

# An Eulerian-ALE Embedded Boundary Method for Turbulent Fluid-Structure Interaction Problems

Vinod Lakshminarayan<sup>\*</sup>, Alex Main<sup>†</sup>, Kevin Wang<sup>‡</sup>, Charbel Farhat<sup>§</sup>

*Stanford University, Stanford, CA, 94305-3035, USA*

The FInite Volume method with Exact two-phase Riemann problems (FIVER) is a robust Eulerian semi-discretization method for compressible multi-material (fluid-fluid, fluid-structure, or multi-fluid-structure) problems characterized by large density jumps and highly nonlinear structural motions and deformations. Its key components include an embedded boundary method for Computational Fluid Dynamics (CFD), the construction and solution of local, exact, two-phase Riemann problems at the material interfaces, and a conservative algorithm for computing the finite element representation of flow-induced loads on the structure. Originally developed for inviscid multi-material problems, FIVER is extended in this paper to turbulent viscous fluid-structure interaction problems. To this effect, it is equipped with a carefully designed extrapolation scheme for populating the ghost fluid values required for the construction of a second-order spatial approximation of the viscous and source terms of the governing flow equations. Its load distribution algorithm is also extended to account for the contribution of the viscous stress tensor. To maintain the boundary layers resolved during large displacements, rotations, and/or deformations of the structure, the governing flow equations are formulated in a rigid instance of the Arbitrary Lagrangian Eulerian (ALE) framework, and FIVER is further equipped with the corotational method for updating the rigid body motion of the CFD mesh. This non deformable mesh motion enables the original non body-fitted CFD mesh to follow the boundary layers, and therefore minimize the need for complex adaptive mesh refinement. To achieve nonlinear stability, temporal discretization is performed using an extension of the second-order three point backward difference implicit scheme that satisfies its discrete geometric conservation law. Finally, the resulting Eulerian-ALE FIVER method is verified with the Large Eddy Simulation (LES) of turbulent flows past two counter-rotating cylinders and a heaving airfoil. Its potential for the solution of challenging viscous fluid-structure interaction problems characterized by large structural motions is demonstrated with the simulation, using the Spalart-Allmaras and Detached Eddy Simulation (DES) turbulence models, of pitch-up and roll maneuvers of an aeroelastic F/A-18 configuration driven by suitable deployments of its control surfaces.

## I. Introduction

Turbulent viscous Fluid-Structure Interaction (FSI) problems arise in many scientific and engineering applications. Examples include flutter, limit cycle oscillation, buffet, dynamic loads analysis at high angles of attack, parachute dynamics, weapon bay acoustics, store separation trajectory predictions, boom refueling and egress operations, aeroelastic tailoring of aircraft and automotive systems, gate sliding, wind turbine and tire noise analysis, and hemodynamics and cardiovascular technology. All three Lagrangian, Arbitrary Lagrangian Eulerian (ALE), and Eulerian computational frameworks have been developed for the solution

---

<sup>\*</sup>Post-Doctoral Student, Department of Aeronautics and Astronautics, William F. Durand Building, Room 023B, Stanford University, Stanford, CA 94305-3035, USA

<sup>†</sup>Graduate Student, Institute for Computational and Mathematical Engineering, William F. Durand Building, Room 028D, Stanford University, Stanford, CA 94305-3035, USA

<sup>‡</sup>Graduate Student, Institute for Computational and Mathematical Engineering, William F. Durand Building, Room 028D, Stanford University, Stanford, CA 94305-3035

<sup>§</sup>Vivian Church Hoff Professor of Aircraft Structures, Department of Aeronautics and Astronautics, William F. Durand Building, Room 257, Stanford University, Stanford, CA 94305-3035, USA; AIAA Fellow

of such problems. The Lagrangian and ALE computational frameworks move the Computational Fluid Dynamics (CFD) mesh, distort it with the fluid-structure interface, and move this interface with the local fluid velocity. Unfortunately, large mesh distortions induced by large displacements, rotations, or deformations of the fluid-structure interface can reduce the accuracy and numerical stability of a Lagrangian method to the point where it becomes impractical. Similarly, large structural motions and/or deformations challenge most if not all CFD mesh motion schemes<sup>1,2,3,4</sup> on which the ALE computational framework<sup>5,6,7,8,9,10</sup> rely. Topological changes such as those induced, for example, by topology optimization<sup>11</sup> or crack propagation typically make the ALE approach unfeasible. The Eulerian computational framework avoids all aforementioned issues associated with complex or large transformations of the fluid-structure interface by embedding the wet boundary surface of the structure of interest in a fixed CFD mesh, and relying on computational geometry tools<sup>12,13</sup> for capturing or tracking the evolution of the position, shape, and topology of this dynamic boundary surface representing the fluid-structure interface.

Adopting the Eulerian computational framework for FSI problems and embedding the boundary surface of a rigid or flexible dynamic structure in a fixed computational fluid domain not only addresses the aforementioned limitations of the Lagrangian and ALE approaches, but also leads to the concept of CFD computations on non body-fitted meshes and therefore simplifies the task of mesh generation. For all these reasons, Eulerian methods for computing flows on embedding CFD meshes have gained popularity, albeit under different names such as “immersed boundary”, “embedded boundary”, “fictitious domain”, and “Cartesian” methods (for example, see<sup>14,15,16,17</sup>). All of these and related computational approaches are collectively referred to here as Embedded Boundary Methods (EBMs).

Unfortunately, because they operate on non body-fitted CFD meshes, EBMs complicate the treatment of wall boundary conditions in general, and fluid-structure transmission conditions in particular. Furthermore, they tend to be first-order space-accurate at the fluid-structure interface. In some cases, they are even provably inconsistent at this interface.<sup>18</sup> An equally important drawback of EBMs is that they do not track the boundary layers around dynamic rigid or flexible bodies, essentially because they are Eulerian methods. Consequently, they either require high mesh resolutions in large percentages of the fluid computational domain in order to capture the viscous effects — which would make them very inefficient for viscous fluid-structure applications — or they must be equipped with adaptive mesh refinement — which is seldom a simple task in practice.

Recent developments in EBMs have focused primarily on the treatment of wall boundary conditions and the accuracy of the resulting overall semi-discretization scheme at the material interfaces, primarily in the context of incompressible viscous flows and static obstacles. In this case, the major issue becomes more specifically the treatment of the velocity wall boundary condition. For such applications, recently proposed algorithms for interface treatment have focused either on some form of interpolation<sup>19</sup> with particular attention to numerical stability<sup>20</sup> or higher-order accuracy,<sup>19,21,22</sup> or on the concept of a ghost cell,<sup>23,24</sup> some variant of the penalty method,<sup>25</sup> and the mirroring technique.<sup>26</sup> For compressible inviscid flows, a new approach for the treatment of a fluid-structure interface that is equally applicable to static, dynamic, rigid, and flexible embedded boundary surfaces was proposed in.<sup>27</sup> This approach is based on ideas previously developed in<sup>28</sup> for the treatment of fluid-fluid interfaces in multi-fluid problems. It is a departure from the methods outlined above and related works pertaining to incompressible flows in that it treats the velocity and pressure boundary conditions at the embedded boundary surface simultaneously, rather than disjointly. Furthermore, instead of relying exclusively on interpolation and/or extrapolation, the method proposed in<sup>27</sup> enforces the appropriate value of the normal component of the fluid velocity at a wall boundary or fluid-structure interface and recovers the value of the fluid pressure at this location by solving an appropriate local, one-dimensional, exact fluid-structure Riemann (or more precisely half Riemann) problem. Originally developed for the case of a fluid characterized by a relatively simple and yet pervasive Equation Of State (EOS) such as the perfect gas EOS or stiffened gas EOS for which an exact fluid-structure half-Riemann problem is easily solvable, this method was extended in<sup>29</sup> to the case of arbitrarily complex EOS and named FIVER (FInite Volume method with EXact two-phase Riemann problems).

A first extension of FIVER to viscous flows was proposed in.<sup>30</sup> This extension was based however on a constant extrapolation scheme for populating the fluid ghost values needed for semi-discretizing the viscous and source terms of the governing flow equations. Therefore, one objective of this paper is to present a higher-order extrapolation scheme suitable for second-order spatial approximations of viscous and source terms. A second objective of this paper is to couple FIVER with a specific ALE technique in order to construct an EBM for FSI that maintains the boundary layers resolved during large displacements, rotations, and/or

deformations of the structure, without inheriting the aforementioned pitfalls of the ALE computational framework.

To this effect, the remainder of this paper is organized as follows. The governing multi-disciplinary equations of dynamic equilibrium and the fluid and structural semi-discretization methods of interest are presented in Section 1. The original FIVER method developed for the computation of inviscid multi-material (fluid-fluid, fluid-structure, or multi-fluid-structure) problems is outlined in Section 2, in the specific context of Eulerian FSI problems. Its coupling with the specific instance of the ALE computational framework where the CFD mesh is rigidly moved in time is motivated and described in Section 3, together with the corotational scheme chosen for determining the rigid body mesh motion. The extension of FIVER to viscous flows is presented in Section 4. The time-integration of the resulting semi-discrete fluid equations of dynamic equilibrium and that of the coupled semi-discrete fluid-structure equations of motion are outlined in Section 5. A conservative algorithm for computing the Finite Element (FE) representation of the flow-induced loads on the structure is also outlined in Section 6. Finally in Section 7, the resulting Eulerian-ALE FIVER method is verified with the Large Eddy Simulation (LES) of turbulent flows past two counter-rotating cylinders and a heaving airfoil. Also in this section, the potential of the Eulerian-ALE FIVER method for the solution of challenging viscous FSI problems characterized by large structural motions is demonstrated with the simulation, using the Spalart-Allmaras and Detached Eddy Simulation (DES) turbulence models, of pitch-up and roll maneuvers of an aeroelastic F/A-18 configuration driven by appropriate deployments of its control surfaces.

## II. Applications

The Euler-ALE FIVER method is implemented in the AERO Suite of Codes for CFD-based nonlinear aeroelasticity.<sup>34,35</sup> Here, it is first verified with the Large Eddy Simulation (LES) of turbulent flows past two counter-rotating cylinders and a heaving airfoil. Then, its potential for the solution of challenging viscous FSI problems characterized by large structural motions is demonstrated with the simulation, using the Spalart-Allmaras and Detached Eddy Simulation (DES) turbulence models, of pitch-up and roll maneuvers of an aeroelastic F/A-18 configuration driven by appropriate deployments of its control surfaces.

### A. LES simulation of a turbulent flow past two counter-rotating cylinders

The first problem considered is an LES simulation of a doublet-like counter-rotating cylinder pair (see Figure 1) separated by  $g = 3d$ , where  $d$  is the diameter of the cylinder. The cylinders are one diameter long. The freestream Mach number is 0.1 and the Reynolds number is 5000. The non-dimensional rotational frequency  $\Omega$ , given by  $\frac{\omega d}{2U}$  is set to 1.0. The results for this simulation will be included in the final paper.

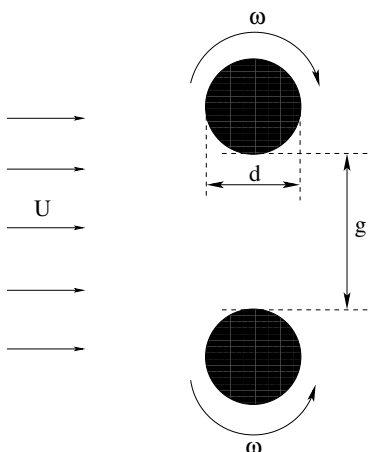


Figure 1. Schematic of doublet-like counter rotating cylinder

## B. LES simulation of a heaving airfoil

The second problem considered is an LES simulation of a heaving NACA0015 airfoil at a Reynolds number of 5000 and a Mach number of 0.1. The span of the airfoil is set to one chord length. The heaving motion of the airfoil is given by the equation:

$$Z = Z_0 + \alpha \sin(2\pi ft) \quad (1)$$

where,  $Z_0$  is the initial  $z$ -coordinate of the airfoil,  $Z$  is  $z$ -coordinate of the airfoil at time  $t$ ,  $\alpha$  and  $f$  are the heaving amplitude and frequency, respectively. In the current simulation,  $\alpha$  is  $0.1464c$ , where  $c$  is the chord of the airfoil. The non-dimensional frequency given by  $fc/U_\infty$  corresponds to 1.604. The mesh used for simulation (shown in Figure 2) has 10 million tetrahedral elements and 1.8 million node points. In order to make the simulation feasible with the embedded grid, the entire mesh is heaved with the airfoil such that the boundary layer lies within the refined portion of the grid. Without this strategy, the entire area covered by the heaving motion will need to have the resolution of a boundary layer grid and thus can lead to tremendous increase in the computational cost.

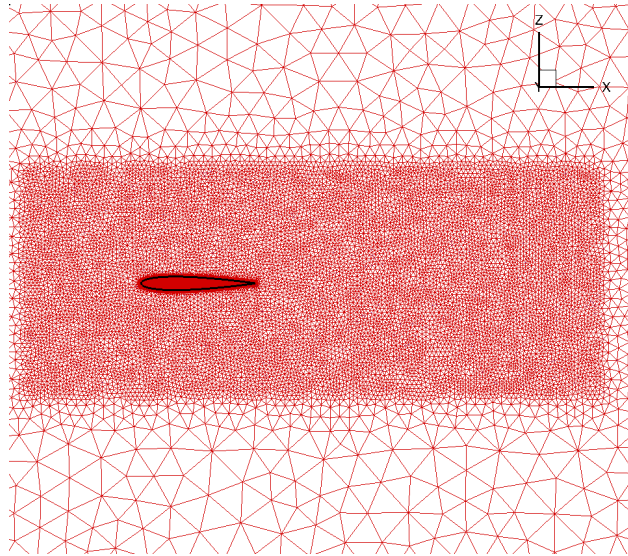


**Figure 2. Mesh used for the heaving NACA0015 airfoil simulation with embedded surface in black**

The Eulerian-ALE FIVER simulation is validated by comparing the force histories with a ALE FV simulation having comparable grid quality. Figure 3 show the converged lift and drag coefficients over two time periods for both the methodology. Clearly, the embedded simulation is seen to predict the forces well. A sample flow visualization plot showing iso-surface of  $\lambda_2$ -criterion colored with pressure contour is shown in Figure 4 to demonstrate the quality of simulation. The vortices shed during the heaving motion is well-captured for about 2-chords.

## C. DES simulation of a pitch-up maneuver of an aeroelastic F/A-18 configuration

The high-fidelity aeroelastic computational model of an F/A-18 configuration with control surfaces and stores considered here consists of an undamped dynamic FE “stick” model (Figure 5) with 11,290 degrees of freedom, an embedded boundary surface with 128,000 grid points (Figure 6), and an embedding mesh with 32,000,000 grid points. The control surfaces are programmed to pitch-up the aircraft at  $M_\infty = 0.35$  to  $90^\circ$ . Snapshots from the coupled fluid-structure simulation performed for this pitch-up maneuver using the Eulerian-ALE FIVER method and the DES turbulence model are shown in Figure 7. They will be explained in details in the final manuscript.

## D. RANS simulation of a high-speed roll maneuver of an aeroelastic F/A-18 configuration

Using the same high-fidelity aeroelastic computational model of the same F/A-18 configuration with control surfaces and stores considered above, a coupled fluid-structure simulation of a roll maneuver at  $M_\infty = 0.75$  is

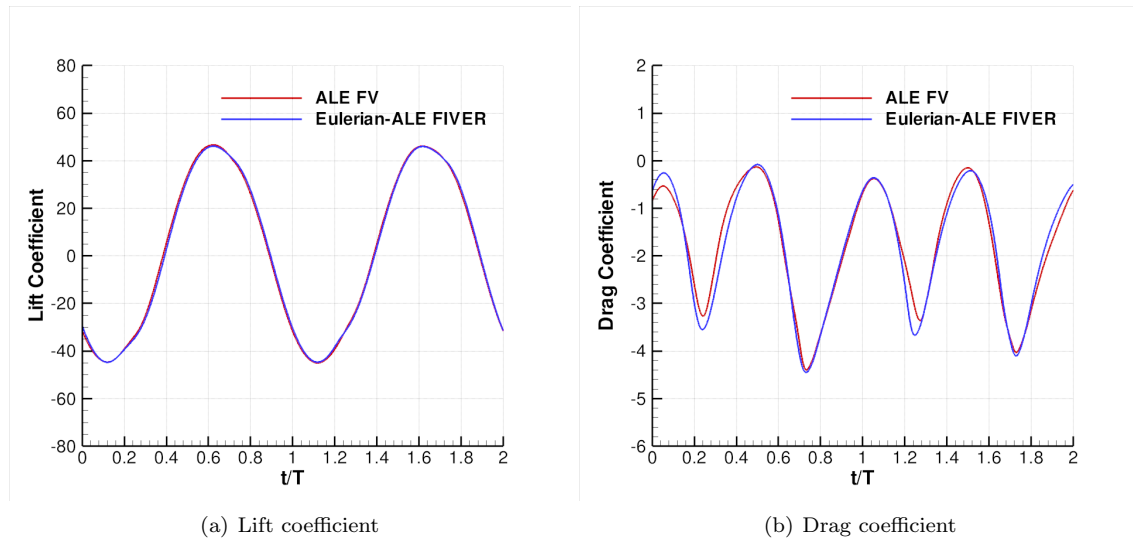


**Figure 3. Comparison of lift and drag coefficient between Eulerian-ALE FIVER and ALE FV simulation for the heaving airfoil over two time periods**

performed using the Eulerian-ALE FIVER method and the Spalart-Allmaras turbulence model. Snapshots of this simulation are shown in Figure 8, and the predicted roll damping effect is highlighted in Figure 9. They will be explained in details in the final manuscript.

## Acknowledgments

The first, third, and fourth authors acknowledge partial support by the Army Research Laboratory through the Army High Performance Computing Research Center under Cooperative Agreement W911NF-07-2-0027, and the Office of Naval Research under Grant N00014-06-1-0505, Grant N00014-09-C-015, and Grant XXX. The second author acknowledges the support of the DOE NNSA Stewardship Science Graduate Fellowship. The content of this publication does not necessarily reflect the position or policy of any of these sponsors, and no official endorsement should be inferred.

## References

- <sup>1</sup>Farhat C, Degand C, Koobus B, Lesoinne, M. Torsional springs for two-dimensional dynamic unstructured fluid meshes. *Computer Methods in Applied Mechanics and Engineering* 1998; **163**:231–245.
- <sup>2</sup>Degand C, Farhat C. A three-dimensional torsional spring analogy method for unstructured dynamic meshes. *Computers and Structures* 2002; **80**:305–316.
- <sup>3</sup>Kanchil H, Masud A. A 3D adaptive mesh moving scheme. *International Journal for Numerical Methods in Fluids* 2007; **54**: 923–944.
- <sup>4</sup>Fossati M, Khurram RA, Habashi WG. An ALE mesh movement scheme for long-term in-flight ice accretion. *International Journal for Numerical Methods in Fluids* 2012; **68**: 958–976.
- <sup>5</sup>Donea J. An arbitrary Lagrangian-Eulerian finite element method for transient fluid-structure interactions. *Computer Methods in Applied Mechanics and Engineering* 1982; **33**:689–723.
- <sup>6</sup>Farhat C, Lesoinne M, Maman N. Mixed explicit/implicit time integration of coupled aeroelastic problems: three-field formulation, geometric conservation and distributed solution. *International Journal for Numerical Methods in Fluids* 1995; **21**:807–835.
- <sup>7</sup>Lesoinne M, Farhat C. Geometric conservation laws for flow problems with moving boundaries and deformable meshes, and their impact on aeroelastic computations. *Computer Methods in Applied Mechanics and Engineering* 1996; **134**:71–90.
- <sup>8</sup>Farhat C, Geuzaine P, Grandmont C. The discrete geometric conservation law and the nonlinear stability of ALE schemes for the solution of flow problems on moving grids. *Journal of Computational Physics* 2001; **174**: 669–694.
- <sup>9</sup>Farhat C, vanderZee, KG, Geuzaine P. Provably second-order time-accurate loosely-coupled solution algorithms for transient nonlinear computational aeroelasticity. *Computer Methods in Applied Mechanics and Engineering* 2006; **195**:1973–2001.
- <sup>10</sup>Farhat C, Rallu A, Wang K, Belytschko T. Robust and provably second-order explicit-explicit and implicit-explicit staggered time-integrators for highly nonlinear fluid-structure interaction problems. *International Journal for Numerical Methods in Engineering* 2010; **84**: 73–107.

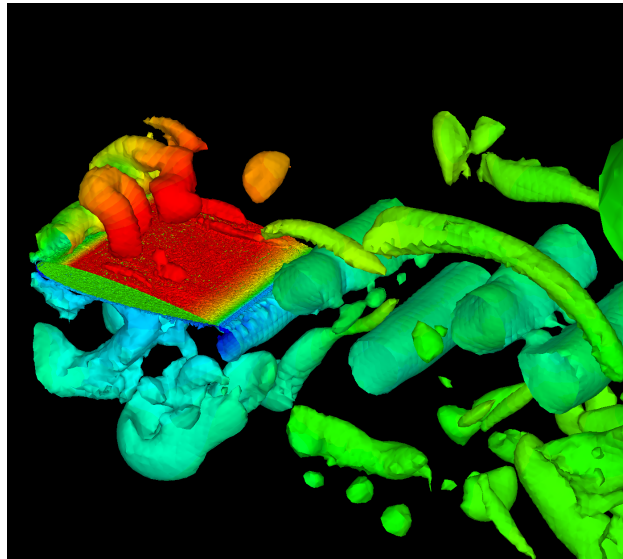


Figure 4. Iso-surface of  $\lambda_2$  criterion colored with pressure contour

<sup>11</sup>Farhat C, Maute K, Argrow B, Nikbay M. A shape optimization methodology for reducing the sonic boom initial pressure rise. *AIAA Journal of Aircraft* 2007; **45**:1007–1018.

<sup>12</sup>Löhner R. Applied computational fluid dynamics techniques: an introduction based on finite element methods (second ed.), John Wiley & Sons (2008) ISBN 978-0-470-51907-3.

<sup>13</sup>Wang K, Grétarsson J, Main A, Farhat C. Computational algorithms for tracking dynamic fluid-structure interfaces in embedded boundary methods. *International Journal for Numerical Methods in Fluids*, DOI: 10.1002/fld.3659

<sup>14</sup>Peskin CS. Flow patterns around heart valves: a numerical method. *Journal of Computational Physics* 1972; **10**:252–271.

<sup>15</sup>Kreiss HO, Petersson A. A second-order accurate embedded boundary method for the wave equation with Dirichlet data. *SIAM Journal of Scientific Computation* 2006; **27**:1141–1167.

<sup>16</sup>Glowinski R, Pan TW, Kearsley AJ, Periaux J. Numerical simulation and optimal shape for viscous flow by a fictitious domain method. *International Journal for Numerical Methods in Fluids* 2005; **20**:695–711.

<sup>17</sup>Johansen H, Colella P. A Cartesian grid embedded boundary method for Poisson’s equation on irregular domains. *Journal of Computational Physics* 1998; **147**:60–85.

<sup>18</sup>Zeng X, Farhat C. A systematic approach for constructing higher-order immersed boundary and ghost fluid methods for fluid and fluid-structure interaction problems. *Journal of Computational Physics*; <http://dx.doi.org/10.1016/j.jcp.2011.12.027>

<sup>19</sup>Fadlun EA, Verzicco R, Orlandi P, Mohd-Yusof J. Combined-immersed boundary finite difference methods for three-dimensional complex flow simulations. *Journal of Computational Physics* 2000; **161**:35–60.

<sup>20</sup>Kim J, Kim D, Choi H. An immersed-boundary finite volume method for simulations of flow in complex geometries. *Journal of Computational Physics* 2001; **171**:132–150.

<sup>21</sup>Glimanov A, Sotiropoulos F. A hybrid Cartesian/immersed boundary method for simulating flows with 3D, geometrically complex, moving bodies. *Journal of Computational Physics* 2005; **207**:457–492.

<sup>22</sup>Choi JJ, Oberoi RC, Edwards JR, Rosati JA. An immersed boundary method for complex incompressible flows. *Journal of Computational Physics* 2007; **224**:757–784.

<sup>23</sup>Tseng YH, Ferziger JH. A ghost-cell immersed boundary method for flow in complex geometry. *Journal of Computational Physics* 2003; **192**:593–623.

<sup>24</sup>Berthelsen PA, Faltinsen OM. A local directional ghost cell approach for incompressible viscous flow problems with irregular boundaries. *Journal of Computational Physics* 2008; **227**:4354–4397.

<sup>25</sup>Löhner R, JBaum JD, Mestreau EL, Sharov D, Charman C, Pelessone D. Adaptive embedded unstructured grid methods. *AIAA-03-1116*, 2003.

<sup>26</sup>Mark A, van Wachem BGM. Derivation and validation of a novel implicit second-order accurate immersed boundary method. *Journal of Computational Physics* 2008; **227**:6660–6680.

<sup>27</sup>Wang K, Rallu A, Gerbeau JF, Farhat C. Algorithms for interface treatment and load computation in embedded boundary methods for fluid and fluid-structure interaction problems. *International Journal for Numerical Methods in Fluids* 2011; **67**:1175–1206.

<sup>28</sup>Farhat C, Rallu A, Shankaran S. A higher-order generalized ghost fluid method for the poor for the three-dimensional two-phase flow computation of underwater implosions. *Journal of Computational Physics* 2008; **227**:7674–7700.

<sup>29</sup>Farhat C, Gerbeau JF, Rallu A. FIVER: a finite volume method based on exact two-phase Riemann problems and sparse grids for multi-material flows with large density jumps. *Journal of Computational Physics*; <http://dx.doi.org/10.1016/j.jcp.2012.05.026>

<sup>30</sup>Farhat C, Larat A, Main A, Avery P, Wang K, Saint-Jalm C. An embedded boundary method for viscous fluid/structure

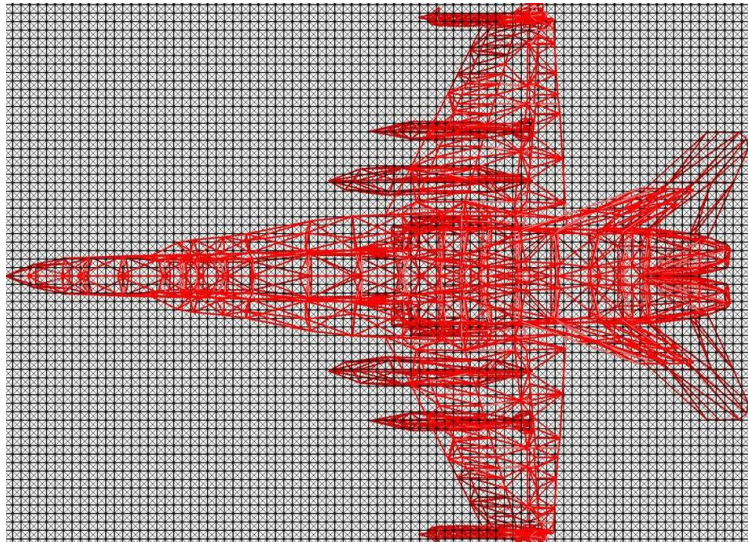


**Figure 5. Undamped dynamic FE stick model of an F/A-18 configuration with control surfaces and stores**

interaction problems and application to flexible flapping wings, AIAA-2012-2688, 42nd AIAA Fluid Dynamics Conference and Exhibit, New Orleans, Louisiana, June 25-28, 2012.

<sup>31</sup>Farhat C, Fézoui L, Lantéri S. Two-dimensional viscous flow computations on the Connection Machine: unstructured meshes, upwind schemes, and massively parallel computations. *Computer Methods in Applied Mechanics and Engineering* 1993; **102**: 61–88.

<sup>32</sup>Steger J, Warming RF. Flux vector splitting for the inviscid gas dynamic with applications to finite-difference methods. *Journal of Computational Physics* 1981; **40**:263–293.

<sup>33</sup>Farhat C, Lesoinne M, LeTallec P. Load and motion transfer algorithms for fluid/structure interaction problems with non-matching discrete interfaces: momentum and energy conservation, optimal discretization and application to aeroelasticity. *Computer Methods in Applied Mechanics* 1998; **157**:95–114.

<sup>34</sup>Farhat C, Geuzaine P, Brown G. Application of a three-field nonlinear fluidstructure formulation to the prediction of the aeroelastic parameters of an F-16 fighter. *Computers & Fluids* 2003; **32**:3–29.

<sup>35</sup>Geuzaine P, Brown G, Harris C, Farhat C. Aeroelastic dynamic analysis of a full F-16 configuration for various flight conditions. *AIAA Journal* 2003; **41**:363–371.

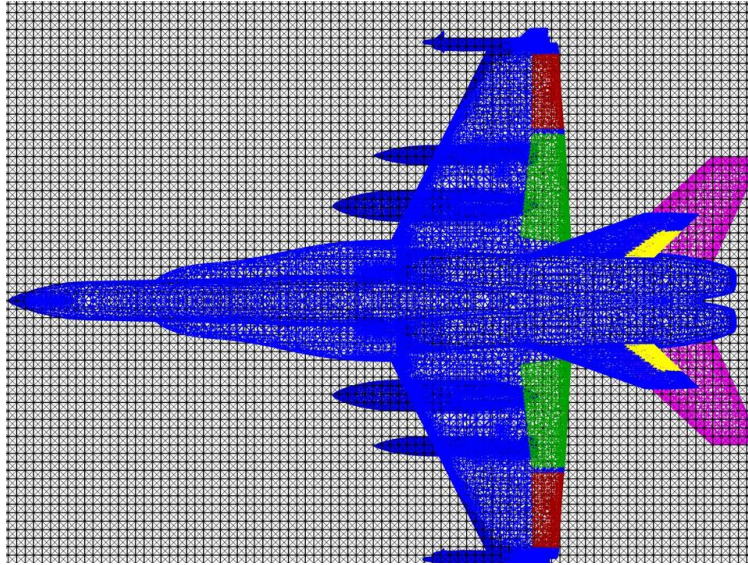


Figure 6. Embedded boundary surface associated with an F/A-18 aeroelastic configuration with control surfaces and stores

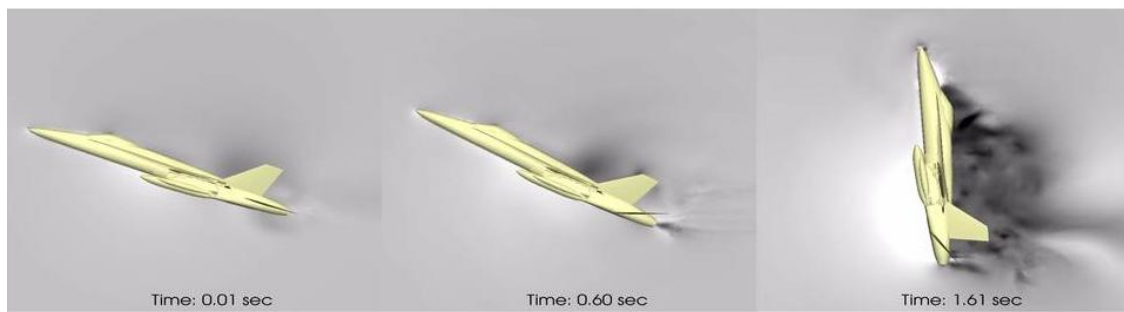


Figure 7. Snapshots from the coupled fluid-structure simulation of a pitch-up maneuver using the Eulerian-ALE FIVER method and the DES turbulence model

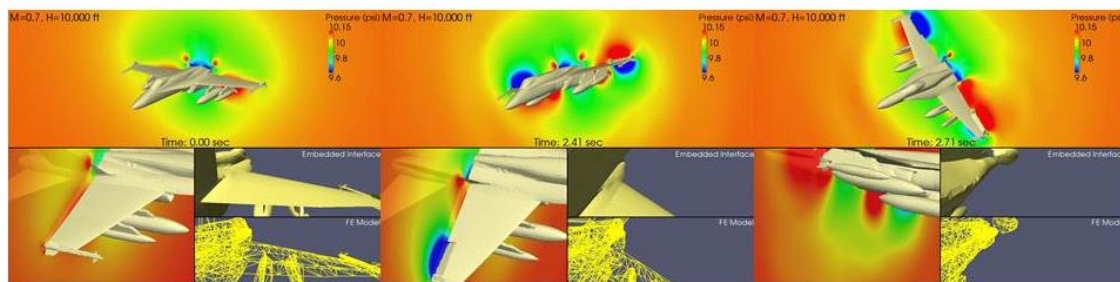


Figure 8. Snapshots from the coupled fluid-structure simulation of a high-speed roll maneuver using the Eulerian-ALE FIVER method and the Spalart-Allmaras turbulence model

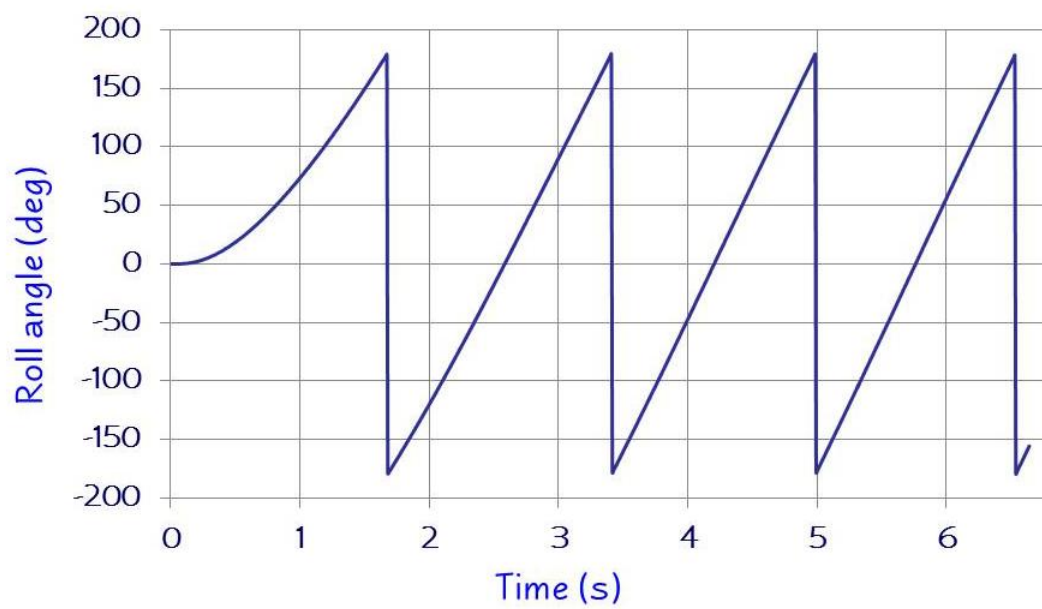


Figure 9. Roll damping predicted using the Eulerian-ALE FIVER method and the Spalart-Allmaras turbulence model