

VI International Conference on Computational Methods for Coupled Problems in Science and Engineering
COUPLED PROBLEMS 2015
B. Schrefler, E. Oñate and M. Papadrakakis(Eds)

A PARALLEL FULLY-COUPLED FLUID-STRUCTURE INTERACTION SIMULATION OF A CEREBRAL ANEURYSM

ALI EKEN[†] AND MEHMET SAHIN[†]

[†]Astronautical Engineering Department
Faculty of Aeronautics and Astronautics
Istanbul Technical University
Maslak, 34469 Istanbul, Turkey

e-mail: msahin.ae00@gtalumni.org, web page: <http://web.itu.edu.tr/~msahin/>

Key words: Fluid-Structure Interaction, Monolithic Methods, ALE Methods, Cerebral Aneurysm

Abstract. A parallel fully-coupled approach has been developed for the fluid-structure interaction problem in a cerebral artery with aneurysm. An Arbitrary Lagrangian-Eulerian formulation based on the side-centered unstructured finite volume method [2] is employed for the governing incompressible Navier-Stokes equations and the classical Galerkin finite element formulation is used to discretize the constitutive law for the Saint Venant-Kirchhoff material in a Lagrangian frame for the solid domain. A special attention is given to construct an algorithm with exact fluid mass/volume conservation while obeying the global discrete geometric conservation law (DGCL). The resulting large-scale algebraic linear equations are solved using a one-level restricted additive Schwarz preconditioner with a block-incomplete factorization within each partitioned sub-domains. The parallel implementation of the present fully coupled unstructured fluid-structure solver is based on the PETSc library for improving the efficiency of the parallel algorithm. The proposed numerical algorithm is applied to a complicated problem involving unsteady pulsatile blood flow in a cerebral artery with aneurysm as a realistic fluid-structure interaction problem encountered in biomechanics.

1 INTRODUCTION

The development of increasingly accurate and reliable simulation tools in hemodynamics allows to better understand development of cardiovascular diseases, design and evaluation of medical devices and prediction of surgical outcomes. It is known that hemodynamic factors like the wall shear stress (WSS) play a major role in cardiovascular diseases. In particular, areas of turbulence, flow recirculation or places where the artery

wall is subject to low or oscillating shear stress are at higher risk for plaque formation and disease.

To investigate the influence of hemodynamic factors in blood vessels, we have developed a parallel fluid-structure interaction (FSI) analysis technique [1] and applied to a complicated problem involving unsteady pulsatile blood flow in a cerebral artery with aneurysm as a realistic fluid-structure interaction problem encountered in biomechanics. The side-centered unstructured finite volume method based on an arbitrary Lagrangian-Eulerian formulation [2] is employed for the governing incompressible Navier-Stokes equations and the classical Galerkin finite element formulation is used to discretize the constitutive law for the compressible Saint Venant-Kirchhoff material in a Lagrangian frame. The present arrangement of the primitive variables leads to a stable numerical scheme and it does not require any *ad-hoc* modifications in order to avoid odd-even pressure decoupling or spurious pressure modes on unstructured meshes [3]. To the authors' best knowledge, the present arrangement of the primitive variables is not considered for the FSI problems. In the present work, a special attention will be given to satisfy the continuity equation exactly within each element and the summation of the continuity equations can be exactly reduced to the domain boundary. In addition, a special attention is given to construct a second-order ALE algorithm obeying the local DGCL [4]. Furthermore, a more compatible discrete kinematic boundary condition is enforced at the common fluid-structure interface in order to satisfy the global DGCL, which is required in order to conserve total fluid volume/mass at machine precision.

FSI simulations in general can be solved in a monolithic or partitioned way. In the partitioned approach, separate solvers are utilized for the fluid and structure subproblems. The main advantage of the partitioned approach is the ability to reuse existing solvers which allows the application of different, possibly more efficient, computational methods specifically developed for either the fluid or the structure subproblem. Although the implementation of this approach is relatively easy, it does, however, suffer some serious drawbacks. The fixed point iterations tend to converge slowly and the iterations may diverge in the presence of strong fluid-structure coupling due to the high fluid/structure density ratio which causes to the so-called artificial added mass effect [5]. In a fully coupled (monolithic) approach, the fluid and structure equations are discretized and solved simultaneously as a single equation system for the entire problem. However, this requires an efficient numerical technique for the solution of a large system of coupled nonlinear algebraic equations, which poses the major challenge of monolithic FSI approaches, especially in large scale problems. Although monolithic solvers are believed to be too expensive for use in large-scale problems, more recent studies [6, 7] demonstrate that monolithic solvers are competitive even in the case of weak fluid-structure interactions problems. In the present paper, the original system of equations is preconditioned with an upper triangular right preconditioner which results in a scaled discrete Laplacian instead of a zero block in the original system due to the divergence-free constraint. Then a one-level restricted additive Schwarz preconditioner with a block-incomplete factorization within each parti-

tioned sub-domains is utilized for the resulting fully coupled system. The implementation of the preconditioned Krylov subspace algorithm, matrix-matrix multiplication and the restricted additive Schwarz preconditioner are carried out using the PETSc [8] software package developed at the Argonne National Laboratories.

2 MATHEMATICAL AND NUMERICAL FORMULATION

The integral form of the incompressible Navier-Stokes equations over an arbitrary moving control volume are discretized using the arbitrary Lagrangian-Eulerian (ALE) based side-centered finite volume method [2]. In this approach, the velocity vector components are defined at the mid-point of each cell face, while the pressure term is defined at element centroids. The present arrangement of the primitive variables leads to a stable numerical scheme and it does not require any *ad-hoc* modifications in order to enhance the pressure-velocity coupling. In the current discretization, the continuity equation is satisfied within each hexahedral elements at machine precision and the summation of the discrete equations can be exactly reduced to the domain boundary, which is important for the global mass conservation. In addition, a special attention is given to construct a second-order accurate arbitrary Lagrangian-Eulerian algorithm obeying the discrete geometric conservation law (DGCL) [4]. The classical Galerkin finite element formulation is used to discretize the governing solid equations for the compressible Saint Venant-Kirchhoff material. In here, the displacements at any point in the isoparametric hexahedral element are approximated by a linear combination of the displacements at the nodal points of the element. The numerical simulation of fluidstructure interaction problems requires to fulfill two coupling conditions: the kinematic and the dynamic continuity across the fluid-solid interface at all times. The kinematic boundary condition on the fluid-structure interface is driven by requiring continuity of the velocity while the dynamic condition means that the equilibrium equation holds for the surface traction at the common fluid-structure interaction boundary. However, the problem with the classical application of kinematic boundary condition is that the discrete equations are not compatible with the global DGCL and the total volume/mass of the fluid domain will not conserved at machine precision if an incompressible fluid is fully enclosed in the solid domain. A more compatible application of the kinematic boundary condition is given in [1] in order to conserve total fluid volume/mass at machine precision. In the present paper, the original system of equations is preconditioned with an upper triangular right preconditioner which results in a scaled discrete Laplacian instead of a zero block in the original system due to the divergence-free constraint. Then a one-level restricted additive Schwarz preconditioner with a block-incomplete factorization within each partitioned sub-domains is utilized for the resulting fully coupled system. The implementation of the preconditioned Krylov subspace algorithm, matrix-matrix multiplication and the restricted additive Schwarz preconditioner are carried out using the PETSc [8] software package developed at the Argonne National Laboratories. The METIS library [9] is used to partition the computational domain for a balanced domain decomposition.

3 NUMERICAL RESULTS

In this section, the numerical method is applied to a more complicated problem involving unsteady pulsatile blood flow in a cerebral artery with aneurysm. The geometry of the arterial lumen is taken from the work of Marchandise et al. [10]. The average radius of the arterial lumen is 0.1075cm at the inlet and the values are 0.0855cm and 0.0729cm at the two outflow ends. All hexahedral conforming mesh generation for the present geometry is rather challenging and the DISTENE MeshGems-Hexa algorithm based on the octree method is used to generate the initial all hexahedral conforming coarse mesh. Then the CUBIT Geometry and Mesh Generation Toolkit developed at the Sandia National Laboratories [11] is employed to refine the initial mesh with optimization-based smoothing techniques. The boundary layer mesh is constructed by extruding the mesh in the outward normal direction. However, this approach is not suitable for the construction of the mesh for the vascular wall due its relatively large thickness which leads to untangled elements. For this purpose, we use the radial basis function (RBF) based mesh deformation algorithm given in [2] which moves the coarsened number of mesh vertices on the lumen surface in the outward normal direction exactly with a distance of 0.02cm , meanwhile the rest of vertices on the lumen surface vertices are moved using the RBF interpolation. In addition, we added two layers of hexahedral meshes at the inflow and outflow ends in order to make these surfaces planar. The computational mesh shown in Figure 1-a consists of 501,065 vertices and 472,794 hexahedral elements for the fluid domain. The solid domain is constructed using 5 layers of hexahedral elements across the arterial wall and consists of 262,870 vertices and 262,275 elements leading to 7,364,244 DOF for the whole domain.

The boundary condition at the inflow is set to the periodic velocity boundary condition across a single cell distance with the time-dependent prescribed mass flow rate given in [12]. The time variation of the cross-sectional average velocity at the inlet is provided in Figure 1-b. At the outflow the natural (traction-free) boundary condition is imposed. Although more realistic outflow conditions such as the resistive boundary condition [13] may be applied, neither the resistance parameter nor the physical properties of the arterial system is available to us. For the solid domain, the arterial walls are clamped near to the inflow and a Neumann homogeneous boundary condition is applied on the other ends of the structure as in the work of Crosetto et al. [14]. On the exterior surface of the solid domain, we impose the following condition due to the support of exterior tissue [14]

$$\boldsymbol{\sigma}_s \mathbf{n} + \alpha_s \mathbf{d} = 0 \tag{1}$$

with $\alpha_s = 1 \times 10^4$. Rayleigh damping $[C] = \alpha[M] + \beta[K]$ with $\alpha = 6 \times 10^3$ and $\beta = 0$ is also introduced in order to model the damping effect of surrounding tissue. The present simulations are carried out with a constant time step of 0.004s . In these simulations, the blood is assumed to be a Newtonian fluid and the material properties are provided in Table 1.

The numerical simulation of the blood flow within the present saccular aneurysm is presented by Salmon et al. [15] by assuming that the vessel wall is rigid. The non-dimensional Reynolds number based on the peak cross-sectional average velocity and the trunk radius at the entrance cross section $Re = \rho_f \bar{U}_{max} \bar{R} / \mu_f$ is equal to 159.38. The Womersley number, based on the trunk radius at the entrance cross section $Wo = \bar{R} \sqrt{\omega / \nu_f}$ equals to 1.6 with an angular frequency of $\omega = 2\pi \text{ rad/s}$. The Strouhal number $St = \omega \bar{R} / \bar{U}_{max}$ is 0.016. In the current numerical simulations, the vessel wall is modelled as an isotropic hyperelastic material using the compressible Saint Venant-Kirchhoff model. The present numerical calculations are started impulsively and the numerical solutions presented in here correspond to third cardiac cycle. The velocity vector magnitude contours are presented in several planes normal to y -axis along with the stream traces in Figure 2-a at

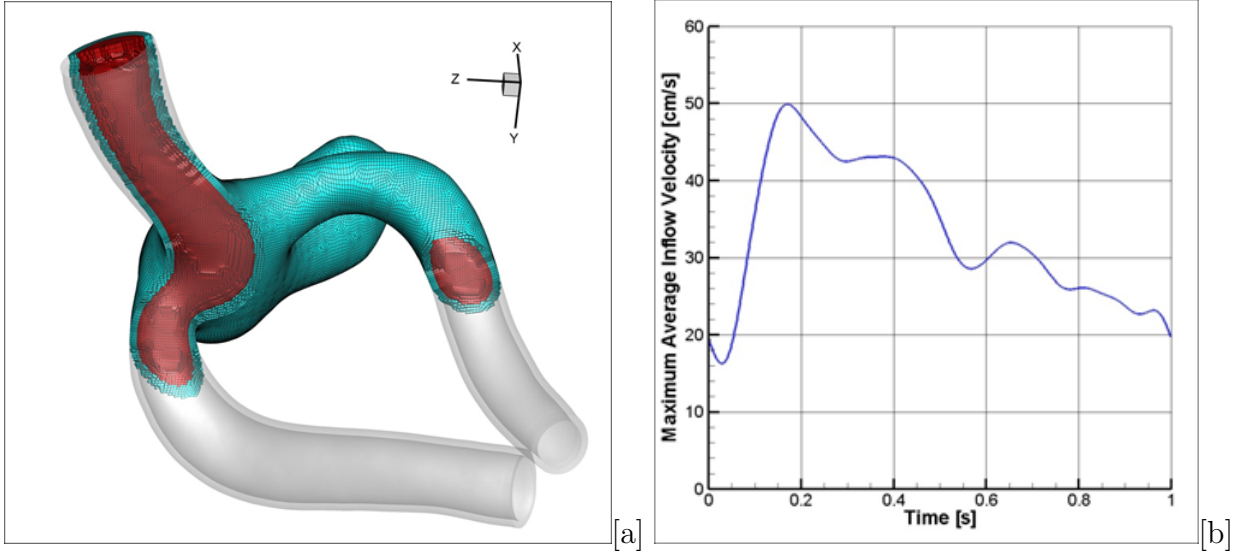


Figure 1: The computational all hexahedral conformal mesh with 735,069 hexahedral elements and 763,935 nodes (7,364,244 DOF) for fluid and solid domains [a] and the time variation of the total mass flow rate at inlet [b].

Table 1: Material properties.

Fluid	Density ρ_f [g/cm^3]	1
	Dynamic viscosity μ_f [$g/cm\dot{s}$]	0.04
Solid	Density ρ_s [g/cm^3]	1.2
	Poisson ratio ν_s	0.45
	Young modulus E_s [$dynes/cm^2$]	6×10^6

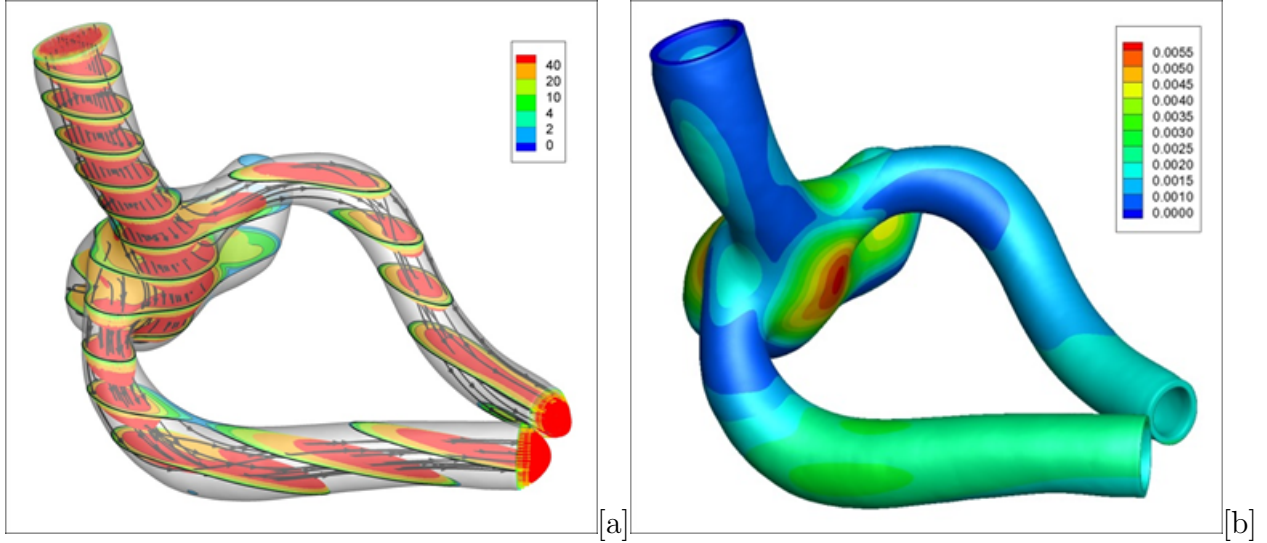


Figure 2: The computed velocity vector magnitude contours at several planes normal to y -axis with streamtraces [a] and the arterial wall displacement [b] at $t = 2.2s$

$t = 2.2s$. The velocity contours indicate that a low velocity zone exists towards to the end of the saccular aneurysm. In addition, a large swirl is observed within the saccular aneurysm. Furthermore, three-dimensional Dean vortices are observed within the two curved branches of the saccular aneurysm. These three-dimensional flow structures may be seen more clearly from Figure 3-b. The simulations also indicate that the initial artery wall deformation at the inlet section is not radially symmetric due to the elliptical geometry of the inlet cross section. The maximum deformation is observed at the middle of the bifurcation of the branch where the lumen surface curvature is very small, as seen from Figure 2-b. In these simulations, it is important to impose the support due to exterior tissue since the branches tend to move away from each other due to the employed Neumann homogeneous boundary condition at the outlets. The computed wall shear stress (WSS) lines in Figure 3-a indicate separation line just before the saccular aneurysm. Large shear stresses are also observed close to the shoulder of the branch bifurcation. However wall shear stress is rather low towards to the end of the saccular aneurysm where the velocity magnitude is relatively low. It should be noted that the use of present octree based all hexahedral meshes leads to a more effective solution technique since the background mesh consists of mostly uniform Cartesian meshes. Therefore, the combination of the present algorithm with the octree based mesh generation is very efficient for treating complex FSI problems. However, we should mention that there are still open issues related to all hexahedral conformal mesh generation [16].

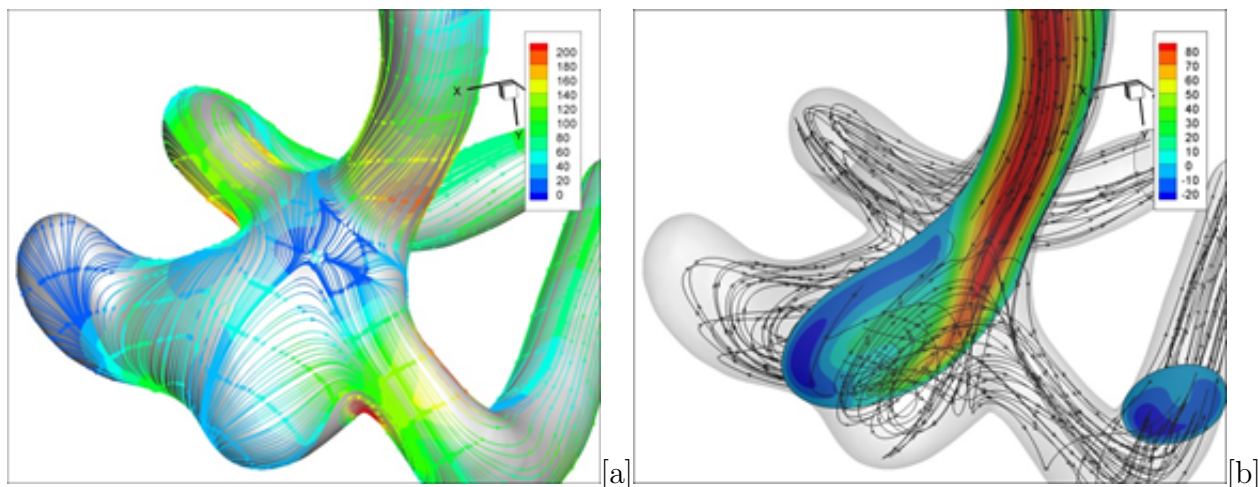


Figure 3: The wall shear stress lines [a] and the vertical velocity component contours with the 3d streamtraces [b] at $t = 2.16s$.

4 CONCLUSIONS

The parallel monolithic approach [1] has been applied to the unsteady pulsatile blood flow in a cerebral artery with aneurysm. The governing equations for the incompressible Navier-Stokes equations for the fluid domain is discretized using the side-centered finite volume method based on an Arbitrary Lagrangian Eulerian formulation meanwhile the nonlinear Saint Venant-Kirchhoff equations for the solid domain is discretized using the classical Galerkin finite element. The continuity equation is satisfied within each element exactly and the summation of the continuity equations can be exactly reduced to the domain boundary, which is important for the global mass conservation. In addition, a special attention is given to construct a second-order ALE algorithm obeying the DGCL. Furthermore, a more compatible application of kinematic boundary condition is introduced at the common fluid-structure interface in order to conserve total fluid volume/mass at machine precision. The resulting large-scale nonlinear equations from the discretization of fluid and solid domains are solved in a fully coupled form using a monolithic approach based on a one-level restricted additive Schwarz preconditioner with a block-incomplete factorization within each partitioned sub-domains.

5 ACKNOWLEDGMENTS

The authors acknowledge financial support from Turkish National Scientific and Technical Research Council (TUBITAK) through project number 112M107. The authors also gratefully acknowledge the use of the computing resources provided by the National Center for High Performance Computing of Turkey (UYBHM) under grant number 10752009 and the computing facilities at TUBITAK ULAKBIM, High Performance and Grid Computing Center.

REFERENCES

- [1] A. Eken and M. Sahin, A parallel monolithic algorithm for the numerical simulation of large-scale fluid structure interaction problems. (submitted).
- [2] B. Erzincanli and M. Sahin, An arbitrary Lagrangian-Eulerian formulation for solving moving boundary problems with large displacement and rotations. *Journal of Computational Physics* (2013) **255**:660–679.
- [3] Y. H. Hwang, Calculations of incompressible flow on a staggered triangle grid, Part I: Mathematical formulation. *Numer. Heat Transfer B*(1995) **27**:323–1995.
- [4] P. D. Thomas and C. K. Lombard, Geometric conservation law and its application to flow computations on moving grids. *AIAA J.* (1979) **17**:1030–1037.
- [5] C. Förster and W. A. Wall and E. Ramm, Artificial added mass instabilities in sequential staggered coupling of nonlinear structures and incompressible viscous flows. *Comput. Meth. Appl. Mech. Engrg.* (2007) **196**:1278-1293.
- [6] M. Heil and A. L. Hazel and J. Boyle, Solvers for large-displacement fluid structure interaction problems: segregated versus monolithic approaches. *Comput. Mech.* (2008) **43**:91–101.
- [7] U. Küttler, M. Gee, Ch. Förster, A Comerford and W. A. Wall, Coupling strategies for biomedical fluid-structure interaction problems. *Int. J. Numer. Meth. Bio.* (2010) **26**:305–321.
- [8] S. Balay, J. Brown, K. Buschelman, V. Eijkhout, W. D. Gropp, D. Kaushik, M. G. Knepley, L. C. McInnes, B. F. Smith and H. Zhang, PETSc Users Manual. *ANL-95/11 - Revision 3.3* Argonne National Laboratory (2012).
- [9] G. Karypis and V. Kumar, A fast and high quality multilevel scheme for partitioning irregular graphs. *SIAM J. Sci. Comput.* (1998), **20**:359–392.
- [10] E. Marchandise, P. Crosetto, C. Geuzaine, J-F, Remacle and E. Sauvage, Quality open source mesh generation for cardiovascular flow simulations. *MSE&A Modeling, Simulation and Applications*. Springer Milan, (2012) 395–414.
- [11] T. D. Blacker, S. Benzley, S. Jankovich, R. Kerr, J. Kraftcheck, R. Kerr, P. Knupp, R. Leland, D. Melander, R. Meyers, S. Mitchell, J. Shepard, T. Tautges and D. White, CUBIT Mesh Generation Environment Users Manual Volume 1. Sandia National Laboratories: Albuquerque, NM (1999).
- [12] Thiriet, M. *Biology and Mechanics of Blood Flow. Part II: Mechanics and Medical Aspects*. Springer, (2008).

- [13] Y. Wua and X.-C. Cai, A parallel two-level method for simulating blood flows in branching arteries with the resistive boundary condition. *Comput. Fluids* (2011) **45**:92–102.
- [14] P. Crosetto, P. Reymond, S. Deparis, D. Kontaxakis, N. Stergiopoulos and A. Quarteroni, Fluidstructure interaction simulation of aortic blood flow. *Computers & Fluids* (2011) **43**:46–57.
- [15] S. Salmon, M. Thiriet and J.-F. Gerbeau, Medical image-based computational model of pulsatile flow in saccular aneurisms. *ESAIM: Mathematical Modelling and Numerical Analysis* (2003) **37**:663–679.
- [16] R. Schneiders, Octree based hexahedral mesh generation. *Int. Journal of Comp. and Applications* (2000) **10**:383–398.