

MULTILEVEL NON-CONFORMING FINITE ELEMENT METHODS FOR COUPLED FLUID-STRUCTURE INTERACTIONS

E. AULISA, S. GARCIA, E. SWIM, AND P. SESHAIYER

Abstract. Computational mathematics is constantly evolving to develop novel techniques for solving coupled processes that arise in multi-disciplinary applications. Often such analysis may be accomplished by efficient techniques which involve partitioning the global domain (on which the coupled process evolves) into several sub-domains on each of which local problems are solved. The solution to the global problem is then constructed by suitably piecing together solutions obtained locally from independently modeled sub-domains. In this paper we develop a multilevel computational approach for coupled fluid-structure interaction problems. The method relies on computing coupled solutions over different sub-domains with different multigrid levels. Numerical results for the reliability of the schemes introduced are also presented.

Key words. finite element methods, fluid-structure interaction, Arbitrary Lagrangian-Eulerian formulation, non-conforming, multilevel.

1. Introduction

The past few decades have seen significant advances in the development of computational methods to obtain efficient solutions to complex coupled systems that consist of interactions between functionally distinct components. Coupled with advances in finite element methods, these methods have provided new algorithms for large scale simulations [13, 14]. Often in such methods, the interface continuity between solutions in independently modeled sub-domains is enforced weakly via Lagrange multipliers that are defined on the interface. The *mortar finite element method* is one example of such a technique (see e.g. [8, 4, 5, 6, 13, 15, 16, 7, 17] and references therein) where precise choices are described for the two fields (the interior solution variable and the interface Lagrange multiplier) to ensure *stability*. One can also employ more general three-field methods, where one field represents the solution variable on the interface and is modeled independently from the interior solution variables on either side of the interface. Here, two Lagrange multipliers will be required in order to enforce continuity between each interior variable and the interface variable. In either case, Lagrange multiplier methods allow for optimal rates of convergence along the interface between distinct components of a coupled system.

In recent years flexible multilevel multigrid methods have been introduced [20, 19, 12, 1, 2, 3], whose solvers are based on the iterative solution of several problems over smaller domains. These techniques allow solutions to be computed at the element level and also help us to achieve proper accuracy, load balancing and computational efficiency. Such novel techniques provide motivation for us to develop fast and efficient algorithms to solve complex fluid-structure interaction (FSI) problems [18, 3].

Received by the editors May 2011 and, in revised form, June 2012.

2000 *Mathematics Subject Classification.* 65N30, 65N55.

Corresponding Author: Padmanabhan Seshaiyer, Department of Mathematical Sciences, George Mason University, Fairfax, VA 22030. Email: pseshaiy@gmu.edu.

It is well known that the one of the most difficult parts of numerically approximating the fluid-structure coupling arises from the fact that the structural equations are usually formulated with material (Lagrangian) coordinates, while the fluid equations are typically written using spatial (Eulerian) coordinates. This is important since the nodes on the fluid mesh are attached to the surface of the structure and hence should move with the displacement of the structure. Therefore, a straightforward approach to the solution of the coupled fluid-structure dynamic equations requires moving at each time step at least the portions of the fluid grid that are close to the moving structure. This can be appropriate for small displacements of the structure but may lead to severe grid distortions when the structure undergoes large motion. Several different approaches have emerged as an alternative to partial re-gridding in transient computations, one of which is the Arbitrary Lagrangian Eulerian (ALE) formulation [9, 11, 18].

When constructing a numerical method for time-dependent coupled systems that involve a moving boundary, differences in scale between the solutions in each sub-domain should be incorporated into the approximation. For example, in a fluid-structure interaction where the geometry of the problem evolves due to the deformation of an elastic structure, the magnitude of the strain rate of the solid body may be much smaller than the velocity of molecules in the fluid region. Non-conforming finite element methods offer a promising framework for this situation since the scale of the computational grid and degree of polynomial approximation can be refined in each sub-domain independently. In this setting, each sub-domain is independently partitioned by regular families of meshes, where the intersection of any two distinct elements is either a vertex, an edge, or an empty set, and a restriction on the ratio between edges and diameters of the elements prevents them from becoming arbitrarily thin. This approach will avoid the necessity of creating transition elements between the sub-domains, which often lead to solution inaccuracy due to severe distortions, especially in cases where an initial numerical grid is allowed to move in response to deformation of the original domain. Our objective will be to develop a non-conforming finite element methodology to couple a Lagrangian model describing a structure interacting with a fluid that is described by the ALE strategy in order to simulate a full unsteady physical phenomenon.

The outline of the paper is as follows. Section 2 introduces a model fluid-structure interaction problem and presents a brief background on the methods that are employed to accomplish coupling. The mathematical formulation of the non-conforming technique is illustrated on a one-dimensional problem for simplicity. Numerical experiments for the one-dimensional model problem are presented that indicate the robustness of the method introduced. Section 3 presents the extension of the problem to higher dimensions and presents the solution methodology as well as a numerical validation through a model problem involving a beam and fluid interaction. Finally, section 4 presents discussions and future research.

2. A One-Dimensional Model Problem and Governing Equations

For simplicity, let us now describe the mathematical formulation and solution methodology for a fully coupled system of equations governing the interaction between a fluid and a structure in a one-dimensional setting. We will present both the continuous problem and a discrete approximation of the model that incorporates an ALE formulation, allowing the numerical grid in the fluid region to move along with the interface between the two sub-domains. Such models can help to provide insight into fluid-structure interaction effects for a totally or partially submerged

body in a flowing liquid that are typical problems which are of great relevance in civil and offshore engineering and naval architecture among many other fields.

Assume an initial configuration in which a fluid occupies the interval $(0, 1)$ and an elastic structure occupies the interval $(1, 2)$. As the fluid flow deforms the adjacent solid and the interface between the two moves, we will denote by $\gamma(t)$ the position of this interface at any time $t \geq 0$.

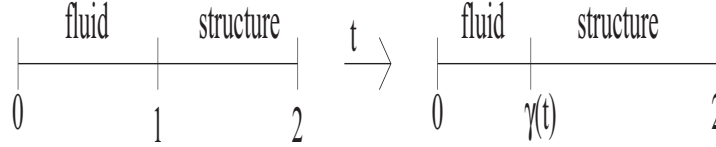


FIGURE 1. Evolution of a one-dimensional domain

The position of the interface between the two media must then satisfy

$$\gamma(t) = 1 + d(t, 1), \quad t \geq 0,$$

where $d(t, 1)$ represents the displacement of the left boundary of the elastic structure at time $t = 1$ (See Figure 1). For each point $x \in (0, \gamma(t))$, we model the velocity u and pressure p using a generalization of models employed by [10, 18]

$$(1) \quad \frac{\partial u}{\partial t}(t, x) - \alpha \frac{\partial^2 u}{\partial x^2}(t, x) + (1 + \beta)u \frac{\partial u}{\partial x}(t, x) + \varepsilon \frac{\partial p}{\partial x}(t, x) = f(t, x).$$

Here, $\alpha > 0$ is a parameter depending on the kinematic viscosity ν of the fluid. The latter is a ratio of the absolute viscosity to the mass density. The constants $\beta \in [0, \frac{1}{2}]$ and $\varepsilon \geq 0$ vary depending on the material properties of the fluid, *e.g.*, compressibility. For example, the parameter choices $\alpha = \frac{4}{3}\nu$, $\beta = 0$, $\varepsilon = \frac{1}{\rho}$, where $\rho = \rho(t)$, are used for modeling monatomic gases. Another example is a modified version of Burgers' equation, given by parameter choices of $\alpha = \nu$, $\beta = \frac{1}{2}$, $\varepsilon = 0$. For stability considerations [18] let us enforce,

$$(2) \quad \left(\beta - \frac{1}{2}\right) u \frac{\partial u}{\partial x}(t, x) + \varepsilon \frac{\partial p}{\partial x}(t, x) = 0.$$

At the left endpoint, the boundary condition $u(t, 0) = 0$ is enforced for all $t \geq 0$. Let the displacement d of any point in the adjacent structure from its initial position be modeled by,

$$(3) \quad \frac{\partial^2 d}{\partial t^2}(t, x) - \mu \frac{\partial^2 d}{\partial x^2}(t, x) = g(t, x), \quad x \in (1, 2),$$

where $\mu \geq 0$, $d(t, 2) = 0$ for all $t \geq 0$ and $d(0, x) = 0$, $x \in (1, 2)$. Here we are using the wave equation as a simplified version of linear elasticity in one dimension, set in a Lagrangian formulation. Thus we should assume that deformations of the elastic structure are small relative to the length scale.

At the interface between fluid and structure, we enforce continuity of velocities,

$$u(t, \gamma(t)) = \frac{\partial d}{\partial t}(t, 1)$$

and the continuity of the fluxes,

$$\alpha \frac{\partial u}{\partial x}(t, \gamma(t)) = \mu \frac{\partial d}{\partial x}(t, 1).$$

For the changing nature of the fluid domain, we will rewrite the fluid equation (1) using an ALE formulation that avoids extreme mesh distortion near the interface.

To do this, we will move the numerical grid independently of the fluid velocity on the fluid domain. Let us define the grid velocity w at any point $x \in (0, \gamma(t))$ by

$$w(t, x) = \frac{x}{\gamma(t)} \dot{\gamma}(t).$$

Note that w satisfies the conditions

$$w(t, \gamma(t)) = \dot{\gamma}(t) \quad w(t, 0) = 0.$$

Thus the grid velocity is consistent with the velocity of the fluid at the endpoints of the fluid domain. Additionally, note that we assume that $0 \leq \gamma(t) \leq 2$ for all time. We now define a family of characteristic curves, $x_s(t, \xi)$, on which the fluid velocity is independent of partial derivatives in other directions. Thus the characteristic curve associated with w must satisfy

$$\frac{dx_s}{dt}(t, \xi) = w(t, x_s(t, \xi))$$

and $x_s(s, \xi) = \xi$ for all $\xi \in (0, \gamma(s))$. Hence,

$$x_s(t, \xi) = \xi \frac{\gamma(t)}{\gamma(s)}.$$

Let $v(t, \xi) = u(t, x_s(t, \xi))$, *i.e.*, let v be the fluid velocity along the characteristic curve $x_s(t, \xi)$. Then, applying the chain rule, (1) can be rewritten for values of $\xi \in (0, \gamma(s))$ as,

$$(4) \quad \frac{\partial v}{\partial t}(t, \xi) - \alpha \frac{\partial^2 u}{\partial x^2}(t, x_s) + [(1 + \beta)u - w] \frac{\partial u}{\partial x}(t, x_s) + \varepsilon \frac{\partial p}{\partial x}(t, x_s) = f(t, x_s)$$

where $x_s = x_s(t, \xi)$. Employing this ALE formulation for the fluid and an *implicit* formulation for the interface position, we now wish to construct a discrete variational form for our coupled system of equations (2)-(4). First, we choose a time step size $\Delta t > 0$ and let $t^n = n\Delta t$. For any function ϕ defined on the continuous domain $[0, 2] \times [0, \infty)$, we denote by ϕ^n the restriction of ϕ to the n^{th} time step, *i.e.* $\phi^n(x) = \phi(t^n, x)$. Additionally, we subdivide the initial computational domains, defined by the intervals $(0, 1)$ and $(1, 2)$, using regular partitions

$$\begin{aligned} \{0 &= x_1, x_2, \dots, x_M = 1\}, \\ \{1 &= \nu_1, \nu_2, \dots, \nu_P = 2\}. \end{aligned}$$

We then choose finite element spaces

$$V_F^n \subset \{\phi \in H^1(0, \Gamma^n) : \phi(0) = 0\}$$

and

$$V_S^n \subset \{\psi \in H^1(1, 2) : \psi(2) = 0\}$$

Given a basis for each of these spaces, we then wish to construct finite element approximations for u^n , p^n , and d^n at each successive time step, namely U^n , P^n , and D^n . We define the movement of the fluid partition using an approximation of w^n , given by

$$W^n(x) = \frac{x}{\Gamma^n} U^n(\Gamma^n) \quad \forall x \in (0, \Gamma^n),$$

where Γ^n is the approximate position of the interface between the fluid and structure at the n^{th} time step, *i.e.*

$$\Gamma^{n+1} = \Gamma^n + \Delta t U^n(\Gamma^n), \quad n = 0, 1, 2, \dots$$

and $\Gamma^0 = 1$. Additionally, we approximate the characteristic curve $x_s(t, \xi)$ between successive time steps t^n and t^{n+1} by

$$(5) \quad X^n(t) = X^n + (t - t^n)W^n(X^n) \quad \forall X^n \in (0, \Gamma^n), \quad t \in [t^n, t^{n+1}].$$

Thus, if we let

$$X^{n+1} = X^n + \Delta t W^n(X^n) = X^n(t^{n+1}),$$

then X^{n+1} represents the map of a mesh point X^n to the next time step along the characteristic curve defined by W^n .

Note that the partition on the structural domain (1, 2) remains fixed since at each time step we are approximating the displacement from the *initial* position. We then approximate $\partial_t u(t, x_s(t, \xi))$ by:

$$\frac{U^{n+1}(X^{n+1}) - U^n(X^n)}{\Delta t}$$

Additionally, we approximate $\partial_{tt}d(t, x)$ with a central differencing scheme:

$$\frac{D^{n+1} - 2D^n + D^{n-1}}{(\Delta t)^2}.$$

For $x \in (0, \Gamma^n)$, let

$$\bar{U}^{n+1}(x) = U^{n+1}(x + \Delta t W^n(x)).$$

Splitting the convective term in (4) and using forward and central differencing schemes yields a discrete system for \bar{U}^{n+1} , \bar{P}^{n+1} , and D^{n+1} :

$$(6) \quad \frac{1}{\Delta t}(\bar{U}^{n+1} - U^n) - \alpha \partial_{xx} \bar{U}^{n+1} + (U^n - W^n) \partial_x \bar{U}^{n+1} + \beta \bar{U}^{n+1} \partial_x U^n + \varepsilon \partial_x \bar{P}^{n+1} = \bar{f}^{n+1}$$

$$(7) \quad \left(\beta - \frac{1}{2}\right) \bar{U}^{n+1} \partial_x U^n + \varepsilon \partial_x \bar{P}^{n+1} = 0$$

$$(8) \quad \frac{1}{(\Delta t)^2} (D^{n+1} - 2D^n + D^{n-1}) - \mu \partial_{xx} D^{n+1} = g^{n+1}$$

$n = 0, 1, 2$, etc., where \bar{P}^{n+1} and \bar{f}^{n+1} are defined in the same way as \bar{U}^{n+1} . The discrete fluid and structure equations are coupled via the continuity constraints

$$\bar{U}^{n+1}(\Gamma^n) = \frac{1}{\Delta t} (D^{n+1} - D^n)(1),$$

$$\alpha \partial_x \bar{U}^{n+1}(\Gamma^n) = \mu \partial_x D^{n+1}(1),$$

and

$$\Gamma^{n+1} = 1 + D^{n+1}(1).$$

Note that continuity of velocity is only imposed using an *implicit* difference approximation for $\partial_t d(t, x)$ at the interface. This choice, along with the choice to use an *explicit* scheme to approximate the characteristic curve in equation (5), leads to stability requirements that are explored in detail in [10]. The primary restriction on the time step here is

$$\Delta t \leq \left(\frac{\epsilon(1 - \delta)}{C} \right)^{4/3},$$

where $\epsilon \in (0, 1)$ is small enough to ensure that $\varepsilon \leq C_0$ for a constant C_0 depending only on initial conditions and external applied forces, where

$$\|U^{m+1}\|_{L^2(0, \Gamma^{m+1})}^2 + \delta \alpha \Delta t \|\partial_x U^{m+1}\|_{L^2(0, \Gamma^{m+1})}^2 \leq C_0^2$$

and

$$\|D^{m+1}\|_{L^\infty(1, 2)} \leq C_0.$$

Moreover, $\delta \in (0, 1)$ is small enough to ensure that

$$\frac{|U^n(\Gamma^n)|}{\Gamma^n} \leq \frac{1 - \delta}{\Delta t},$$

and C is a constant depending only on data.

Next we multiply (6) and (7) by a test function $\phi \in V_F^n$ and (8) by a test function $\psi \in V_S^n$ and integrate over the intervals $(0, \Gamma^n)$ and $(1, 2)$, respectively, to obtain the following discrete variational problem for finding \bar{U}^{n+1} , \bar{P}^{n+1} , D^{n+1} , and Γ^{n+1} . For any test functions $\phi \in V_F^n$ and $\psi \in V_S^n$,

$$\begin{aligned} & (\bar{U}^{n+1}, \phi)_n - \alpha \Delta t \partial_x \bar{U}^{n+1}(\Gamma^n) + \alpha \Delta t (\partial_x \bar{U}^{n+1}, \partial_x \phi)_n + \epsilon \Delta t (\partial_x \bar{P}^{n+1}, \phi)_n \\ & + \Delta t ((U^n - W^n) \partial_x \bar{U}^{n+1}, \phi)_n + \beta \Delta t (\bar{U}^{n+1} \partial_x U^n, \phi)_n \\ & = (U^n, \phi)_n + \Delta t (\bar{f}^{n+1}, \phi)_n \\ & \left(\beta - \frac{1}{2} \right) (\bar{U}^{n+1} \partial_x U^n, \phi)_n + \epsilon (\partial_x \bar{P}^{n+1}, \phi)_n = 0 \\ & \frac{1}{\Delta t} (D^{n+1}, \psi) - \mu \Delta t \partial_x D^{n+1}(1) \psi(1) + \mu \Delta t (\partial_x D^{n+1}, \partial_x \psi) \\ & = \frac{1}{\Delta t} (2D^n - D^{n-1}, \psi) + \Delta t (g^{n+1}, \psi) \\ & \bar{U}^{n+1}(\Gamma^n) = \frac{1}{\Delta t} (D^{n+1} - D^n)(1) \\ & \alpha \partial_x \bar{U}^{n+1}(\Gamma^n) = \mu \partial_x D^{n+1}(1) \\ & \Gamma^{n+1} = 1 + D^{n+1}(1) \end{aligned}$$

where $(\cdot, \cdot)_n$ is the scalar product on $L^2(0, \Gamma^n)$, and $(\cdot, \cdot)_n$ is the scalar product on $L^2(1, 2)$. Note that the continuity of velocity is only imposed using an *implicit* difference approximation for $\partial_x d(t, x)$ at the interface. This method is implemented using the finite element method and the numerical results are discussed next.

2.1. Numerical Results. In this section we demonstrate the robustness and the performance of the non-conforming domain decomposition method introduced for the one-dimensional model problem. In particular we consider the model problem (1) with $\beta = 0.5$ which yields the modified Burgers' equation defined over the interval $(0, 1)$. This fluid equation is coupled to the wave equation representing the structure over the interval $(1, 2)$. The evolution of the computational domain is expected to be as in (1) and the results of implementing the finite element method for the coupled model fluid-structure interaction problem is presented next.

Figures 2 and 3 plot the displacement and the velocity the profiles for time $t = 0, 0.25, 0.5, 0.75, 1$. The boundary condition for the velocity of the fluid is prescribed to be $0.1 \sin(2\pi t)$. The results clearly illustrate the dynamics of the fluid-structure interaction problem as time evolves.

Next, we explored the influence of the parameter $\mu = \mu_s$ in the wave equation. The results of the displacement and velocity profiles are summarized in Figures 4 and 5. Our results indicate that as the parameter in the solid equations increase from 1 to 1000, the deflection decreases.

3. Extension to higher dimensions

Consider the coupled model problem where an isotropic, non-isothermal, Newtonian, incompressible fluid (defined in $\Omega_F(t)$) interacts with an elastic structure

(defined on $\Omega_S(t)$) through an energy equation (defined on $\Omega_S(t)$ and $\Omega_F(t)$):

$$(9) \quad \rho_f \left(\frac{\partial \vec{u}}{\partial t} + \vec{u} \cdot \nabla \vec{u} \right) = -\nabla p + \mu \Delta \vec{u} + \vec{f} + \vec{F}(T_F) \quad \text{in } \Omega_F(t)$$

$$(10) \quad \nabla \cdot \vec{u} = 0 \quad \text{in } \Omega_F(t)$$

$$(11) \quad \rho_f c_{p_f} \left(\frac{\partial T_F}{\partial t} + \vec{u} \cdot \nabla T_F \right) = K_f \Delta T_F \quad \text{in } \Omega_F(t)$$

$$(12) \quad \rho_s c_{p_s} \frac{\partial T_S}{\partial t} = K_s \Delta T_S \quad \text{in } \Omega_S(t)$$

$$(13) \quad \rho_s \frac{\partial^2 \vec{d}}{\partial t^2} + \nabla \cdot \tilde{\sigma}_S(\vec{d}) = \vec{G}(T_S) \quad \text{in } \Omega_S(t)$$

Here \vec{u}, p are the velocity and pressure unknowns of the fluid flow described by Navier-Stokes equation (9) along with the continuity equation (10); T_F, T_S are the unknown temperature in the energy equations (11)-(12) coupled to the fluid velocity through a convection term; \vec{d} is the unknown structural displacement (13). Note that $\vec{F}(T_F)$ introduces buoyancy forces in the fluid equation which allows the density of the fluid ρ_f to vary with the temperature T_F . Also note that \vec{d} depends on the temperature through the loading term $\vec{G}(T_S)$. All other parameters and material properties are specified along with the fluid body force \vec{f} . Moreover, the displacement \vec{d} also modifies the flow domain. Here $\tilde{\sigma}_S(\vec{d})$ is the linearized Cauchy stress tensor which is related to the displacement \vec{d} through an appropriate constitutive formulation. Here, we have considered a linear elastic behavior for the structure for simplicity and the proposed methodology may be modified suitably to accommodate more complex nonlinear (geometric and material) models that describe hyperelastic membrane materials. On the fluid-structure interface we enforce the continuity of the fluid and structural velocities ($\vec{u} = \frac{\partial \vec{d}}{\partial t}$), fluxes ($\tilde{\tau}^S(\vec{d}) \cdot \vec{n}_S = \tilde{\tau}^F(\vec{u}, p) \cdot \vec{n}_F$) and temperatures ($T_F = T_S$). Also, system (9)-(13) are complemented by appropriate boundary conditions governed by the physical

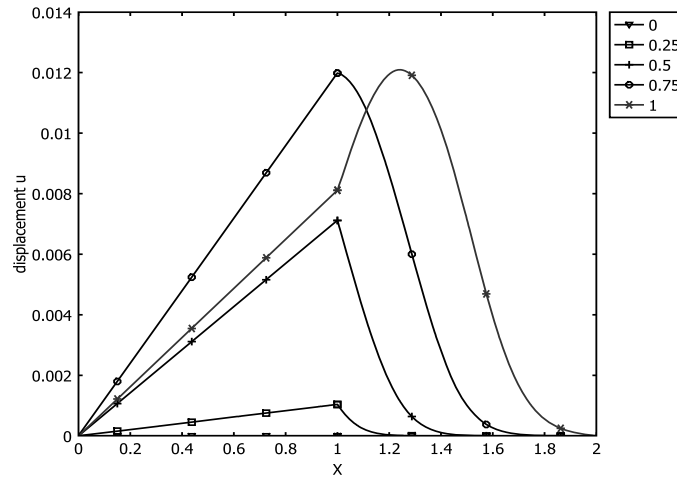


FIGURE 2. Displacement profiles for increasing time

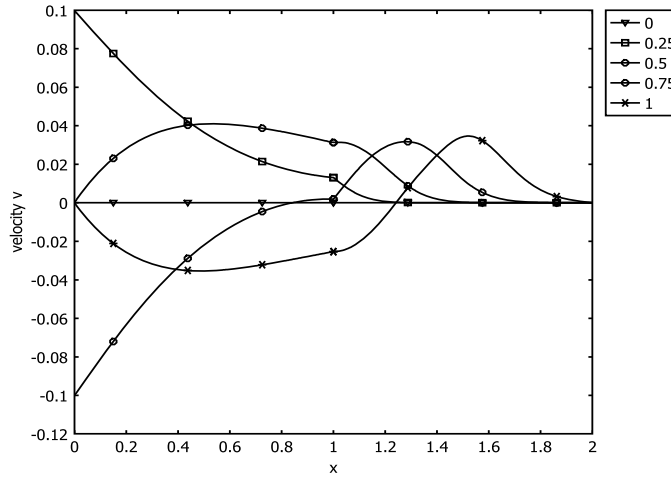
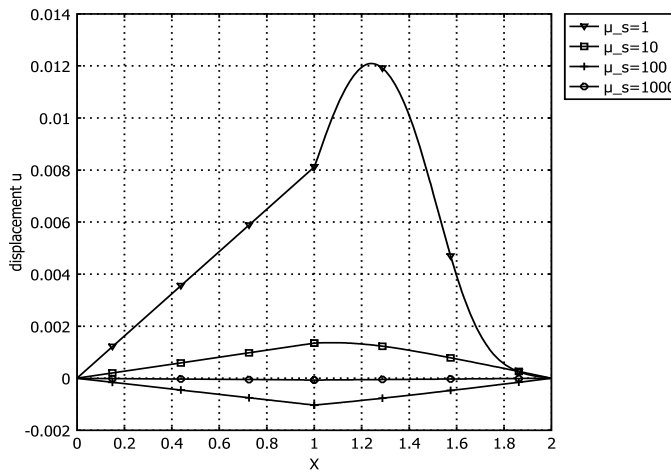


FIGURE 3. Velocity profiles for increasing time

FIGURE 4. Influence of the parameter $\mu = \mu_s$ to the FSI problem on the displacement

system. One can employ the mathematical formulation and solution methodology discussed previously for the one-dimensional (1-D) problem, to solve much more complex coupled problems in higher dimensions of the type (9)-(13).

3.1. Solution to the coupled problem. The solution to the associated fluid-structure-thermal interaction problem is then achieved via an iterative strategy, where the weak formulation of the coupled systems of equations (9)-(13) are solved separately and in succession, always using the latest information, until convergence is reached. An iterative multigrid solver is used for both the Navier-Stokes and the energy equation systems since the number of unknowns could be quite large.

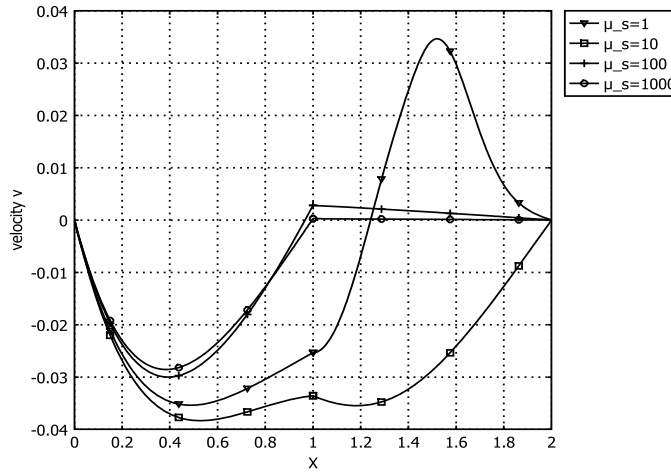


FIGURE 5. Influence of the parameter $\mu = \mu_s$ to the FSI problem on the velocity

For the solution of the beam equation a direct LU decomposition is used. The solution methodology employed is summarized in Figure 6. For the solution of the non-linear beam equation a direct LU decomposition is used. At each iteration, the linearized Navier-Stokes system is assembled, using the latest updated value of the temperature and the latest updated value of the grid velocity in the nonlinear term. In the nonlinear term, the first of the two velocity vectors is considered explicitly. On the interface, Dirichlet boundary conditions are imposed according to the latest updated value of the beam displacement time derivative. A V-cycle multigrid algorithm is used to obtain a new update solution for the pressure and the velocity. Then the energy equation system is assembled, using the previously evaluated velocity and grid velocity in the advection term. A multigrid V-cycle is solved and updated values of the temperature are found. Finally the beam equation system is built, where the load field is computed using the previous evaluated pressure. Since the number of the subdomain unknowns is limited a direct LU decomposition solver can be used for computing the new displacement and its time derivatives. The grid velocity is then computed and the grid nodes are advected along the corresponding characteristic lines. The whole procedure is repeated until convergence is finally reached.

The Navier-Stokes, energy and grid-velocity systems are solved using a fully coupled iterative multigrid solver [20] with a Vanka-type smoother. Multigrid solvers for coupled velocity/pressure system compute simultaneously the solution for both the pressure and the velocity field, and they are known to be one of the best classes of solvers for laminar Navier-Stokes equations [19]. An iterative coupled solution for the linearized discretized Navier-Stokes system requires the solution of a large number of sparse saddle point problems. In order to optimally solve the coupled equation system, involving the unknown stress vector, we use the block Gauss-Seidel method, where each block consists of a small number of degrees of freedom. The characteristic feature of this type of smoother is that in each smoothing step a large number of small linear systems of equations has to be solved. Each block

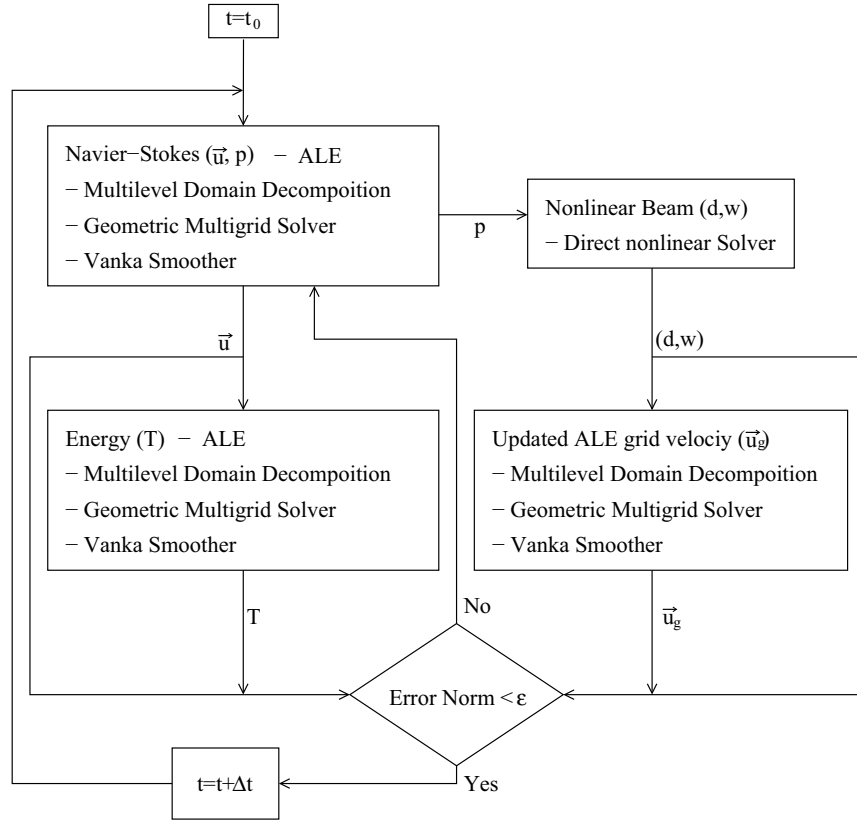


FIGURE 6. Solution methodology for the coupled problem

of equations corresponds to all the degrees of freedom which are connected to few elements. For example, for conforming finite elements, the block may consist of all the elements, containing some pressure vertices. Thus, a smoothing step consists of a loop over all the blocks, solving only the equations involving the unknowns inside the elements that are around the considered pressure vertices. The velocity and pressure variables are updated many times in one smoothing step. The Vanka smoother employed in our multigrid solver involves the solution of a small number of degrees of freedom given by the conforming Taylor-Hood finite element discretization used. For this kind of element the pressure is computed only at the vertices while the velocity field is computed also at the midpoints. Over the internal part of the generic subregion where there are no boundary elements, our Vanka-block consists of an element and all its neighboring elements. We solve for all the degrees of freedom inside the block, with boundary condition taken on the external boundaries. For example, our block consists of four vertex points and 12 midpoints to be solved, for a total of 36 unknowns. We have also used different blocks with different performances but we have found this particular block to be very robust and reliable even at high Reynolds numbers. Examples of computations with this kind of solver can be found in [1, 2, 3]. In order to increase the convergence rate, the considered Vanka-type smoother has been coupled with a standard V-cycle

multigrid algorithm. The multigrid does not change the nature of the solver, but allows the information to travel faster among different parts of the domain. A rough global solution is evaluated on the coarsest mesh $l = 0$ and projected on the finer grid $l = 1$, where Vanka-loops are performed improving its details. The updated solution is then projected on the mesh level $l = 2$ and improved. The procedure is repeated until the finest mesh is reached. Solving the equation system in fine meshes improves solution details, but at the same time reduces the communication speed over the domain. However, this does not affect the global convergence rate since a considerable information exchange among different parts of the domain has been already done when solving in coarser mesh levels. All these considerations can be directly extended to the energy equation solver, where the same element block is considered.

Numerical results of this implementation is illustrated for the interaction of a fluid with an elastic beam. Figure 7 shows the computational domain for the multi-level mesh discretization and the FSI with the moving grid.

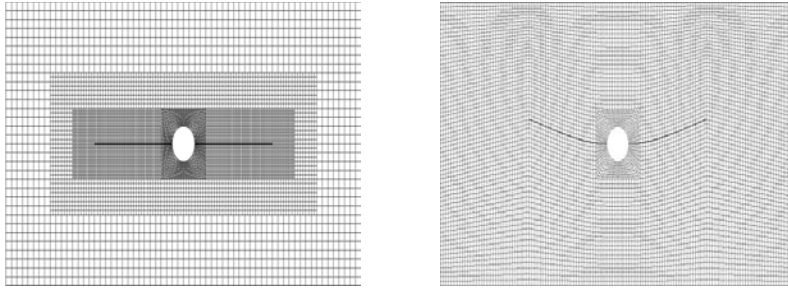


FIGURE 7. Computational domain showing the various mesh discretization levels

4. Discussion and Future Research

In this paper, a model fluid-structure interaction problem was formulated mathematically and a non-conforming computational method to solve the coupled problem was presented. Numerical results for the one-dimensional model problem validated the robustness of the solution methodology. An extension of the coupled fluid structure thermal interaction problem to higher dimensions for a model problem involving a beam and fluid interaction in a thermal field was also presented. A multilevel computational approach for the associated fluid-structure thermal interaction problem was presented that relied on computing coupled solutions over different sub-domains with different multigrid levels. Numerical results for the reliability of the scheme was also presented for a benchmark problem.

As novel procedures for solving fluid-structure interaction problems are developed, it is important to build an efficient parallel infrastructure that is designed to optimize the computational process. Hence there is a necessity to study the advantages of these non-conforming methods to implement the resulting global system on a parallel machine in an easier and faster way. Therefore the goal is to continue to improve the efficiency (i.e. the ratio of the computation time to the communication time) by employing these techniques.

Also, there has been considerable amount of interest in *multiscale* modeling specific to either a structural problems or fluid problems and a new theory for multiscale approach to fluid-structure modeling is yet to be established. There is therefore a need to design new algorithms using our multi-disciplinary non-conforming finite element methodology for solving fluid-structure-control interaction problems, by taking into account the *multiscale and nonlinear behavior* of various components associated with the coupled problem.

Finally, it is important to test the performance of the computational algorithms developed realistic fluid-structure applications such as blood flow in a parent-artery/aneurysm multistructure or computational aeroelasticity of micro-air vehicle.

Acknowledgments

This research was supported in part by the corresponding author's grant from the National Institutes for Health.

References

- [1] E. Aulisa, S. Manservigi and P. Seshaiyer, A non-conforming computational methodology for modeling coupled problems. *Nonlinear Analysis*, 63: 1445-1454, 2005.
- [2] E. Aulisa, S. Manservigi and P. Seshaiyer A computational multilevel approach for solving 2D Navier-Stokes equations over non-matching grids, *Computer Methods in Applied Mechanics and Engineering*, 56(8): 1139-1145, 2008.
- [3] E. Aulisa, S. Manservigi, P. Seshaiyer and A. Idesman Distributed Computational Methods for Coupled Fluid Structure Thermal Interaction Applications, *Journal of Algorithms and Computational Technology*, 291-310, 2010.
- [4] F. Ben Belgacem. The mortar finite element method with Lagrange Multipliers. *Numerische Mathematik*, 84(2):173-197, 1999.
- [5] F. Ben Belgacem, L. K. Chilton and P. Seshaiyer. The *hp* mortar finite element method for the mixed elasticity and stokes problems. *Computers and Mathematics with Applications*, 46:35-55, 2003
- [6] F. Ben Belgacem, L. K. Chilton and P. Seshaiyer. Non-conforming computational methods for mixed-elasticity problems. *Computational Methods in Applied Mathematics*, 3(1):1-12, 2003.
- [7] F. Ben Belgacem, P. Seshaiyer and M. Suri. Optimal convergence rates of *hp* mortar finite element methods for second-order elliptic problems. *RAIRO Mathematical modeling and numerical analysis*, 34:591-608, 2000.
- [8] C. Bernardi, Y. Maday, and A. T. Patera. Domain decomposition by the mortar element method. In *Asymptotic and Numerical Methods for PDEs with Critical Parameters* (H. G. Kaper and M. Garbey editors), 269-286, 1993.
- [9] J. Donea An Arbitrary Lagrangian-Eulerian Finite Element Method for Transient Fluid-Structure Interactions. *Comp. Meth. in Appl. Mech. and Engg.*, 33:689-723, 1982.
- [10] C. Grandmont, V. Guimet and Y. Maday. Numerical analysis of some decoupling techniques for the approximation of the unsteady fluid structure interaction, *Mathematical Models and Methods in Applied Sciences*, 11:1349-1377, 2001.
- [11] T. Hughes, W. Liu and T. Zimmermann. Lagrangian-Eulerian finite element formulation for incompressible viscous flows. *Computer Methods in Applied Mechanics and Engineering*, 29:329-349, 1981.
- [12] V. John and L. Tobiska. Numerical Performance of Smoothers in Coupled Multigrid Methods for the Parallel Solution of the Incompressible Navier-Stokes Equations, *International Journal for Numerical Methods in Fluids*, 33:453-473, 2000.
- [13] P. Seshaiyer. Stability and convergence of non-conforming *hp* finite element methods. *Computers and Mathematics with Applications*, 46: 165-182, 2003.
- [14] P. Seshaiyer and L. K. Chilton. The *hp*-mortar domain decomposition method for problems in fluid mechanics. *Int. Journal of Numerical Methods in Fluids*, 40(12):1561-1570, 2002.
- [15] A. Patel, A. Pani and N. Natraj. Mortar element method for parabolic problems. *Numerical Methods for Partial Differential Equations*, 24:1460-1484, 2008.
- [16] P. Seshaiyer and M. Suri. Uniform *hp* convergence results for the mortar finite element method. *Mathematics of Computations*, 69:521-546, 2000.

- [17] P. Seshaiyer and M. Suri. *hp* submeshing via non-conforming finite element methods. *Computer Methods in Applied Mechanics and Engineering*, 189:1011–1030, 2000.
- [18] E. Swim and P. Seshaiyer. A nonconforming finite element method for fluid-structure interaction problems. *Computer Methods in Applied Mechanics and Engg.*, 195:2088–2099, 2006.
- [19] S. Turek. *Efficient solvers for incompressible flow problems: an algorithmic and computational approach*, Vol. 6, Lecture Notes in comp. science and engineering, Springer, 1999.
- [20] S. Vanka. Block-implicit multigrid calculation of two-dimensional recirculation flows. *Computer Methods in Applied Mechanics and Engineering*, 59:29–48, 1986.

Eugenio Aulisa, Mathematics and Statistics, Texas Tech University, Lubbock, TX

Sonia Garcia, Department of Mathematics, US Naval Academy, Annapolis, MD

Edward Swim, Mathematics and Statistics, Sam Houston State University, Huntsville, TX

Padmanabhan Seshaiyer, Mathematical Sciences, George Mason University, Fairfax, VA