IN PARTNERSHIP WITH:
Université de Bordeaux

Activity Report 2019

Project-Team MEMPHIS

Modeling Enablers for Multi-PHysics and InteractionS

RESEARCH CENTER
Bordeaux - Sud-Ouest

THEME
Numerical schemes and simulations

Table of contents

Project-Team MEMPHIS

Creation of the Team: 2015 January 01, updated into Project-Team: 2016 October 01

Keywords:

Computer Science and Digital Science:

A6. - Modeling, simulation and control
A6.1.1. - Continuous Modeling (PDE, ODE)
A6.1.5. - Multiphysics modeling
A6.2.1. - Numerical analysis of PDE and ODE
A6.3.1. - Inverse problems
A6.3.2. - Data assimilation
A6.3.4. - Model reduction
A6.5.1. - Solid mechanics
A6.5.2. - Fluid mechanics
A9.2. - Machine learning

Other Research Topics and Application Domains:

B2.2.1. - Cardiovascular and respiratory diseases
B4.2. - Nuclear Energy Production
B4.3.2. - Hydro-energy
B4.3.3. - Wind energy
B5.2.3. - Aviation
B5.2.4. - Aerospace
B5.5. - Materials

1. Team, Visitors, External Collaborators

Research Scientists

Michel Bergmann [Inria, Researcher, HDR]
Tommaso Taddei [Inria, Researcher]

Faculty Members

Angelo Iollo [Team leader, Univ de Bordeaux, Professor, HDR]
Afaf Bouharguane [Univ de Bordeaux, Associate Professor]
Charles-Henri Bruneau [Univ de Bordeaux, Professor]

Post-Doctoral Fellow

Lei Zhang [Inria, Post-Doctoral Fellow, from Oct 2019]

PhD Students

Luis Henrique Benetti Ramos [ONERA, PhD Student]
Mathias Braun [IRSTEA, PhD Student, until Feb 2019]
Michele Giuliano Carlino [Inria, PhD Student]
Antoine Fondaneche [Univ de Bordeaux, PhD Student]
Baptiste Lambert [Inria, PhD Student, until Jan 2019]
Guillaume Ravel [Univ de Bordeaux, PhD Student]
Sebastien Riffaud [Univ de Bordeaux, PhD Student]
Giulia Sambataro [ANDRA, PhD Student, from Oct 2019]

Technical staff

Gwladys Ravon [Inria, Engineer, from Mar 2019]

Intern and Apprentice

Matteo Fois [Inria, from Sep 2019]

Administrative Assistant

Anne-Laure Gautier [Inria, Administrative Assistant]

Visiting Scientist

Victor Zucatti Da Silva [Universidade Estadual de Campinas, from Nov 2019]

External Collaborator

Mejdi Azaiez [Institut National Polytechnique de Bordeaux, from Sep 2019]

2. Overall Objectives

2.1. Multi-physics numerical modeling

We aim at a step change in multi-physics numerical modeling by developing two fundamental enablers:

- **reduced-order models**;
- **hierarchical Cartesian schemes**.

Reduced-order models (ROMs) are simplified mathematical models derived from the full set of PDEs governing the physics of the phenomenon of interest. ROMs can be obtained exploiting first principles or be data-driven. With ROMs one trades accuracy for speed and scalability, and counteracts the curse of dimensionality of traditional high-fidelity solvers by significantly reducing the computational complexity. ROMs represent an ideal building block for systems with real-time requirements, like interactive decision support systems that offer the possibility to rapidly explore various alternatives.

Hierarchical Cartesian schemes allow the multi-scale solution of PDEs on non body-fitted meshes with a drastic reduction of the computational setup overhead. These methods are easily parallelizable and they can efficiently be mapped to high-performance computer architectures. They avoid dealing with grid generation, a prohibitive task when the boundaries are moving and the topology is complex and unsteady.

3. Research Program

3.1. Reduced-order models

Massive parallelization and rethinking of numerical schemes will allow the use of mathematical models for a broader class of physical problems. For industrial applications, there is an increasing need for rapid and reliable numerical simulators to tackle design and control tasks. To provide a concrete example, in the design process of an aircraft, the flight conditions and manoeuvres, which provide the largest aircraft loads, are not known *a priori*. Therefore, the aerodynamic and inertial forces are calculated for a large number of conditions to give an estimate of the maximum loads, and hence stresses, that the structure of the detailed aircraft design might experience in service. As a result, the number of simulations required for a realistic design problem could easily be in the order of tens of millions. Even with simplistic models of the aircraft behavior this is an unfeasible number of separate simulations. However, engineering experience is used to identify the most likely critical load conditions, meaning that approximately hundreds of thousands simulations are required for conventional aircraft configurations. Furthermore, these analyses have to be repeated every time that there is an update in the aircraft structure.

Compared to existing approaches for ROMs [35], our interest will be focused on two axes. On the one hand, we start from the consideration that small, highly nonlinear scales are typically concentrated in limited spatial regions of the full simulation domain. So for example, in the flow past a wing, the highly non-linear phenomena take place in the proximity of the walls at the scale of a millimeter, for computational domains that are of the order of hundreds of meters. Based on these considerations, we propose in [31] a multi-scale model where the large scales are described by far-field models based on ROMs and the small scales are simulated by high-fidelity models. The whole point for this approach is to optimally decouple the far field from the near field.

A second characterizing feature of our ROM approach is non-linear interpolation. We start from the consideration that dynamical models derived from the projection of the PDE model in the reduced space are neither stable to numerical integration nor robust to parameter variation when hard non-linear multi-scale phenomena are considered.

However, thanks to Proper Orthogonal Decomposition (POD) [41], [47], [30] we can accurately approximate large solution databases using a low-dimensional base. Recent techniques to investigate the temporal evolution of the POD modes (Koopman modes [42], [28], Dynamic Mode Decomposition [45]) and allow a dynamic discrimination of the role played by each of them. This in turn can be exploited to interpolate between modes in parameter space, thanks to ideas relying on optimal transportation [50], [32] that we have started developing in the FP7 project FFAST and H2020 AEROGUST.

3.2. Hierarchical Cartesian schemes

We intend to conceive schemes that will simplify the numerical approximation of problems involving complex unsteady objects together with multi-scale physical phenomena. Rather than using extremely optimized but non-scalable algorithms, we adopt robust alternatives that bypass the difficulties linked to grid generation. Even if the mesh problem can be tackled today thanks to powerful mesh generators, it still represents a severe difficulty, in particular when highly complex unsteady geometries need to be dealt with. Industrial experience and common practice shows that mesh generation accounts for about 20% of overall analysis time, whereas creation of a simulation-specific geometry requires about 60%, and only 20% of overall time is actually devoted to analysis. The methods that we develop bypass the generation of tedious geometrical models by automatic implicit geometry representation and hierarchical Cartesian schemes.

The approach that we plan to develop combines accurate enforcement of unfitted boundary conditions with adaptive octree and overset grids. The core idea is to use an octree/overset mesh for the approximation of the solution fields, while the geometry is captured by level set functions [46], [40] and boundary conditions are imposed using appropriate interpolation methods [27], [49], [44]. This eliminates the need for boundary-conforming meshes that require time-consuming and error-prone mesh generation procedures, and opens the door for simulation of very complex geometries. In particular, it will be possible to easily import the industrial geometry and to build the associated level set function used for simulation.

Hierarchical octree grids offer several considerable advantages over classical adaptive mesh refinement for body-fitted meshes, in terms of data management, memory footprint and parallel HPC performance. Typically, when refining unstructured grids, like for example tetrahedral grids, it is necessary to store the whole data tree corresponding to successive subdivisions of the elements and eventually recompute the full connectivity graph. In the linear octree case that we develop, only the tree leaves are stored in a linear array, with a considerable memory advantage. The mapping between the tree leaves and the linear array as well as the connectivity graph is efficiently computed thanks to an appropriate space-filling curve. Concerning parallelization, linear octrees guarantee a natural load balancing thanks to the linear data structure, whereas classical unstructured meshes require sophisticated (and moreover time consuming) tools to achieve proper load distribution (SCOTCH, METIS etc.). Of course, using unfitted hierarchical meshes requires further development and analysis of methods to handle the refinement at level jumps in a consistent and conservative way, accuracy analysis for new finite-volume or finite-difference schemes, efficient reconstructions at the boundaries to recover appropriate accuracy and robustness. These subjects, that are currently virtually absent at Inria, are among the main scientific challenges of our team.

4. Application Domains

4.1. Energy conversion

4.1.1. Fluid-structure interaction

We apply the methods developed in our team to the domain of wind engineering and sea-wave converters. In Figure 1, we show results of a numerical model for a sea-wave energy converter. We here rely on a monolithic model to describe the interaction between the rigid floater, air and water; material properties such as densities, viscosities and rigidity vary across the domain. The appropriate boundary conditions are imposed at interfaces that arbitrarily cross the grid using adapted schemes built thanks to geometrical information computed via level set functions [46]. The background method for fluid-structure interface is the volume penalization method [27] where the level set functions is used to improve the degree of accuracy of the method [3] and also to follow the object. The underlined mathematical model is unsteady, and three dimensional; numerical simulations based on a grid with $\mathcal{O}(10^8)$ degrees of freedom are executed in parallel using 512 CPUs .

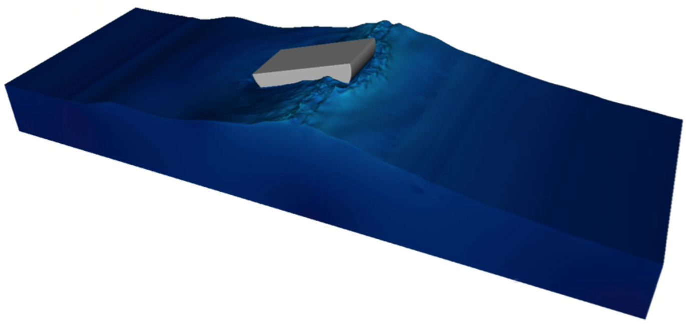

Figure 1. numerical modeling of a sea-wave converter by a monolithic model and Cartesian meshes.

In the context of the Aerogust (Aeroelastic gust modelling) European project, together with Valorem, we investigated the behavior of wind turbine blades under gust loading. The aim of the project was to optimize the design of wind turbine blades to maximize the power extracted. A meteorological mast (Figure 2(a)) has been installed in March 2017 in Brittany to measure wind on-site: data provided by the mast have been exploited to initialize the mathematical model. Due to the large cost of the full-order mathematical model, we relied on a simplified model [38] to optimize the global twist. Then, we validated the optimal configuration using the full-order Cartesian model based on the NaSCar solver. Figure 2(b) shows the flow around the optimized optimized wind turbine rotor.

4.1.2. Schemes for turbulent flow simulations using Octrees

We have initially developed and tested a 3D first-order Octree code for unsteady incompressible Navier-Stokes equations for full windmill simulations with an LES model and wall laws. We have validated this code on Occigen for complex flows at increasing Reynolds numbers. This step implied identifying stable and feasible schemes compatible with the parallel linear Octree structure. The validation has been conducted with respect to the results of a fully Cartesian code (NaSCAR) that we run on Turing (with significantly more degrees of freedom) and with respect to experimental results.

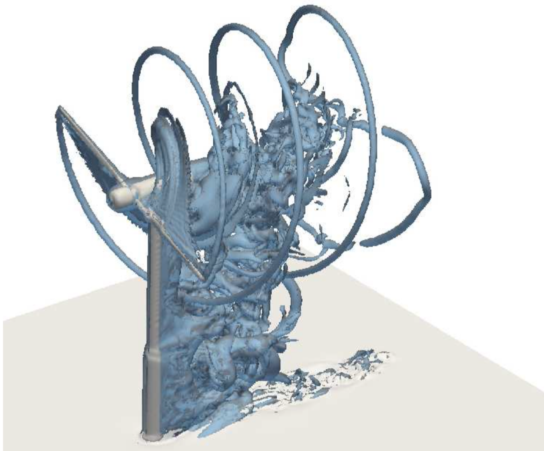

Figure 2. Aerogust project. Left: met mast after its installation. Right: flow around the optimized wind turbine rotor (as predicted by NaSCar).

Subsequently, we have developed a second-order Octree scheme that has been validated on Occigen for a sphere at a moderate Reynolds number ($\mathrm{Re} = 500$), see Table 1. Then, for a cylinder at ($\mathrm{Re} = 140000$) (Figures 3(a) and 3(b)), close to real applications, we have preliminary validation results for the second-order scheme with respect to experimental drag coefficient (Table 2). Additional resources will be asked on Occigen to complete the study.

Table 1. Flow past a sphere at Re =500. Results in the literature are spread between C D =0.48 and C D =0.52.

Mesh	$\Delta x_{\min}$	number of cells	C_{D} (1^{st}-order scheme)	C_{D} (2^{nd}-order scheme)
1	0.094	$0.72 \cdot 10^5$	N.A.	0.526
2	0.047	$4.9 \cdot 10^5$	0.595	0.522
3	0.023	$4.7 \cdot 10^6$	0.546	0.492
4	0.012	$37.6 \cdot 10^6$	0.555	0.496

Table 2. Flow past a sphere at Re =14000.

Case	C_{D}
Octree, 1^{st}-order scheme	1.007
Octree, 2^{nd}-order scheme	1.157
Cartesian	1.188
Experimental estimate [34]	1.237

4.2. Vascular flows

A new research direction pursued by the team is the mathematical modelling of vascular blood flows in arteries. Together with the start-up Nurea (http://nurea-soft.com/) and the surgeon Eric Ducasse, we aim at developing reliable and automatic procedures for aneurysm segmentation and for the prediction of aneurysm rupture

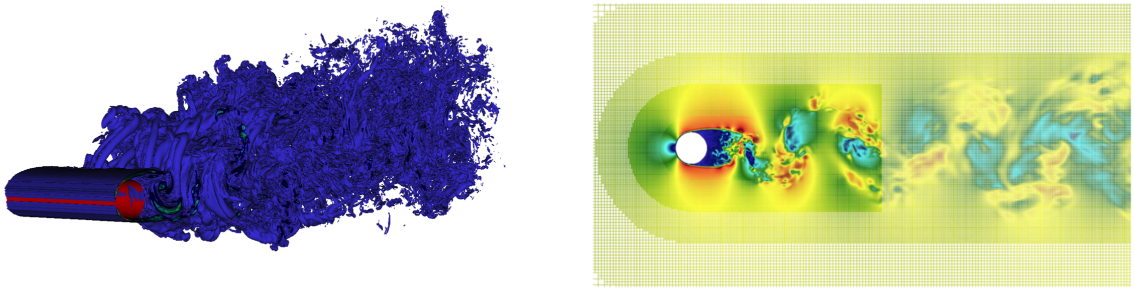

Figure 3. flow past a cylinder at $\mathrm{Re} = 140000$*. Left: vorticity contour lines. Right: streamwise velocity section and grid for the second-order Octree scheme.*

risk. Our approach exploits two sources of information: (i) numerical simulations of blood flows in complex geometries, based on an octree discretization, and (ii) computed tomography angiography (CTA) data. Figure 4 shows the force distribution on the walls of the abdominal aorta in presence of an aneurysm; results are obtained using a parallelized hierarchical Cartesian scheme based on octrees.

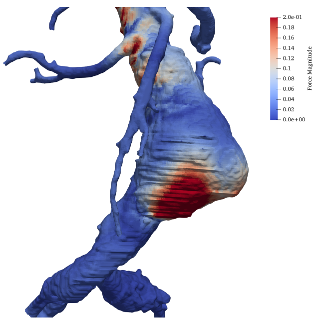

Figure 4. force distribution on the walls of the abdominal aorta in presence of an aneurysm.

4.3. Eulerian non-linear elasticity models

Mathematical and numerical modelling of continuum systems undergoing extreme regimes is challenging due to the presence of large deformations and displacements of the solid part, and due to the strongly non-linear behaviour of the fluid part. At the same time, proper experiments of impact phenomena are particularly dangerous and require expensive facilities, which make them largely impractical. For this reason, there is a growing interest in the development of predictive models for impact phenomena.

In MEMPHIS, we rely on a fully Eulerian approach based on conservation laws, where the different materials are characterized by their specific constitutive laws, to address these tasks. This approach was introduced in [37] and subsequently pursued and extended in [43], [36], [29], [33] In Figure 5, we show the results of the numerical simulation of the impact of a copper projectile immersed in air over a copper shield. Results are obtained using a fully parallel monolithic Cartesian method, based on a 4000^2 fixed Cartesian grid. Simulations are performed on a cluster of 512 processors, and benefits from the isomorphism between grid partitioning and processor topology.

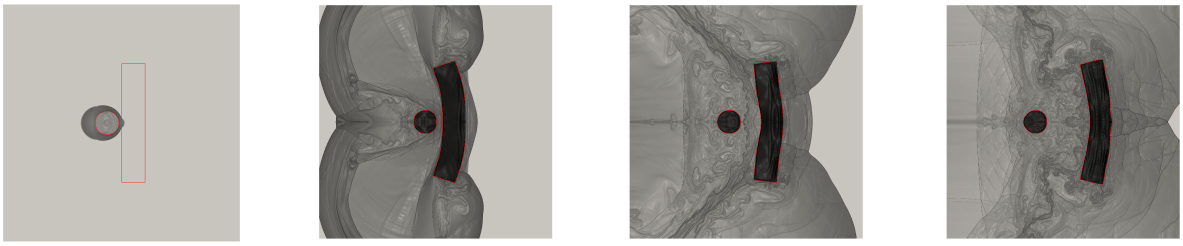

Figure 5. impact and rebound of a copper projectile on a copper plate. Interface and schlieren at $50\mu s$, $199\mu s$, $398\mu s$ and $710\mu s$.

5. Highlights of the Year

5.1. Highlights of the Year

5.1.1. 3D Numerical Model of a Zebra Fish Larva

The full reconstruction of a 3D larval zebrafish (5 days post fertilization) was realized using a serial-section electron microscopy data set combined with the technique of level-set and optimal transportation for shape interpolation. From an experimental video of zebrafish escape swimming, the kinematics of the swimming is extracted removing both translational and rotating displacements. Based on this video-extracted body deformation, 3D zebrafish snapshots of the body surface were generated deforming the 3D model according to the midline motion. The escape response of the zebrafish larva has been simulated using the NaSCar solver. The numerical simulation of the hydrodynamic zebrafish-locomotion provides a full range of the energetic performance performed by the larva during an escape response that are used by the MRGM biology lab in Bordeaux for toxicology evaluations. See figures 6 and 7.

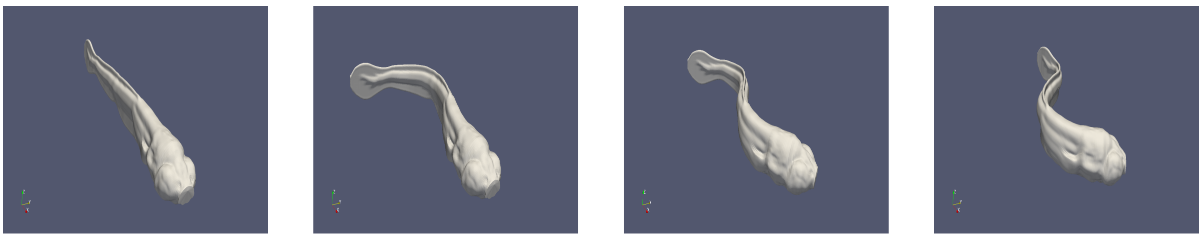

Figure 6. Snaphshots from left to right: reconstruction from electron microscopy and experimental video provided by MRGM Bordeaux of a zebra fish larva swimming movement.

6. New Software and Platforms

6.1. COCOFLOW

KEYWORDS: 3D - Elasticity - MPI - Compressible multimaterial flows

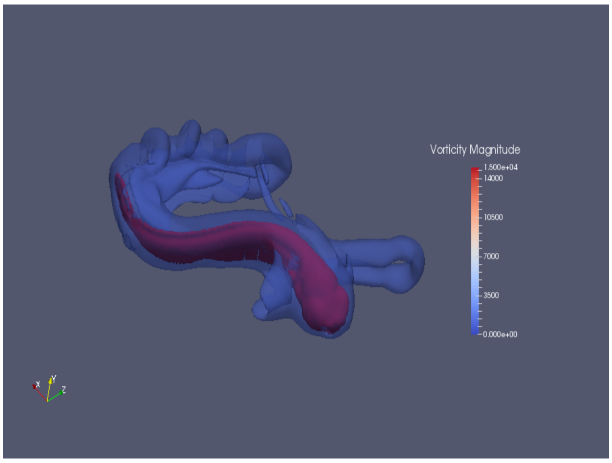

Figure 7. Numerical simulation of the swimming displacement of a zebra fish larva.

FUNCTIONAL DESCRIPTION: The code is written in fortran 95 with a MPI parallelization. It solves equations of conservation modeling 3D compressible flows with elastic models as equation of state.

- Authors: Alexia De Brauer, Florian Bernard, Yannick Gorsse, Thomas Milcent and Angelo Iollo
- Partners: CNRS - Université Bordeaux 1
- Contact: Florian Bernard
- URL: https://gforge.inria.fr/projects/cocoflow

6.2. KOPPA

Kinetic Octree Parallel PolyAtomic

KEYWORDS: C++ - 3D - MPI

FUNCTIONAL DESCRIPTION: KOPPA is a C++/MPI numerical code solving a large range of rarefied flows from external to internal flows in 1D, 2D or 3D. Different kind of geometries can be treated such as moving geometries coming from CAO files or analytical geometries. The models can be solved on Octree grids with dynamic refinement.

- Participant: Florian Bernard
- Partners: Université de Bordeaux - INP Bordeaux - CNRS
- Contact: Angelo Iollo
- URL: https://git.math.cnrs.fr/gitweb/?p=plm/fbernard/KOPPA.git;a=summary

6.3. NaSCar

Navier-Stokes Cartesian

KEYWORDS: HPC - Numerical analyse - Fluid mechanics - Langage C - PETSc

SCIENTIFIC DESCRIPTION: NaSCar can be used to simulate both hydrodynamic bio-locomotion as fish like swimming and aerodynamic flows such wake generated by a wind turbine.

FUNCTIONAL DESCRIPTION: This code is devoted to solve 3D-flows in around moving and deformable bodies. The incompressible Navier-Stokes equations are solved on fixed grids, and the bodies are taken into account thanks to penalization and/or immersed boundary methods. The interface between the fluid and the bodies is tracked with a level set function or in a Lagrangian way. The numerical code is fully second order (time and space). The numerical method is based on projection schemes of Chorin-Temam's type. The code is written in C language and use Petsc library for the resolution of large linear systems in parallel.

NaSCar can be used to simulate both hydrodynamic bio-locomotion as fish like swimming and aerodynamic flows such wake generated by a wind turbine.

- Participant: Michel Bergmann
- Partner: Université de Bordeaux
- Contact: Michel Bergmann
- URL: https://gforge.inria.fr/projects/nascar/

6.4. NS-penal

Navier-Stokes-penalization

KEYWORDS: 3D - Incompressible flows - 2D

FUNCTIONAL DESCRIPTION: The software can be used as a black box with the help of a data file if the obstacle is already proposed. For new geometries the user has to define them. It can be used with several boundary conditions (Dirichlet, Neumann, periodic) and for a wide range of Reynolds numbers.

- Partner: Université de Bordeaux
- Contact: Charles-Henri Bruneau

7. New Results

7.1. DGDD Method for Reduced-Order Modeling of Conservation Laws

Reduced-order models are attractive method to decrease significantly the computational cost of the simulations. However, the ability of reduced-order models to accurately approximate solutions containing strong convection, sharp gradients or discontinuities can be challenging. The discontinuous Galerkin domain decomposition (DGDD) reduced model for systems of conservation laws couples at the discrete level sub-domains of high-fidelity polynomial approximation to regions of low-dimensional resolution as shown in figure 8.

In this approach, the high-dimensional model solves the equations where a given degree of accuracy is required, while the reduced-order model approximates the solution elsewhere. Since the high-dimensional model is used in a small part of the domain, the computational cost is significantly reduced. To perform the coupling, we develop a reduced-order model based on Proper Orthogonal Decomposition in the offline stage and on discontinuous Galerkin method in the online stage instead of the standard Galerkin method. In this way, the domain decomposition is applied transparently thought the numerical fluxes. We investigate the prediction of unsteady flows over a NACA 0012 airfoil. The results demonstrate the accuracy of the proposed method and the significant reduction of the computational cost. In the figures (9 and 10) we show examples of predictions obtained by the low-order model compared to the actual solutions as a function of the Mach number at infintiy and the angle of attack. In the last figure we present the overall space-time L^2 errors as a function of the low-dimensional space size.

7.2. Segmentation of aortic aneurism: collaboration with Nurea

Starting from February 2019, AMIES granted a one-year contract engineer in close collaboration with the Nurea start-up.

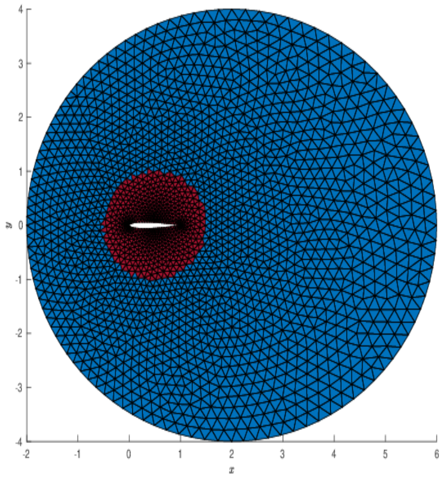

Figure 8. Decomposition of the domain: high-dimensional model (red), reduced-order model (blue).

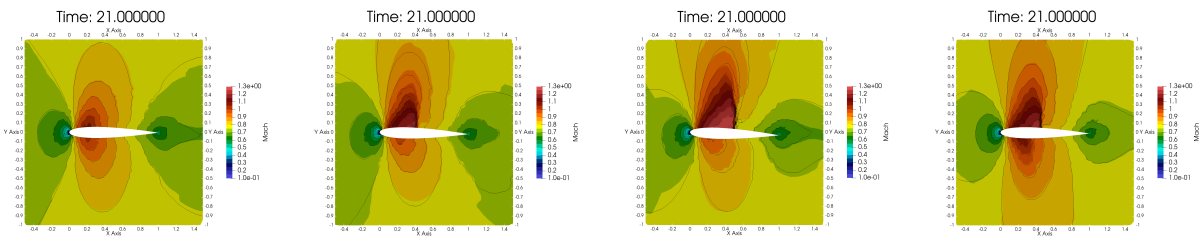

Figure 9. From left to right: $M_\infty = 0.754$ *and* $\alpha = 0.2$, $M_\infty = 0.763$ *and* $\alpha = 1.1$, $M_\infty = 0.776$ *and* $\alpha = 1.8$, $M_\infty = 0.784$ *and* $\alpha = 0.6$

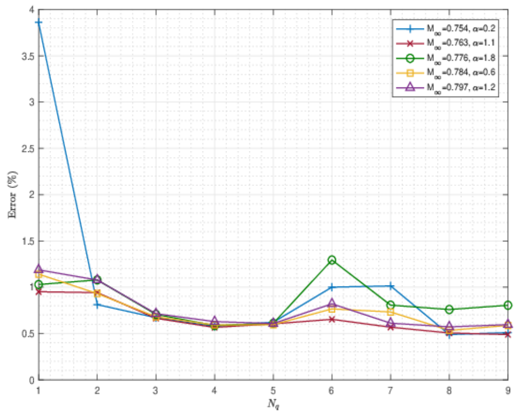

Figure 10. With $N_q = 9$ basis functions, ***the approximation error is less than 1% and the run time is reduced by approximately 72%*** *with respect to the high-fidelity solutions.*

The main objective of the project is to improve the quality and the robustness of automatic segmentation of aortic aneurism. An important part of the work was to decide if patient data needed to be pre-processed or not. To do so a criteron was developed to apply or not a smoothing filter. Several filters were tested and their performance were compared in order to choose the filter the more appropriate to our problem.

Another issue was the bones wrongly taken into the segmentation. A cleaning function was created to deal with it and remove the bones from the segmentation. An usual issue when working with medical images is to deal with the gradient of intensity. Existing tools need to be adapted to take into account these variations within images. This is currently worked out.

The code is implemented in C++ and mostly relies on itk and vtk libraries. An example in figure 11.

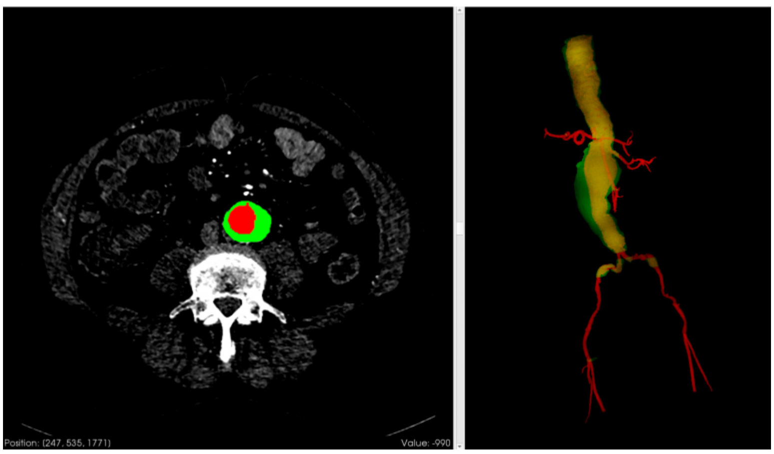

Figure 11. Example of aortic aneurism segmentation. Left: CT scan image; Right: 3D visualization. Red: blood; green: aortic wall

7.3. Fluid-structure interactions on AMR enabled quadree grids

We develop a versatile fully Eulerian method for the simulation of fluid-structure interactions. In the context of a monolithic approach, the whole system is modeled through a single continuum model. The equations are numerically solved using a finite-volume scheme with a compact stencil on AMR enabled quadtree grids where the dynamic refinement is adapted in time to the fluid-structure system.

The geometry is followed using a level-set formulation. In the Eulerian representation, a smooth Heaviside function is defined according to a level-set function on the cartesian mesh to distinguish between fluid and elastic phases. The temporal deformation of the structure is described according to the backward characteristics which are employed to express the Cauchy stress of a two-parameter hyperelastic Mooney-Rivlin material. This model is particularly adapted to elastomeric materials undergoing large deformations, see figure 12.

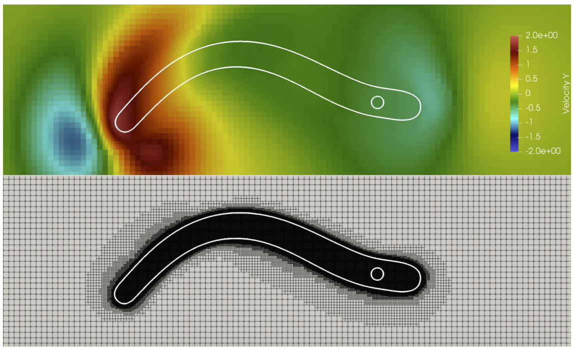

Figure 12. Y-component of the velocity for an oscillating elastomeric membrane actuated by a rigid holder at the tip, immersed in glycerin after 3 periods of oscillations. The criterion used for the dynamic AMR mesh is based on the level-set function.

7.4. Overset grids

One of the difficulties in the simulation of a fluid flow problem is the representation of the computational domain with a static mesh. As a matter of fact, not only the geometry could be particularly complex in itself, but it could change during the simulation and this necessary involves an *in itinere* geometrical adaptation of the mesh, with a consequent high computational cost. One of the ways to overcome this problem is to use multiple overlapping mesh blocks that together define a *Chimera* or *overset* grid. Once the different mesh blocks are generated, they are properly composed by the creation of holes (*hole cutting*) and, consequently, an *overlapping zone* between two overlapping blocks is defined. Figure 13 shows a Chimera grid in the computational domain $[-\pi, \pi]^2$; in black there is the background mesh, in blue the foreground mesh. In particular, the foreground mesh can move and deform (consequently, the hole in the background mesh can change its configuration). The overlapping zone is necessary for the the communication and data transfer from one mesh to another. These operations are possible through an appropriate definition of local stencils of cells, both within and at the border of the individual blocks.
The Navier-Stokes equations for incompressible flows are going to be approximated through a projection method (*Chorin-Temam*), for this reason we have studied two Finite Volume (FV) solvers, for the pure diffusive equation and the unsteady convective-diffusive equation, on Chimera configurations. The first numerical experiments were conducted on 2D problems. The order of convergence of the error of the mismatch between the exact solution and its FV approximation in L^∞- and L^2-norms is 2.

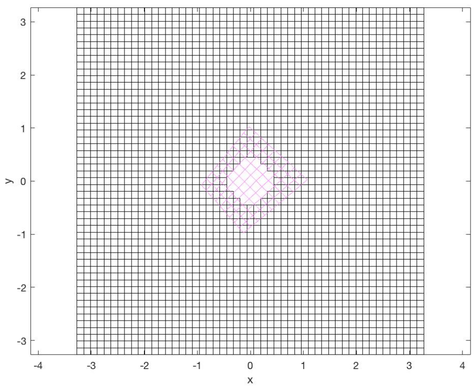

Figure 13. A Chimera grid configuration for the computational domain $[-\pi,\pi]^2$.

7.5. Collective propulsion: collaboration with ONERA

Motivated by recent studies and the locomotion of animal groups for robotics, we investigated the influence of hydrodynamic interactions on the collective propulsion of flapping wings. We studied the horizontal locomotion of an infinite array of flapping wings separated by a constant gap using unsteady non-linear simulations. Two control parameters were explored: the flapping frequency and the gap between the wings. Results obtained for different gaps at a fixed frequency are shown in Figure 14. We first observe that for a very large spacing between the wings — greater than 20 times the chord of a wing — the interaction effects are no longer present (Figure 14 (b)) and the average speed of the system tends to the speed of a single wing. For lower gaps, the average speed may become lower or higher than that of a single wing. For certain gaps, one can find two different stable solutions: one at higher propulsion speed and the other lower than a single wing. This phenomenon has already been observed for a fixed spacing and different frequencies of movement. The stability of these solutions is linked to the interaction between the vortex wake generated by the previous wings and the vortex ejected at the leading or trailing edge of the considered wing (Figure 14 (a)). We remark that propulsive efficiency is higher for the collective case both in faster and slower solutions. Understanding the key mechanisms responsible for the stable solutions will provide directions to control strategies aiming to optimize the wing horizontal speed.

7.6. Automatic registration for model reduction

As part of the ongoing team effort on ROMs, we work on the development of automatic registration procedures for model reduction. In computer vision and pattern recognition, registration refers to the process of finding a spatial transformation that aligns two datasets; in our work, registration refers to the process of finding a parametric transformation that improves the linear compressibility of a given parametric manifold. For advection-dominated problems, registration is motivated by the inadequacy of linear approximation spaces due to the presence of parameter-dependent boundary layers and travelling waves.

In [48], we proposed and analysed a computational procedure for stationary PDEs and investigated performance for two-dimensional model problems. In Figure 15, we show slices of the parametric solution for three different parameters before (cf. Left) and after (cf. Right) registration: we observe that the registration procedure is able to dramatically reduce the sensitivity of the solution to the parameter value μ. In Figure 16, we show the behaviour of the normalised POD eigenvalues (cf. Left) and of the relative L^2 error of the corresponding POD-Galerkin ROM (cf. Right) for an advection-reaction problem: also in this case, the approach

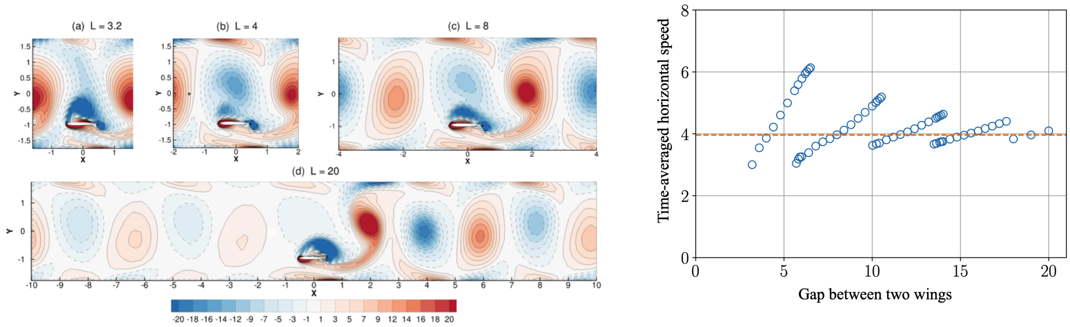

Figure 14. Vorticity contours (a) and time-averaged horizontal speed (b) for a flapping wing interacting in an infinite array. In (b) the average speed of a non interacting flapping wing is represented by a dashed orange line.

is able to improve the approximation properties of linear approximation spaces and ultimately simplify the reduction task.

We aim to extend the approach to a broad class of non-linear steady and unsteady PDEs: in October 2019, we funded a 16-month postdoc to work on the reduction of hyperbolic systems of PDEs. We are also collaborating with EDF (departments PERICLES and LNHE) to extend the approach to the Saint-Venant (shallow water) equations.

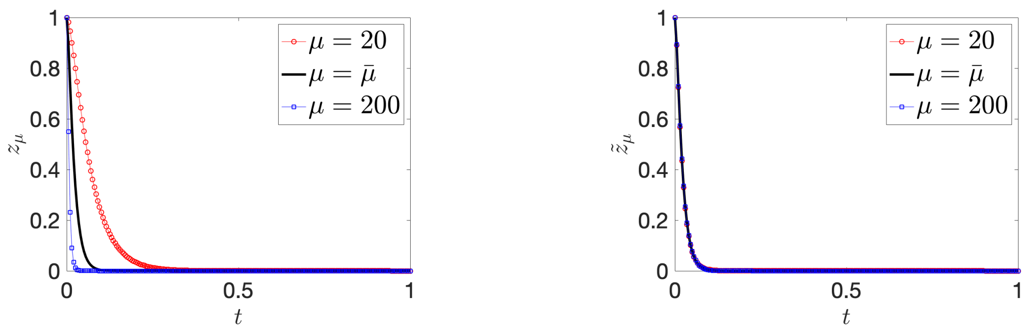

Figure 15. automatic registration for model reduction. Registration of a boundary layer ($\overline{\mu} = \sqrt{20 \cdot 200}$).

7.7. Modeling and numerical simulation of ellipsoidal particle-laden flows and self propelled swimmers in a porous enclosure

Despite being relevant in many natural and industrial processes, suspensions of non-spherical particles have been largely under-investigated compared to the extensive analyses made on the gravity-driven motions of spherical particles. One of the main reasons for this disparity is the difficulty of accurately correcting the short-range hydrodynamic forces and torques acting on complex particles. These effects, also known as lubrication, are essential to the suspension of the particles and are usually poorly captured by direct numerical simulation

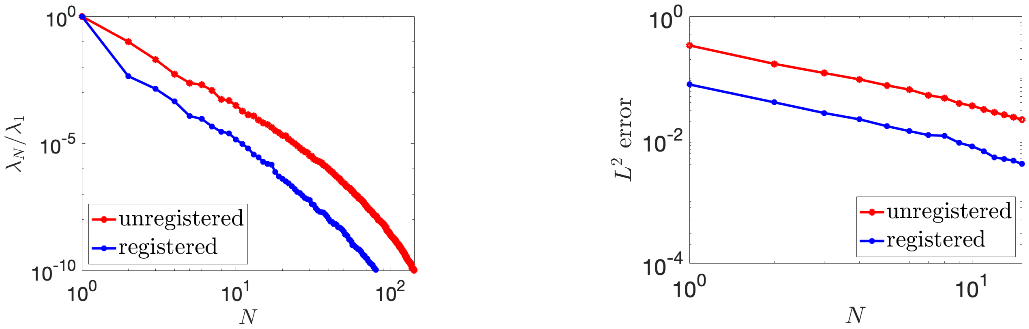

Figure 16. automatic registration for model reduction. Registration of a parameterized advection-reaction problem.

of particle-laden flows. We have proposed a partitioned VP-DEM (Volume Penalization method - Discrete Element Method) solver which estimates the unresolved hydrodynamic forces and torques. Corrections are made locally on the surface of the interacting particles without any assumption on the particle global geometry. This is an extension of our previous work [39]. Numerical validations have been made using ellipsoidal particles immersed in an incompressible Navier-Stokes flow.

Self organization of groups of several swimmers is of interest in biological applications. One of the main question is to determine if the possible organization comes from an uncontrolled or a controlled swimming behavior. This work has been motivated by the recent studies of Hamid Kellay (LOMA). Hamid Kellay has presented his results during a small workshop we organized earlier this year between members of the Memphis team (modeling and numerical methods), the MRGM (experimental zebra-fishes swimming), the LOMA (interaction between self-propelled particles) and the ONERA (flapping wings).

The collision model developed in the previous section has been developed for concave interactions like sphere-sphere or in the limit of sphere-plane wall. For non concave interactions, we have derived a simple approximation considering locally convexity as being a plane wall. We have performed numerical simulations of the interaction of several self propelled swimmers in a porous enclosure (see figure 17). Fishes are organized in small groups that are able to put into motion the enclosure. This behavior is similar to the one observed in the experimental set-up of Hamid Kellay.

8. Bilateral Contracts and Grants with Industry

8.1. Bilateral Contracts with Industry: EDF

40kEuro contract for a study on the development of projection-based reduction strategies for the shallow-water equations, for applications in Hydraulics.

8.2. Bilateral Grants with Industry: ANDRA

36kEuro contract for the development of a projection-based reduced model for a thermo-hydraulic-mechanical (THM) system.

9. Partnerships and Cooperations

9.1. National Initiatives

We are part of the GDR AMORE on ROMs.

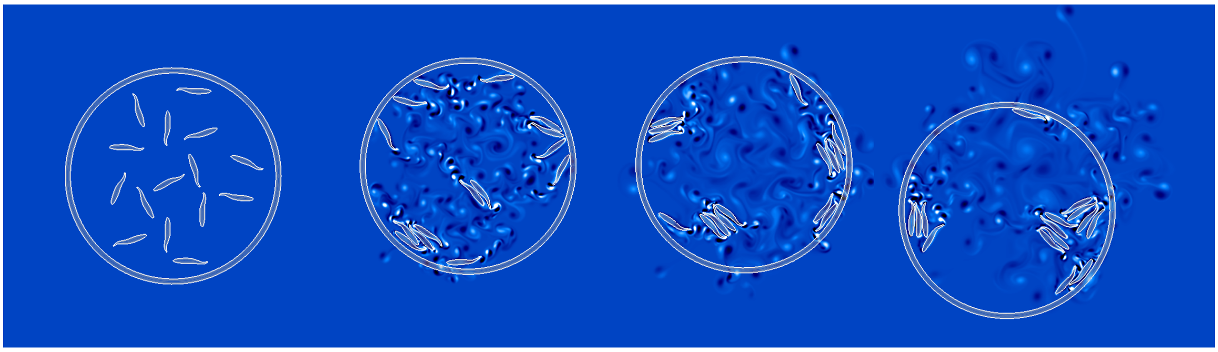

Figure 17. Organization of small self propelled fish groups in a porous enclosure (time evolution is in the usual reading direction). Colormap is the vorticity field.

9.2. European Initiatives

9.2.1. FP7 & H2020 Projects: ARIA RISE project

The overarching objective of ARIA (Accurate Roms for Industrial Applications) project is to form an international and intersectoral network of organizations working on a joint research program in numerical modelling, specifically in the fields of model reduction and convergence between data and models. Memphis team is ccordinating this 926KEuro project. 7 industrial partners are involved (VW, Valorem, Optimad, IEFluids, VirtualMech, Nurea, Esteco), 5 EU academic partners (Inria, Université de Seville, Poitecnico di Milano, Politecnico di Torino, SISSA) and 3 universities in the USA: Stanford University, Virginia Tech and University of South Carolina.

9.3. International Initiatives

9.3.1. Inria International Labs

Inria@SiliconValley
Associate Team involved in the International Lab:

9.3.1.1. MARE

Title: Multiscale Accurate Reduced-order model Enablers

International Partner (Institution - Laboratory - Researcher):

Stanford (United States) - VNU University of Engineering and Technology - Charbel Farhat

Start year: 2019

See also: https://team.inria.fr/memphis/mare-associate-team/

Reduced-order models (ROMs) are simplified mathematical models derived from the full set of partial differential equations governing the physics of the phenomenon of interest. We focus on ROMs that are data-driven as they are based on relevant solution data previously obtained. In particular we will focus on multiscale adaptive models where the large scales are governed by a PDE and the small scales are described by data driven models. To do that we will leverage on tools from data geometry, numerical PDEs and machine learning.

10. Dissemination

10.1. Promoting Scientific Activities

10.1.1. Reviewer - Reviewing Activities

Journal of Computational Physics, International Journal of CFD, Journal of Non-linear Analysis B, ASME Journal of Computational and Nonlinear Dynamics, Journal of Fluid Mechanics, Acta Mechanica, AIAA Journal, International Journal Numerical Methods in Fluids, Computers & Fluids, Journal of Engineering Mathematics, European Journal of Mechanics / B Fluids, Journal Européen de Systèmes Automatisés, Applied Mathematics and Computation. Nuclear Science and Engineering, Computer Methods in Applied Mechanics and Engineering, Journal of Theoretical Biology, Computational Optimization and Applications, Applied science, Meccanica, SIAM journal on scientific computing, SIAM journal on uncertainty quantification, Advances in Computational Mathematics.

10.1.2. Invited Talks

Angelo Iollo

1. June 5th, 2019. Journées scientifiques Inria, Lyon.

   https://project.inria.fr/journeesscientifiques2019/francais-programme/.
2. May 2019. CIMPA School, Tunis. Science des données pour l'ingénierie et la technologie.

   https://www.cimpa.info/fr/node/6217.
3. March 2019. Conférencier invité à la conférence « Fluid-structure interaction», Politecnico di Milano, Milano, 18/3-20/3/2019.

   http://www1.mate.polimi.it/ gazzola/fs.html.

Tommaso Taddei

1. November 2019. MORTECH 2019, Paris.

   https://mortech2019.sciencesconf.org/

10.1.3. Leadership within the Scientific Community

10.1.3.1. Scientific Expertise

Angelo Iollo is an expert for the European Union for the program FET OPEN.

10.2. Teaching - Supervision - Juries

10.2.1. Teaching

Four members of the team are Professors or Assistant Professors at Bordeaux University and have teaching duties, which consist in courses and practical exercises in numerical analysis and scientific computing. Michel Bergmann (CR) also teaches around 64 hours per year (practical exercises in programming for scientific computing). Tommaso Taddei (CR) also teaches around 50 hours per year (practical exercises in numerical analysis and scientific computing).

10.2.2. Supervision

1. 2019-2022. Giulia Sambataro. Bourse ANDRA. *Component-based reduction strategies for THM equations.* Advisors: Angelo Iollo, Tommaso Taddei.
2. 2018-2021. Michele Giuliano Carlino. Bourse Inria. *Fluid-structure models on Chimera grids.* Advisors: Michel Bergmann, Angelo Iollo.
3. 2018-2021. Antoine Fondanèche. Bourse UB. *Monolithic fluid-structure modeles on parallel hierarchical grids.* Advisors: Michel Bergmann, Angelo Iollo.
4. 2017-2020. Sebastien Riffaud. *Convergence between data and numerical models.* Advisor: Angelo Iollo.
5. 2017-2020. Luis Ramos Benetti. Bourse ERC Aeroflex (O. Marquet, ONERA). *Monolithic fluid-structure modeles on parallel hierarchical grids.* Advisors: Michel Bergmann, Angelo Iollo.

10.2.3. Juries

Angeo Iollo: reviewer of 3 PhD theses, president of one PhD jury, member of one PhD jury, in France and abroad.
Michel Bergmann: reviewer of 2 PhD theses.

11. Bibliography

Major publications by the team in recent years

[1] E. ABBATE, A. IOLLO, G. PUPPO. *An all-speed relaxation scheme for gases and compressible materials*, in "Journal of Computational Physics", 2017, vol. 351, pp. 1-24 [*DOI :* 10.1016/J.JCP.2017.08.052], https://hal.inria.fr/hal-01586863

[2] M. BERGMANN, C.-H. BRUNEAU, A. IOLLO. *Enablers for robust POD models*, in "Journal of Computational Physics", 2009, vol. 228, n^{o} 2, pp. 516–538

[3] M. BERGMANN, J. HOVNANIAN, A. IOLLO. *An accurate cartesian method for incompressible flows with moving boundaries*, in "Communications in Computational Physics", 2014, vol. 15, n^{o} 5, pp. 1266–1290

[4] M. BERGMANN, A. IOLLO. *Modeling and simulation of fish-like swimming*, in "Journal of Computational Physics", 2011, vol. 230, n^{o} 2, pp. 329 - 348

[5] M. BERGMANN, A. IOLLO. *Bioinspired swimming simulations*, in "Journal of Computational Physics", 2016, vol. 323, pp. 310 - 321

[6] F. BERNARD, A. IOLLO, G. PUPPO. *Accurate Asymptotic Preserving Boundary Conditions for Kinetic Equations on Cartesian Grids*, in "Journal of Scientific Computing", 2015, 34 p.

[7] A. BOUHARGUANE, A. IOLLO, L. WEYNANS. *Numerical solution of the Monge–Kantorovich problem by density lift-up continuation*, in "ESAIM: Mathematical Modelling and Numerical Analysis", November 2015, vol. 49, n^{o} 6, 1577

[8] A. DE BRAUER, A. IOLLO, T. MILCENT. *A Cartesian Scheme for Compressible Multimaterial Models in 3D*, in "Journal of Computational Physics", 2016, vol. 313, pp. 121-143

[9] F. LUDDENS, M. BERGMANN, L. WEYNANS. *Enablers for high-order level set methods in fluid mechanics*, in "International Journal for Numerical Methods in Fluids", December 2015, vol. 79, pp. 654-675

[10] T. MEUEL, Y. L. XIONG, P. FISCHER, C.-H. BRUNEAU, M. BESSAFI, H. KELLAY. *Intensity of vortices: from soap bubbles to hurricanes*, in "Scientific Reports", December 2013, vol. 3, pp. 3455 (1-7)

[11] Y. L. XIONG, C.-H. BRUNEAU, H. KELLAY. *A numerical study of two dimensional flows past a bluff body for dilute polymer solutions*, in "Journal of Non-Newtonian Fluid Mechanics", 2013, vol. 196, pp. 8-26

Publications of the year

Doctoral Dissertations and Habilitation Theses

[12] M. BRAUN. *Reduced Order Modelling and Uncertainty Propagation Applied to Water Distribution Networks*, Université de Bordeaux, April 2019, https://tel.archives-ouvertes.fr/tel-02278297

Articles in International Peer-Reviewed Journals

[13] E. ABBATE, A. IOLLO, G. PUPPO. *An asymptotic-preserving all-speed scheme for fluid dynamics and nonlinear elasticity*, in "SIAM Journal on Scientific Computing", September 2019, https://hal.archives-ouvertes.fr/hal-02373325

[14] E. ABBATE, A. IOLLO, G. PUPPO. *An implicit scheme for moving walls and multi-material interfaces in weakly compressible materials*, in "Communications in Computational Physics", January 2020, https://hal.archives-ouvertes.fr/hal-02373329

[15] S. AVGERINOS, F. BERNARD, A. IOLLO, G. RUSSO. *Linearly implicit all Mach number shock capturing schemes for the Euler equations*, in "Journal of Computational Physics", 2019 [*DOI :* 10.1016/J.JCP.2019.04.020], https://hal.inria.fr/hal-02419411

[16] F. BERNARD, A. IOLLO, G. PUPPO. *BGK Polyatomic Model for Rarefied Flows*, in "Journal of Scientific Computing", March 2019, vol. 78, n^{o} 3, pp. 1893-1916 [*DOI :* 10.1007/S10915-018-0864-X], https://hal.inria.fr/hal-02419447

[17] M. G. CARLINO, P. RICKA, M. S. PHAN, S. BERTOLUZZA, M. PENNACCHIO, G. PATANÈ, M. SPAGNUOLO. *Geometry description and mesh construction from medical imaging*, in "ESAIM: Proceedings and Surveys", 2019, vol. 2019, pp. 1 - 10, forthcoming, https://hal.inria.fr/hal-02072342

[18] A. FERRERO, A. IOLLO, F. LAROCCA. *Reduced order modelling for turbomachinery shape design*, in "International Journal of Computational Fluid Dynamics", November 2019, pp. 1-12 [*DOI :* 10.1080/10618562.2019.1691722], https://hal.inria.fr/hal-02403455

[19] M. JEDOUAA, C.-H. BRUNEAU, E. MAITRE. *An efficient interface capturing method for a large collection of interacting bodies immersed in a fluid*, in "Journal of Computational Physics", February 2019, vol. 378, pp. 143-177 [*DOI :* 10.1016/J.JCP.2018.11.006], https://hal.archives-ouvertes.fr/hal-01236468

[20] T. TADDEI. *An offline/online procedure for dual norm calculations of parameterized functionals: empirical quadrature and empirical test spaces*, in "Advances in Computational Mathematics", September 2019 [*DOI :* 10.1007/S10444-019-09721-W], https://hal.archives-ouvertes.fr/hal-02369312

Invited Conferences

[21] M. BERGMANN, A. IOLLO. *Sampling and clustering on the POD-Grassmann manifold*, in "CSE19 - SIAM Conference on Computational Science and Engineering", Spokane, United States, February 2019, https://hal.inria.fr/hal-02424383

[22] M. BERGMANN, A. IOLLO, S. RIFFAUD, A. FERRERO, A. SCARDIGLI, E. LOMBARDI, H. TELIB. *Reduced-Order Models: Convergence Between Data and Simulation*, in "CSE19 - SIAM Conference on Computational Science and Engineering", Spokane, United States, February 2019, https://hal.inria.fr/hal-02424387

International Conferences with Proceedings

[23] A. FERRERO, A. IOLLO, F. LAROCCA. *RANS closure approximation by artificialneural networks*, in "ETC 2019 - 13th European Turbomachinery Conference on Turbomachinery Fluid Dynamics and Thermodynamics", Lausanne, Switzerland, April 2019, https://hal.inria.fr/hal-02403432

Conferences without Proceedings

[24] M. BERGMANN, A. FONDANÈCHE, A. IOLLO. *AMR enabled quadtree discretization of incompressible Navier-Stokes equations with moving boundaries*, in "International Congress on Industrial and Applied Mathematics (ICIAM) 2019", Valencia, Spain, July 2019, https://hal.inria.fr/hal-02421748

Other Publications

[25] A. BARONE, M. G. CARLINO, A. GIZZI, S. PEROTTO, A. VENEZIANI. *Efficient Estimation of Cardiac Conductivities: a Proper Generalized Decomposition Approach*, December 2019, working paper or preprint, https://hal.inria.fr/hal-02417508

[26] T. TADDEI. *A registration method for model order reduction: data compression and geometry reduction*, January 2020, https://arxiv.org/abs/1906.11008 - working paper or preprint, https://hal.archives-ouvertes.fr/hal-02430234

References in notes

[27] P. ANGOT, C.-H. BRUNEAU, P. FABRIE. *A penalization method to take into account obstacles in a incompressible flow*, in "Numerische Mathematik", 1999, vol. 81, n^o 4, pp. 497-520

[28] S. BAGHERI. *Koopman-mode decomposition of the cylinder wake*, in "Journal of Fluid Mechanics", 2013

[29] P. BARTON, D. DRIKAKIS, E. ROMENSKI, V. TITAREV. *Exact and approximate solutions of Riemann problems in non-linear elasticity*, in "Journal of Computational Physics", 2009, vol. 228, n^o 18, pp. 7046-7068

[30] M. BERGMANN, C.-H. BRUNEAU, A. IOLLO. *Enablers for robust POD models*, in "Journal of Computational Physics", 2009, vol. 228, n^o 2, pp. 516–538

[31] M. BERGMANN, A. FERRERO, A. IOLLO, E. LOMBARDI, A. SCARDIGLI, H. TELIB. *A zonal Galerkin-free POD model for incompressible flows*, in "Journal of Computational Physics", 2018, vol. 352, pp. 301–325

[32] A. BOUHARGUANE, A. IOLLO, L. WEYNANS. *Numerical solution of the Monge-Kantorovich problem by density lift-up continuation*, in "ESAIM: M2AN", 2015, vol. 49, n^o 6, pp. 1577-1592

[33] A. D. BRAUER, A. IOLLO, T. MILCENT. *A Cartesian scheme for compressible multimaterial models in 3D*, in "Journal of Computational Physics", 2016, vol. 313, pp. 121-143 [*DOI :* 10.1016/J.JCP.2016.02.032], http://www.sciencedirect.com/science/article/pii/S0021999116000966

[34] B. CANTWELL, D. COLES. *An experimental study of entrainment and transport in the turbulent near wake of a circular cylinder*, in "Journal of fluid mechanics", 1983, vol. 136, pp. 321–374

[35] L. CORDIER, M. BERGMANN. *Two typical applications of POD: coherent structures eduction and reduced order modelling*, in "Lecture series 2002-04 on post-processing of experimental and numerical data", Von Kármán Institute for Fluid Dynamics, 2002

[36] S. GAVRILYUK, N. FAVRIE, R. SAUREL. *Modelling wave dynamics of compressible elastic materials*, in "Journal of Computational Physics", 2008, vol. 227, n^{o} 5, pp. 2941-2969

[37] S. GODUNOV. *Elements of continuum mechanics*, Nauka Moscow, 1978

[38] X. JIN. *Construction d'une chaîne d'outils numériques pour la conception aérodynamique de pales d'éoliennes*, Université de Bordeaux, 2014

[39] B. LAMBERT, L. WEYNANS, M. BERGMANN. *Local lubrication model for spherical particles within incompressible Navier-Stokes flows*, in "Phys. Rev. E", Mar 2018, vol. 97, 033313 p. , https://link.aps.org/doi/10.1103/PhysRevE.97.033313

[40] F. LUDDENS, M. BERGMANN, L. WEYNANS. *Enablers for high-order level set methods in fluid mechanics*, in "International Journal for Numerical Methods in Fluids", December 2015, vol. 79, pp. 654-675 [*DOI :* 10.1002/FLD.4070]

[41] J. LUMLEY, A. YAGLOM, V. TATARSKI. *Atmospheric turbulence and wave propagation*, in "The structure of inhomogeneous turbulence, AM Yaglom & VI Tatarski", 1967, pp. 166–178

[42] I. MEZIĆ. *Spectral Properties of Dynamical Systems, Model Reduction and Decompositions*, in "Nonlinear Dynamics", 2005, vol. 41, n^{o} 1 [*DOI :* 10.1007/S11071-005-2824-X]

[43] G. MILLER, P. COLELLA. *A Conservative Three-Dimensional Eulerian Method for Coupled Solid-Fluid Shock Capturing*, in "Journal of Computational Physics", 2002, vol. 183, n^{o} 1, pp. 26-82

[44] R. MITTAL, G. IACCARINO. *Immersed boundary methods*, in "Annu. Rev. Fluid Mech.", 2005, vol. 37, pp. 239-261

[45] P. J. SCHMID. *Dynamic mode decomposition of numerical and experimental data*, in "Journal of Fluid Mechanics", 008 2010, vol. 656, pp. 5-28 [*DOI :* 10.1017/S0022112010001217]

[46] J. A. SETHIAN. *Level Set Methods and Fast Marching Methods*, Cambridge University Press, Cambridge, UK, 1999

[47] L. SIROVICH. *Turbulence and the dynamics of coherent structures*, in "Quarterly of Applied Mathematics", 1987, vol. XLV, n^{o} 3, pp. 561-590

[48] T. TADDEI. *A registration method for model order reduction: data compression and geometry reduction*, in "arXiv preprint arXiv:1906.11008", 2019

[49] K. TAIRA, T. COLONIUS. *The immersed boundary method: a projection approach*, in "Journal of Computational Physics", 2007, vol. 225, n^{o} 2, pp. 2118-2137

[50] C. VILLANI. *Topics in optimal transportation*, 1st, American Mathematical Society, 2003