcal{V}_h$ constructed over a triangulation of the computational domain. The application of the finite element method then reads: Find the approximation solution u_h such that

$$a(u_h, v_h) = L(v_h) \qquad \text{for all } v_h. \tag{3}$$

In the case when a is bi-linear and L is linear, this system represents a system of linear equations.

Such an approach is used in various books or textbooks about the finite element method to support the theoretical analysis of the finite element method, see, e.g., [1]. In this paper it is shown that such an approach is suitable also for the implementation purposes, see also [3, 4]. The program is written in object oriented C language, see [2].

For the implementation purposes, it is usually used that the forms a and L in (2) are given by integrals which together with the use of the finite element space $\mathcal{V}_h$ defined over an triangulation $\mathcal{T}_h$ gives

$$a(U,V) = \sum_{K\in\mathcal{T}_h} \int_K \omega_a(x,U,V)dx + \sum_{S\in\mathcal{S}_h} \gamma_a(x,U,V)dS \tag{4}$$

and

$$L(V) = \sum_{K\in\mathcal{T}_h} \int_K \omega_L(x,V)dx + \sum_{S\in\mathcal{S}_h} \gamma_L(x,V)dS, \tag{5}$$

where $\mathcal{S}_h$ denotes the set of all boundary edges of elements adjacent to the boundary Γ, ω_a, ω_L are expressions (or better operator) in u, v linearly dependent on v.

Using this the Galerkin formulation can be defined in the program by definition of methods of the object scalar problem

```
typedef struct {
    void   (*getBndrCnd)(bpoint *P, short *isfixed, double *val);
    double (*aformdx)(point *P, scalar *u, scalar *v);
    double (*Lformdx)(point *P, scalar *v);
    double (*aformdS)(bpoint *P, scalar *u, scalar *v);
    double (*LformdS)(bpoint *P, scalar *v);
} scalarproblem;
```

where `aformdx`, `aformdS` corresponds to terms ω_a, γ_a, and `Lformdx`, `LformdS` corresponds to terms ω_L, γ_L. The `scalarproblem` is then used for the finite element object `fespace` based on the mesh (triangulation) `gmesh`. This corresponds well to the mathematical construction of the finite element space over a triangulation τ_h.

Acknowledgment

This work was supported by the *Czech Science Foundation* under the *Grant No. 16 - 01246S*.

References

[1] Ciarlet, P.G., The finite element methods for elliptic problems, North-Holland Publishing, 1979.

[2] Schreiner, A., Object-oriented programming with ANSI-C, e-book available from URL <www.cs.rit.edu/ ats/books/ooc.pdf>, 2011.

[3] Fenics project, URL <http://www.fenicsproject.org>.

[4] Hermes (higher-order finite element system), URL <http://hpfem.org/hermes>.