IN PARTNERSHIP WITH:
Université de Bordeaux

Activity Report 2017

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

Other Research Topics and Application Domains:

B1.1.9. - Bioinformatics
B2.2.1. - Cardiovascular and respiratory diseases
B4.3.2. - Hydro-energy
B4.3.3. - Wind energy
B5.2.1. - Road vehicles
B5.2.3. - Aviation
B5.2.4. - Aerospace
B5.5. - Materials
B8.4. - Security and personal assistance
B8.4.1. - Crisis management

1. Personnel

Research Scientist

Michel Bergmann [Inria, Researcher, HDR]

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]
Lisl Weynans [Univ de Bordeaux, Associate Professor, until May 2017]

Post-Doctoral Fellow

Andrea Ferrero [Inria, until Oct 2017]

PhD Students

Emanuela Abbate [Universita degli Studi dell'Insubria / Univ. de Bordeaux]
Luis Henrique Benetti Ramos [ONERA, from Sep 2017]
Mathias Braun [IRSTEA]
Meriem Jedouaa [Inria, until Jan 2017]
Baptiste Lambert [Univ de Bordeaux]
Claire Morel [VALEOL]
Alice Raeli [Univ de Bordeaux, until Sep 2017]
Guillaume Ravel [Univ de Bordeaux, from Oct 2017]

Sebastien Riffaud [Univ de Bordeaux, from Oct 2017]
Federico Tesser [Inria]

Technical staff

Florian Bernard [Inria]
Matias Hastaran [Inria, from Mar 2017]
Romain Leguay [Inria, from Jun 2017]

Interns

Samuel Agius [Inria, from May 2017 until Sep 2017]
Thibaut Guegan [Univ de Bordeaux, from May 2017 until Aug 2017]
Mahmoud Khabou [Inria, from Apr 2017 until Aug 2017]
Stefano Pezzano [Inria, from Sep 2017]
Sebastien Riffaud [Inria, from Feb 2017 until Aug 2017]

Administrative Assistant

Anne-Laure Gautier [Inria]

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 derived from first principles or be data-driven. With ROMs one trades accuracy for speed and scalability, and counteracts the curse of dimension by significantly reducing the computational complexity. ROMs represent an ideal building block of 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. 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 [37], [31] and boundary conditions are imposed using appropriate interpolation methods [22], [39], [35]. 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 non-structured 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 presently virtually absent at Inria, are among the main scientific challenges of our team.

3.2. Reduced-order models

Massive parallelization and rethinking of numerical schemes will allow the solution of new problem in physics and the prediction of new phenomena thanks to simulation. However, in industrial applications fast on line responses are needed for design and control. For instance, 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 at a large number of conditions to give an estimate of the maximum loads, and hence stresses, that the structure of the detailed aircraft design will experience in service. A simplistic estimate of the number of analyses required would multiply the numbers of conditions to give 10^7. 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 loads conditions, meaning that approximately 10^5 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 [28], our interest will be focused on two axis. On the one hand, we start from the consideration that small, highly non-linear 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 close to the walls at the scale of a millimeter for computational domains that are of the order of hundreds of meters. In this context our approach is characterized by 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) [32], [38], [25] we can accurately approximate large solution databases using a small base. Recent techniques to investigate the temporal evolution of the POD modes (Koopman modes [33], [23], Dynamic Mode Decomposition [36]) allow a dynamic discrimination of the role played by each of them. This in turn can be exploited to interpolate between the modes in parameter space, thanks to ideas relying on optimal transportation [40], [27] that we have started developing in the FP7 project FFAST and H2020 AEROGUST. In the following we precise these ideas on a specific example.

4. Application Domains

4.1. Energy conversion

We consider applications in the domain of wind engineering and sea-wave converters. As an example of application of our methods, we show a recent realization where we model a sea-wave energy converter, see figure 1. In this unsteady example, the full interaction between the rigid floater, air and water is described by a monolithic model, the Newton's law, where physical parameters 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 [37]. The background method for fluid structure interface is the volume penalization method [22] where the level set functions is used to improve the degree of accuracy of the method [26] and also to follow the object. The simulations are unsteady, three dimensional, with $O(10^8)$ grid points on 512 CPUs.

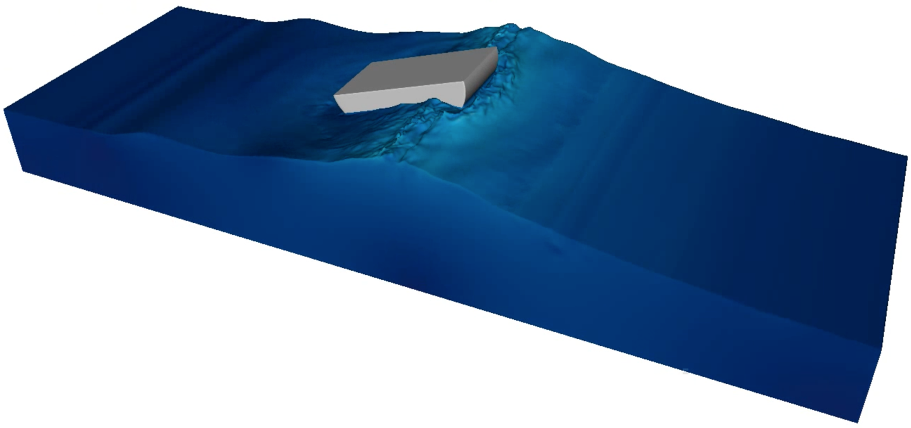

Figure 1. Numerical modeling of a sea-wave converter by a monolithic model and Cartesian meshes.

4.2. Impacts

The numerical modelling of multimaterial rapid dynamics in extreme conditions is an important technological problem for industrial and scientific applications. Experiments are dangerous, need heavy infrastructures and hence are difficult and expensive to realize. The simulation of such phenomena is challenging because they couple large deformations and displacements in solids to strongly non-linear behaviour in fluids. In what follows, we privilege a fully Eulerian approach based on conservation laws, where the different materials are characterized by their specific constitutive laws. This approach was introduced in [30] and subsequently pursued and extended for example in [34], [29], [24], [41].

We study hyper-velocity phenomena where several materials are involved. An example of this approach is the impact of a projectile immersed in air over a shield, see figure 2. Using the same set of equations across the entire domain, we model the compressible fluid, the hyperelastic material and the interaction at the interface that models possible rebounds. Only the constitutive laws characterize the different materials.

The simulation is performed over a 4000^2 fixed Cartesian grid so that the resulting numerical scheme allows an efficient parallelization (512 processors in this case) with an isomorphism between grid partitioning and processor topology. The challenge for our team is to increase the accuracy of the simulation thanks to grid refinement in the vicinity of the moving interfaces, still guaranteeing scalability and a simple computational set up.

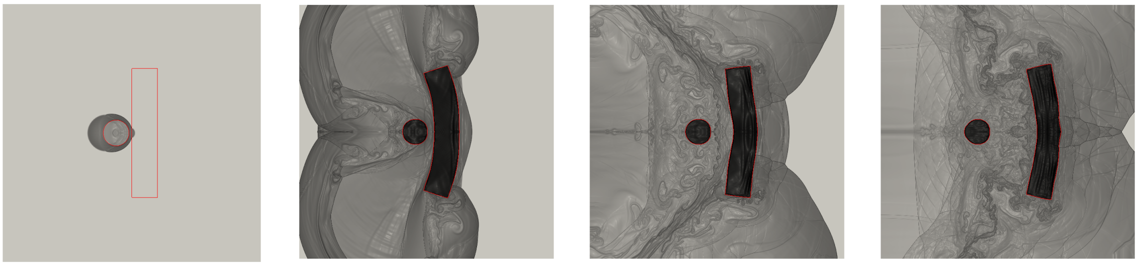

Figure 2. Impact and rebound of a copper projectile on a copper plate. Interface and schlieren at $50\mu s$, $199\mu s$, $398\mu s$ and $710\mu s$. From left to right, top to bottom.

4.3. New materials

Thanks to the multi-scale schemes that we develop, we can characterize new materials from constituents. As an example, consider the material presented in figure 3 left. It is a picture of a dry foam that is used as dielectric material. This micrography is taken at the scale of the dry bubbles, where on the surface of the bubble one can observe the carbon nanotubes as white filaments. The presence of nanotubes in the dry emulsion makes the electrical capacitance of this material significantly affected by its strain state by creating aligned dipoles at a larger scale compared to the size of the dielectric molecules. It is a typical multi-scale phenomenon in presence of widely varying physical properties. This material is used to generate micro currents when it undergoes vibrations. The schemes that we devise allow to model this multi-scale irregular material by a monolithic model (same equation in the whole domain), in this case a variable coefficient diffusion equation. In order to recover adequate accuracy, the numerical scheme is adapted near the interfaces between the different subdomains. The computational hierarchical mesh is directly derived by the micrography of the material (figure 3 right).

4.4. Bio-inspired robotic swimming

In bioinspired robotic swimming the aim is of simulating a three-dimensional swimmer starting from pictures. The first step is to build the three-dimensional fish profile based on two-dimensional data retrieved from the picture of an undeformed fish at rest. This is done by a skeleton technique and a three-dimensional level set function describing the body surface. Then the skeleton is deformed using an appropriate swimming law to obtain a sequence of level set functions corresponding to snapshots of the body surface uniformly taken at different instants.

Thanks to skeleton deformation we typically reconstruct 20% of the snapshots necessary to simulate a swimming stroke, since the time scale of the simulation is significantly smaller than the time step between two subsequent reconstructed snapshots. Also, the surface deformation velocity is required to set the boundary conditions of the flow problem. For this reason it is necessary to build intermediate level set functions and to compute the deformation velocity field between subsequent fish snapshots. Optimal transportation is well suited to achieve this goal providing an objective model to compute intermediate geometries and deformation velocities.

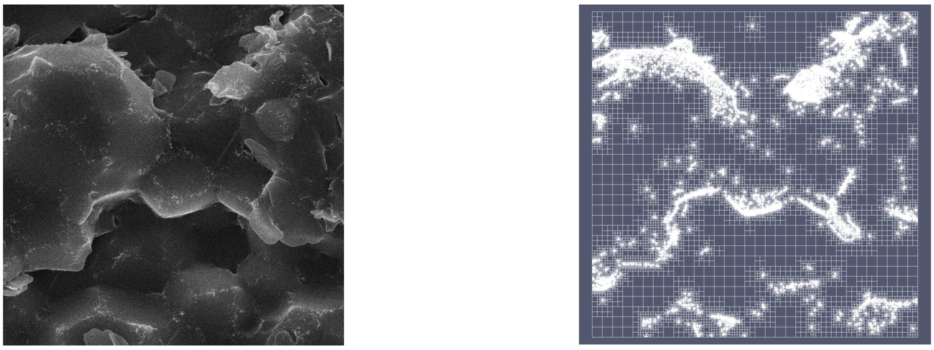

Figure 3. A micrography of an electrostrictive material is shown on the left: the bright regions visualize the carbon nanotubes. The hierarchical grid adapted to the nanotubes is shown on the right. The ratio between the largest and the smallest cell side is 2^7. Project developed in collaboration with the CRPP physics and chemistry lab of the CNRS in Bordeaux (Annie Colin, Philippe Poulin).

Numerical simulations have been performed in 3D, see figure 4. However, it has been observed that these algorithms do not preserve the physics/features of the represented objects. Indeed, the fish tends to compress during the deformation.

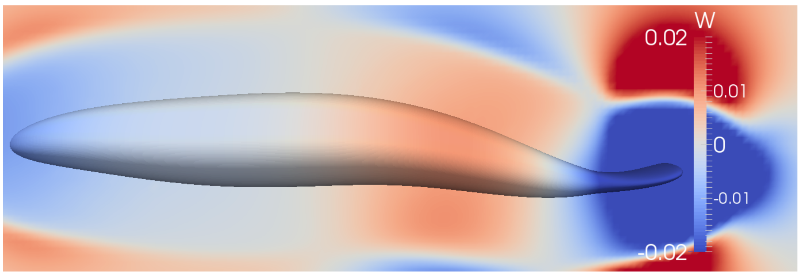

Figure 4. Comparison of the exact deformation velocity (presented inside the swimmer) and the approximated velocity identified using optimal transport (represented outside the fish). The error of the identification scheme is negligible for this component of the velocity, as it can be inferred by comparing the two velocities on the boundary of the swimmer.

For this reason, we will consider incompressible or rigid transports. Another example of bio-inspired swimming is presented in the highlights section.

5. Highlights of the Year

5.1. Highlights of the Year

Memphis team of Inria and VALOREM are both involved in the european project AeroGust (Aeroelastic gust modelling). One of the task aims to investigate the behaviour of wind turbine blades submitted to gust using

incompressible flow model and Octree grids. An other task is to carry on an experimental work on a wind turbine. Interests will be first to have real data and use it to better understand the effects of wind and more precisely of gusts, on wind blades. A second interest is to use experimental data to calibrate our numerical schemes in the high-fidelity CFD code.
The measurement of the wind was considered as the most important data to be obtained from the very start of the project. Indeed, this data will be used as a key input for the numerical simulations. This is needed to represent the wind as it arrives at the wind turbine. Then, wind turbine data collection aims to observe the aero-elastic behaviour of wind blades. So, the measurement of blade deformations will allow to check the structural beam model of the blade and to observe its structural behaviour. To observe the aerodynamic load on the wind blade, the measurement of pressure of air on the blade is of significant interest.
A meteorological mast has so been installed in March 2017 in Brittany (France) to measure wind on-site. In figure 5 can be seen a photograph of the whole mast after its installation. Figure 6 contains a picture focused on the sensors of the met mast which are wind vanes for the direction and anemometers for the velocity.

Figure 5. Photo of the met mast after its installation

For the instrumentation of the wind blade, the setup consists of 4 optical fibres along the blade. Each fibre has 4 sensors (pressure or strain gauges) and also temperature sensors at different lengths in order to calibrate the other sensors with respect to temperature. 10 strain gauges and 6 pressure sensors have so been installed on a wind blade located near the meteorological mast (in a way that in the main wind direction, the met mast and the wind turbine are aligned). In figure 7, the 2 lines of sensors going along the pressure side and the leading edge of the wind blade can be seen.

Work is now in progress with the experimental data in order to identify different gust conditions in the field and to analyse the effects on the blade deformations. One of the outcomes will be then to compute simulations with our high-fidelity numerical tool developed with VALOREM. This comparison will allow us to calibrate the numerical schemes thanks to real test data.

6. New Software and Platforms

6.1. COCOFLOW

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

Figure 6. Photo of the sensors on the met mast

Figure 7. Photo of the pressure side of the wind blade after instrumentation

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.

- Contact: Florian Bernard
- URL: https://gforge.inria.fr/projects/cocoflow

6.2. KOPPA

Kinetic Octree Parallel PolyAtomic
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
- Contact: Florian Bernard
- 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
- 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. Fluid-structure interaction and a monolithic scheme

Fluid-structure interaction (FSI) problems are still today difficult to solve on the numerical point of view. Memphis team works on the development of a new numerical method for the simulation of these phenomenas. This method relies on a FSI coupling scheme called "monolithic", in which an eulerian hyperelastic model (Mooney-Rivlin) predicts the behaviour of an elastic structure, all of this in the context of an implicit inclusion of the geometry. A 2D axi-symmetric incompressible Navier-Stokes model is used to follow the behaviour of a newtonian fluid, interacting with this elastic body.

With this coupling method, the solid and fluid problems are solve as a unique numerical solver. This approach has already been studied in the Memphis team for compressible fluids. This process seems to be interesting while it competes on the accuracy point of view with the partitioned approches, commonly used in the literature. More over, an eulerian formalism releases us from the constraints related to the tracking of the fluid-structure interface, which remains the key difficulty for lagrangian methods. This implicit consideration is therefore coherent from the perspective of including complexe geometries. In responding to difficulties related to the monolithic scheme, we employ a kind of meshing, particularly adapted to AMR (Adaptative Mesh Refinement). Developed by the OPTIMAD society, the library PABLO offers the ability to build conceptually simple meshes, natively parallel, and convenient to use. The hierarchical cartesian meshes are also particularly adapted to complex geometries.

The fluid-structure interface is followed via a level-set function. This one is transported in time with a 2nd order semi-lagrangian scheme which is volume conservative, and it is frequently reinitialize with a redistanciation algorithm. A linear extrapolation algorithm (Aslam) is besides added as a complement to the elastic model in order to limit the "non physical" effects introduced by the monolithic coupling scheme. Finally, a contact model is employed to model the collision between an elastic solid and a rigid solid which can occur in particular in a cardiac pump based on oscillating membranes.

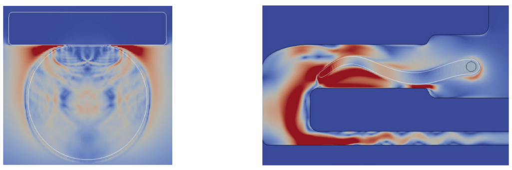

Figure 8. Two FSI problems. On the left: elastic cylinder colliding a rigid plate; on the right: hyperelastic membrane immersed in a pump geometry, moving thanks to a mechanical oscillating actuator.

7.2. A Local Lubrication Model for Spherical Particles within an Incompressible Navier-Stokes Flow

The lubrication effects are short-range hydrodynamic interactions essential to the suspension of the particles, and are usually underestimated by direct numerical simulations of particle laden flows.

A local lubrication correction model for particle laden flow of spherical solid particles has been presented and validated. Interactions between a particle and an obstacle (another particle or a wall) can be decomposed into three types: long range hydrodynamics, short range hydrodynamics also called lubrication effects, and mechanical solid-solid contacts.

Long range hydrodynamic interaction are fully resolved by the Volume Penalization method (VP). The incompressible Navier-Stokes equations have been discretized in time using a scalar projection method and in space with a fully second order penalty method.

Due to unresolved scales associated with the grid, short range hydrodynamic interactions are only partially captured by the numerical approach. We thus introduce a local lubrication model. This correction is based on asymptotic expansions of analytical solutions of particle-particle or particle-wall interactions, assuming that the flow within the gap between the particle and the obstacle is in the Stokes regime. Lubrication forces and torques are corrected in a neighborhood of the contact point of two interacting particles where lubrication is poorly captured, as long as the normalized gap width ϵ is smaller than a critical length ϵ_{lub} (a model parameter).

Finally, a linear soft-sphere collision model is used for solid-solid contacts. This model, widely used in the literature [Costa15,Izard14], represents mechanical contacts as two spring-dashpot systems connected at the contact point. The model allows stretching the collision time, to avoid computational overhead in the calculation of the collision force, making the method particularly efficient.

Our local lubrication correction model have been validated on several benchmarks. First, we considered a single particle falling onto a wall at various approach velocities. The comparison with experimental results [Harada01,Joseph01] enables us to validate the dominant lubrication component resulting from the squeezing of the fluid in the gap. The lubrication force and the torque created by the shearing of the fluid in the gap have been validated on oblique particle-wall collisions in dry and wet systems proposed by Joseph and Hunt [Joseph04] . Since lubrication corrections are made locally, our lubrication model does not required tabulation and is compatible to non-spherical particles. The model will be tested for polydisperse flow of ellipsoidal particles in future works.

7.3. Incompressible flow schemes on octrees.

The incompressible Navier-Stokes solver was validated in 2D last years thanks to the computation of the order of convergence. This year, a comparison has been done with data from literature. A first test-case was the flow around a 2-D cylinder. On the figure 9 can be seen a comparison between results from the developed solver and data from literature [Ploumhans (2000)].

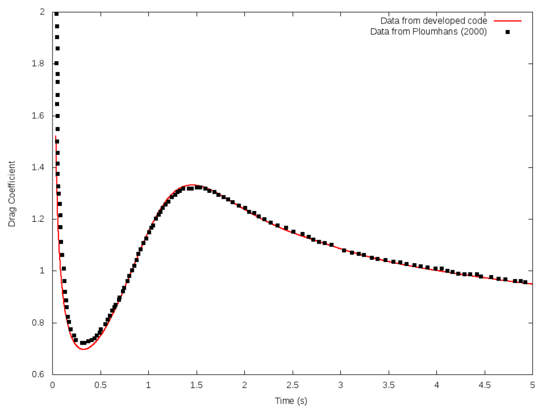

Figure 9. Comparison of drag coefficients between the code developped and data from literature for the flow around a cylinder at $Re = 550$

A second test case was the flow around a Naca0012 airfoil. The figure 10 shows the X-Velocity around the airfoil at Re = 1000 with an angle of attack of 15°. A QuadTree grid has been used as can be seen in figure 11. The aerodynamic coeficients have been computed for this test-case and have been compared with data from literature [D. F. Kurtulus (2015)]. With $C_D mean = 0.3$ and $C_L mean = 0.6$ the results are in good agreement with $C_D mean = 0.32$ and $C_L mean = 0.7$ from literature data gathered in [D. F. Kurtulus (2015)].

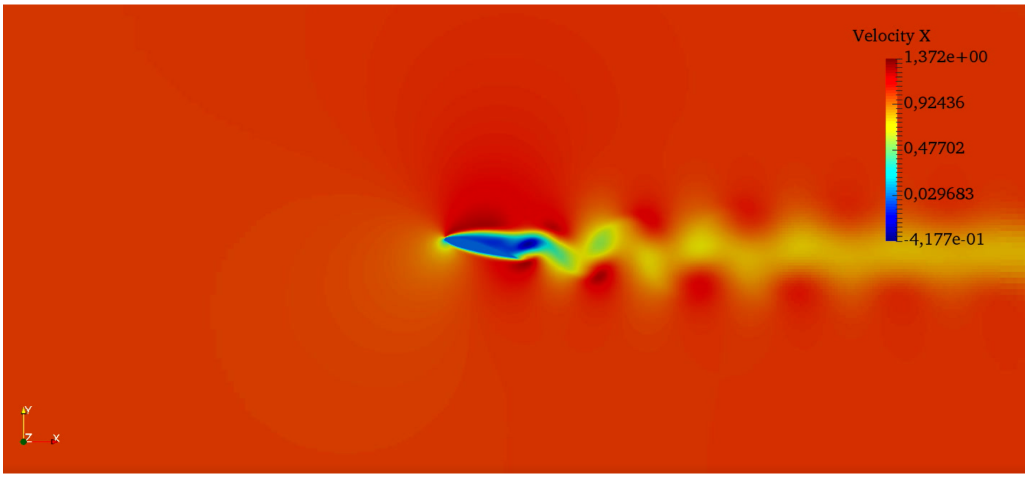

Figure 10. X-Velocity around a Naca0012 airfoil at $Re = 1000$ with an angle of attack of 15°

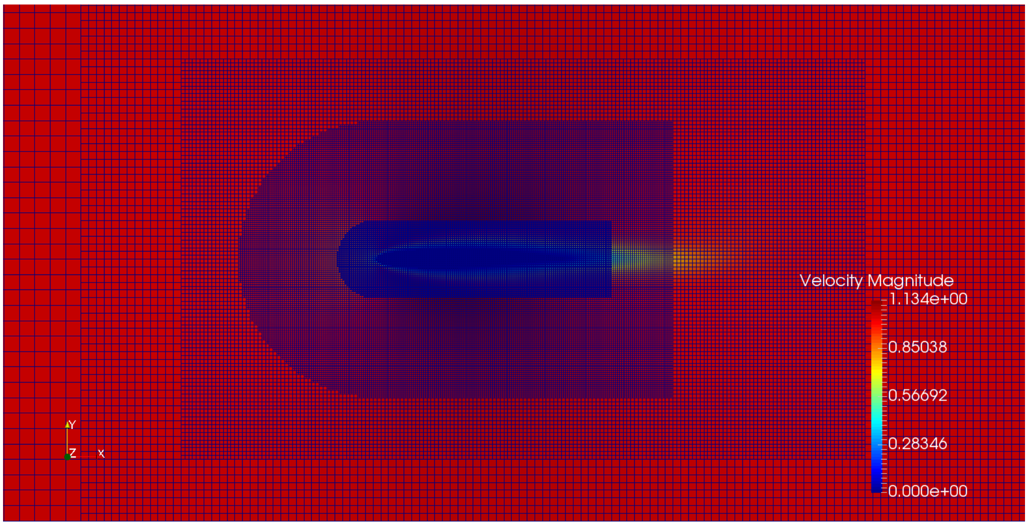

Figure 11. QuadTree grid around a Naca0012 airfoil

Then, a grid adaptation process has been developed. It allows for exemple to deal with moving bodies and to focus on interesting areas in the computational domain. With user defined criteria, the grid is indeed automatically refined or coarsened. So, this code allows a fast meshing of the computational domain thanks to the penalization approach. An interesting compromise between computational time and accuracy is also reached thanks to the adaptive mesh refinement process. A validation of the adaptive mesh refinement process has been done with a comparison between 2 cases : the case of the flow around a fixed body with an inflow of $1m.s^{-1}$ and the case of a moving body with a velocity of 1 $m.s^{-1}$ in a fixed flow. It can be seen on figure 12.

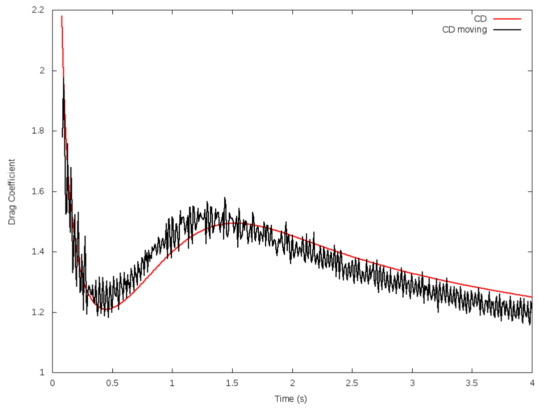

Figure 12. Comparison with drag coeficient obtained with a fixed body in a flow and with a moving body with a velocity of $1m.s^{-1}$ in a fixed flow

An extension of the code to 3-D has been developed and validated. Again, 2 different test-cases has been chosen for the validation. First, the flow around a sphere has been computed at different Reynolds number and a comparison has been done with several data from literature as shown in table 1. The figure 13 shows the X-Velocity of the flow around a sphere at $Re = 500$ with Octree grid. A LES turbulence model has been implemented with a Vreman subgrid model. So, the second test-case is the flow around a cylinder at $Re = 3900$ with LES. The wake profile at different positions has been compared with experimental data as can be seen in figure 14.

Table 1. Comparison of drag coefficients with data from literature at different Reynolds Number

Re	Present work	[Campregher (2009)]	[Fornberg (1988)]	[Fadlun et al. (2000)]	[Kim et al. (2001)]	[Subramanian (2003)]
300	0.6268	0.675	-	-	0.657	0.653
500	0.5488	0.52	0.4818	0.476	-	0.555

As the overall aim is to simulate the aeroelastic effects on a wind turbine subjected to gusts, a dynamic beam model with axial, torsional and flexural deformations have been implemented and coupled wih the Octree Navier-Stokes solver.

7.4. Validation of NaSCar at higher Reynolds numbers and Aeroelastic coupling

A beam finite element model has been implemented in order to study the dynamic behavior of the wind turbine blade. The structural model is linear and can describe bending, torsion and axial deformation. There is the possibility to take into account some coupling effects between bending-torsion and torsion-axial deformation. The implementation of the structural model has been validated by means of different static and dynamic tests.

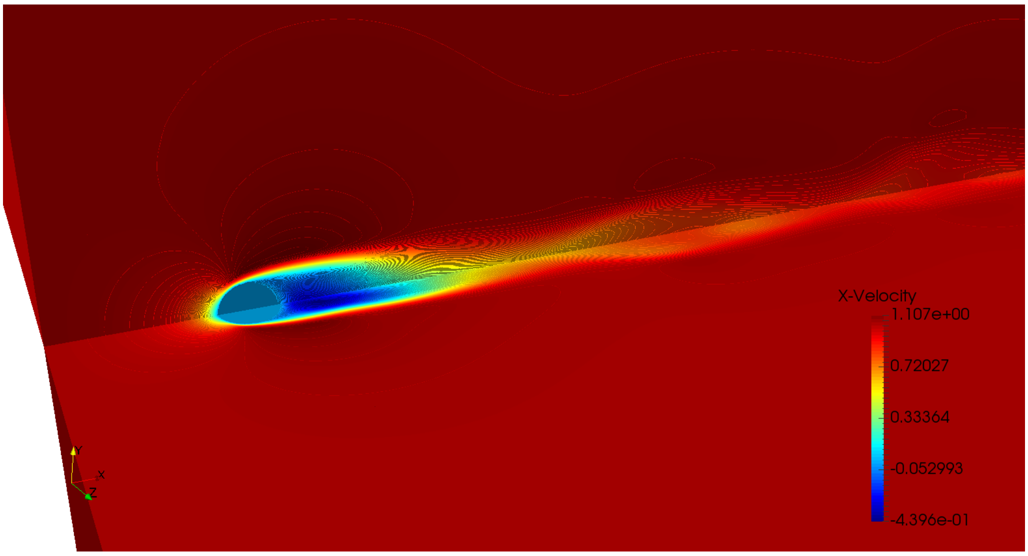

Figure 13. X-Velocity around a sphere at $Re = 500$

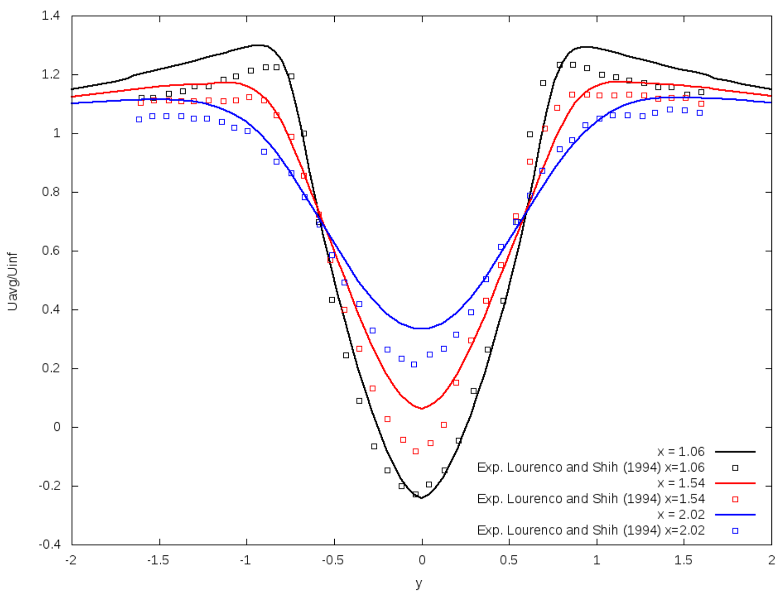

Figure 14. Wake profile at different positions behind a cylinder at $Re = 3900$ *obtained by averaging Velocity after a preliminary simulation*

In 15 the Fast Fourier Transform of the tip deflection history is reported: the frequency of the predicted peaks is in good agreement with the theoretical values.

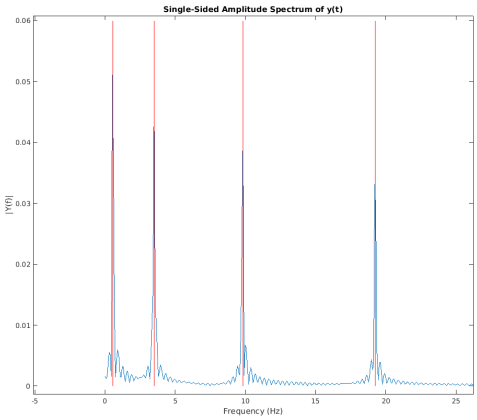

Figure 15. spectrum of the tip deflection

The structural model has been coupled with two different computational fluid dynamics codes: a cartesian code (NASCAR3d) and an octree code (developed by Claire Taymans during her PhD). The coupling requires to compute the loads for the structural model by performing an integral of the fluid forces on a surface mesh. The surface mesh is updated at each time step according to the displacement of the structure and this allows to update the level set which is used to impose the effects of the body on the fluid.
In order to focus the attention on a single blade of the rotor, the inertial terms (centrifugal and Coriolis forces) have been added in both the fluid solver and in the structural model. This makes it possible to perform a preliminary study of the behavior of a single elastic blade by neglecting the interactions between the different blades and the wind turbine's tower (see 16).

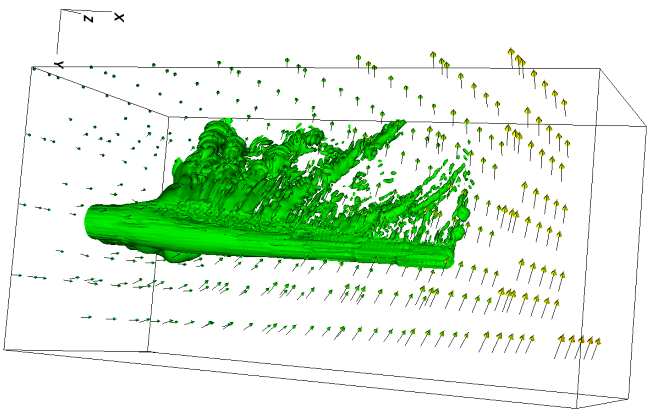

Figure 16. velocity vector field and q-criterion vorticity contour around the turbine blade

The turbulent flow around the blade is studied by means of the Vreman Large Eddy Simulation model [1] which has been tested on the flow around a cylinder at Re=3900 and Re=140000. The validation of the model for high reynolds flows required the use of a very fine mesh in order to appropriately simulate turbulent disspation and accurately predict the mean flow field, the results obtained are in good agreement with the experimental data of Cantwell et al [2], as reported in 17.

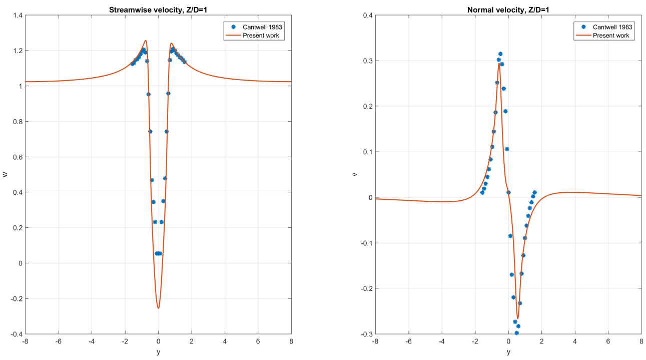

Figure 17. Cylinder mean wake velocity profiles, Re=140000

In order to extend the capability of the code to high Reynolds number a wall function approach has been implemented following the guidelines of De Tullio [3]. The main idea of this approach is to impose the value of the velocity in the first fluid cells close to the wall by performing a non-linear interpolation based on wall function which represents the velocity distribution in the turbulent boundary layer (see 18).

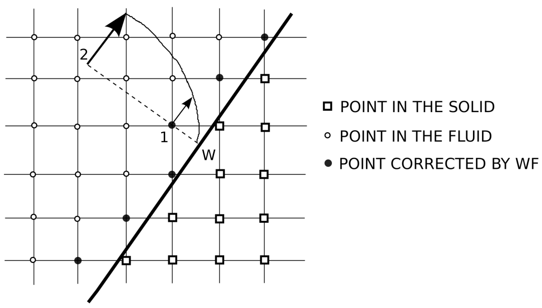

Figure 18. wall correction

[1] A.W. Vreman, "'An eddy-viscosity subgrid-scale model for turbulent shear flow: Algebraic theory and applications,"' Phys. Fluids 16,10 (2004)

[2] B. Cantwell, D. Coles, "'An experimental study of entrainment and transport in the turbulent near wake of a circular cylinder,"' J. Fluid Mech. (1983), vol. 136, pp. 321-374

[3] De Tullio, M.D. (2006) *Development of an Immersed Boundary method for the solution of the preconditioned Navier-Stokes equations* (PhD thesis)

7.5. Thoracic implant

We are interested in the simulation of elastic tissus deformation in order to simulate the skin deformation due to the pose of a thoracic implant. These implants are used to fill the sternum cavity of patients affected by Pectus Excavatum syndrom. As a first step, we simulated the skin deformation with a single layer elastic model from the real bones, skin and implant geometries imported from STL files. The implant geometry has been designed on-demand by Anatomik Modeling. The single layer elastic model representing an under-skin implant, has been implemented on an octree grid to easily and automatically refine around the different geometries and keep accuracy.

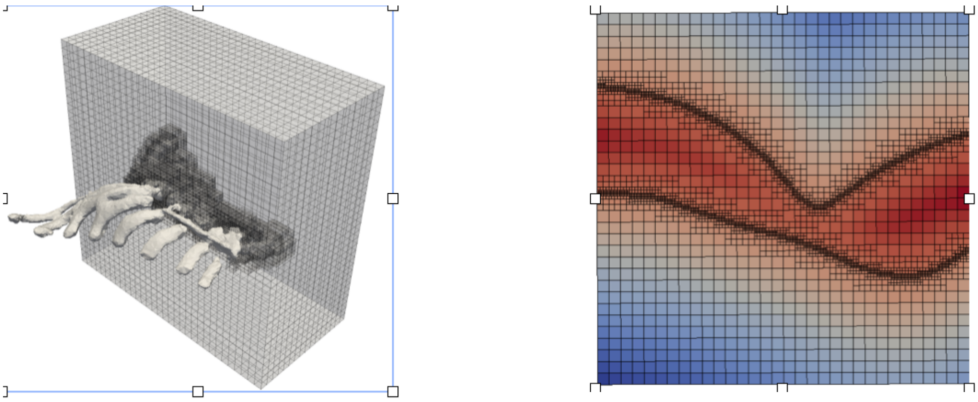

Figure 19. Left: Automatic refinement around a part of the rib cage. Right: Slice of the signed distance fonction from rib cage and skin with automatic refinement

The results obtained were qualitatively validated by Anatomik Modeling. The implant actually lays on the rib cage, under the muscles. The next step will be then to include a multi-layer elasticity model to take into account the muscles and other biological soft tissus.

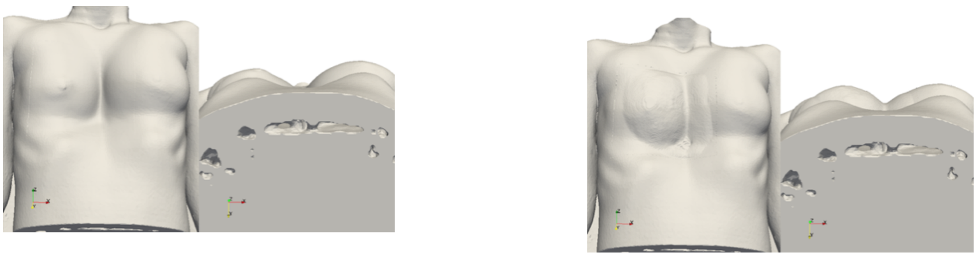

Figure 20. Left: Skin without implant. Right: Skin simulation with implant under skin

Another problem linked to custom made thoracic implants is the extraction of the so-called surgical plan. It is a critical step necessary to design the implant. This plan corresponds to the surface of the rib cage. To extract it, a mass-spring model has been developed and integrated in a software prototype with a graphical interface. The resulting prototype can be used easily from on any rib cage described by a STL file.

8. Bilateral Contracts and Grants with Industry

8.1. Bilateral Contracts with Industry

Ongoing contract with the society VALOREM.

9. Partnerships and Cooperations

9.1. Regional Initiatives

Leading team of the regional project "Investigation and Modeling of Suspensions with the LOMA and LOF labs in Bordeaux

9.2. National Initiatives

We belong to the GDR AMORE on ROMs.

9.2.1. Starting grants

NEMO (A Numerical Enabler for MultiPhysics Simulations on Octrees) is an action to improve and merge all the main MEMPHIS numerical codes. To achieve this goal we have a 12 months financial support (Inria BSO FRM) for a young engineer. This work will be done with strong interaction the the local Inria BSO SED as well as Philippe Depouilly from the IMB "SED".
SMecH is a start-up project in software edition, carried on by Florian Bernard, research engineer in the MEMPHIS team. The project aims at porting to an industrial level the numerical codes developed by the MEMPHIS team. The different collaboration with industrial partners have highlighted the need of new numerical tools to simulate high complexity phenomena such as atmospheric reentries, multi-material flows or fund-structure interactions, but also to highly automatize the numerical simulation workflow to save engineer time. The research codes developed in the MEMPHIS team could match perfectly to this need thanks to:

- the various innovative multi-physics models implemented
- the use of Hierarchical Cartesian schemes that automatize the treatment of moving geometry with accuracy
- the development of schemes suitable for High Parallel Computing.

This year, the project has been submitted to the DGDT, the Inria department in charge of technological transfert, and has been granted an engineer for 6 months as well as the support of IT-Translation.

9.3. European Initiatives

9.3.1. FP7 & H2020 Projects

EU research projects were and will be a privileged instrument of diffusion and transfer of our results. The AEROGUST H2020 project involves aeronautical industry (Airbus, Dassault, Piaggio..) and research labs (University of Bristol, DLR, NLR, University of Cape Town) and is dedicated to modeling of aerodynamic gust response for applications. We take part in this project by developing simulation models for unsteady aeroelastic problems and data-driven reduced-order models. We played a similar role for the past in the FP7 project FFAST with the same partners.

9.3.1.1. AEROGUST

Title: Aeroelastic Gust Modelling

Programm: H2020

Duration: May 2015 - April 2018

Coordinator: University of Bristol

Partners:

Airbus Defence and Space (Germany)

University of Cape Town (South Africa)

Dassault Aviation (France)

Deutsches Zentrum für Luft - und Raumfahrt Ev (Germany)

Stichting Nationaal Lucht- en Ruimtevaartlaboratorium (Netherlands)

Numerical Mechanics Applications International (Belgium)

Optimad Engineering S.R.L. (Italy)

Piaggio Aero Industries Spa (Italy)

The University of Liverpool (United Kingdom)

University of Bristol (United Kingdom)

Valeol (France)

Inria contact: Angelo IOLLO and Michel Bergmann

Encounters with atmospheric turbulence are a vitally important in the design and certification of many manmade structures such as aircraft and wind turbines. Gusts cause rapid changes in the flow about the structures which leads to rigid and flexible unsteady responses. Knowledge of aircraft/gust interactions is therefore vital for loads estimation during aircraft design as it impacts on control systems and often defines the maximum loads that these structures will experience in service. At present industry typically uses the linear doublet lattice method with static loads corrections from expensive wind tunnel data. The wind tunnel data is created using the final aerodynamic surface in the predicted cruise shape. This means that gust loads come relatively late when the design options have been narrowed. Increased competition and environmental concerns are likely to lead to the adoption of more flexible materials and the consideration of novel configurations, in which case the linear assumptions of the current gust loads process will become unacceptable. To introduce non-linearity into the gust loads process without significantly increasing the cost and time, this project has three main objectives: to carry out investigations using CFD so that the non-linearities in gust interactions are understood; to create a gust loads process that does not require wind tunnel data and hence reduces the need for wind tunnel testing; to develop updated reduced order models for gust prediction that account for non-linearity at an acceptable cost. These investigations will reduce the need for expensive wind tunnel testing and hence lead to time and cost savings at the design stage therefore ensuring that the European aerospace and defense industry remain competitive in the future. The wind turbine industry has similar concerns, with gusts and wind shear restricting the locations available for wind farms. The project will also address these issues using common methodology.

9.3.2. Collaborations with Major European Organizations

Partner 1: Chalmers University (Sweden)

This activity is complemented by several international interactions, in particular with Chalmers University in order to converge towards the real implementation of new control technologies on cars, buses and trucks.

Partner 2: Optimad Engineering , Torino (Italy)

We have a crucial partnership with Optimad Engineering, a spin-off of the Politecnico di Torino. This society has implemented in industrial codes several schemes that we have developed for the past. In exchange, we have access to these codes. One example is Pablo, an octree managing parallel library (http://www.optimad.it/products/pablo/). Three former PhD students at Inria are presently employed in Optimad and several others have spent or will spend a research period in this company in order to get acquainted with code architecture and massive parallelism. This company represents for us an ideal partner for the actual industrial feedback on our methods. As mentioned, we plan to create a local start-up in close collaboration with Optimad. This start-up will respond to actual industrial needs by specific software packages built starting from open source tools that are made available to the applied research community via a consortium. Florian Bernard has been recruited in Memphis for two years with the objective of bringing to a higher maturity level a set of modules developed within the team. He plans to fully invest himself in the creation of the start-up. As for the consortium, we are discussing with several partners including Cineca (Italy HPC center) and Optimad about how to structure such a mutual effort. The Storm Inria team is included in the discussions as a possible partner.

Partner 3: W4E (Wave for Energy) (Italy)

One project is the design of an ISWEC (Inertial See Wave Energy Converter) in collaboration with W4E (Wave for Energy), Optimad and others. The ISWEC is a floater prototype that can extract energy form the sea waves. The mechanism is based on a gyroscope that is rotating due to the passive motion of the floater. This prototype is actually tested in the Mediterranean sea in Italy. We will develop the numerical simulation as well as the shape optimization of the ISWEC.

Partner 4: MRGM (Maladies Rares : Génétique et Métabolisme), Bordeaux University (France)

We develop a collaboration with the MRGM lab. They are interested in the swimming of a zebrafish larvae under genetic modifications. One aim is to quantify the power spent by such fishes to swim after a stimuli reaction. The numerical simulation we develop can help computing integral quantities such as the power. This simulation is challenging due to the coupling several methods like image treatment (from movies given by MRGM), optimal transport and numerical simulations.

Partner 5: CRPP (Centre de recherche Paul Pascal), LOF (Laboratoire du Futur) and LOMA (Laboratoire Ondes et Matière d'Aquitaine) labs, Bordeaux University, France.

We established collaborations with physics and chemistry labs in Bordeaux, namely the CRPP, the LOF and the LOMA. They are concerned with the behavior of many passive (CRPP and LOF) and active (LOMA) particles in an incompressible flow. With these partners, we intend to use a combined experimental and computational approach to calibrate models in the case of dilute and concentrated suspensions. The numerical simulations of such particles can help to understand some underlying phenomena at the particles scale and thus to develop mesoscopic models for the whole system (PhD of Baptiste Lambert, oct. 2015).

9.4. International Research Visitors

9.4.1. Visits to International Teams

We have obtained a grant from the Idex Bordeaux of $10keuro$ to start a collaboration with Charbel Farhat of Stanford University on ROMs.

10. Dissemination

10.1. Promoting Scientific Activities

10.1.1. Scientific Events Selection

10.1.1.1. Member of the Conference Program Committees

Michel Bergman has co-organized the international conference "Interaction fluide-Structure: Analyse et controle", October 2017 (https://indico.math.cnrs.fr/event/1366/overview)

10.1.2. Journal

10.1.2.1. Member of the Editorial Boards

Angelo Iollo is in the advisory board of Acta Mechanica.

10.1.2.2. 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.

10.1.3. Invited Talks

The invited conferences are [10], [8], [13], [14], [9], [11], [15], [7].

10.1.4. Scientific Expertise

Angelo Iollo is reviewer for national and international programs such as H2020 (EU), ANR (France), PRIN (Italy).
2016-2017: Angelo Iollo is expert for the Italian Ministry of Research: quality evaluation of research products.
Michel Bergmann: member of the Inria Young Researchers Commission, which allocates PhD and Postdoc grants.
Afaf Bouharguane has participated to the recruitment committee for Associate Professor position in Besancon, May 2017
Angelo Iollo was expert in the Young Investigator Rita Levi Montalcini program, Italy

10.2. Teaching - Supervision - Juries

10.2.1. Teaching

Four members of the team are Professors or Assistant Professors at Bordeaux University and have a teaching duty, which consists in courses and practical exercises in numerical analysis and scientific computing. Michel Bergmann (CR) also teaches around 64 hours per year (practical exercises in programmation for scientific computing).

10.2.2. Supervision

PhD in progress : Claire Morel, Modélisation aerodynamique 3D d une turbine eolienne, 01/01/2015, M., Bergmann M., Iollo A.

PhD in progress : Federico Tesser, Identification of dense suspensions rheology, 01/11/2014, Bergmann M., Iollo A.

PhD in progress : Baptiste Lambert, modélisation et simulations numériques des contacts dans des écoulements chargés en particules, 01/10/2015, Bergmann M., Weynans L.,

PhD in progress : Emanuela Abbate, Méthodes numériques pour problèmes stiff en mécanique des fluides et élasticité, 01/11/2015, Iollo, A.

PhD in progress : Mathias Braun, Modàles réduits et problèmes inverses pour l étude de la résilience des réseaux d eau potable, 01/10/2015, Iollo A. and Mortazavi I.

PhD in progress : Luis Henrique Benetti Ramo,Aeroelastic instabilities, Bergmann M. and Iollo A.

PhD in progress : Guillaume Ravel, Simulation numérique et modélisation de la nage du poisson zèbre pour l étude de maladies humaines d origine génétique et toxicologique, 01/10/2017, Bouharguane A. and Babin P. (MRGM)

PhD in progress : Sebastien Riffaud,Reduced Order Models, classification and data geometry, 01/10/2017, Iollo A.

2013-2017: Meriem Jedoua, Introduction d une méhode efficace de capture d intreface permettant la localisation d un grand nombre d objets immergés dans un fluide. Applications à des solides rigides et des vésicules (membranes élastiques) immergés dans un fluide incompressible, 01/10/2013, Bruneau C.-H. and Maitre E.

2014-2017 : Alice Raeli, Numerical Modelling for Phase Changing Materials, 12/06/2014, Azaiez M., Bergmann M., Iollo A.

10.2.3. Juries

Michel Bergmann has participated to the PhD defense of Pierre Costini, Centrale Marseille, 19/05/2017
Michel Bergmann has participated to the PhD defense of Lei Cheng, DELF (pays-bas), 15/12/2017
Angelo Iollo has been reviewer of the PhD defense *Applicabilité de la réduction de modèles à la conception aérothermique collaborative des systèmes d air secondaires des turbomachines*, Pierre Costini, Ecole Doctorale des Sciences pour l Ingénieur, Aix-Marseille, May 2017.
Angelo Iollo has participated to the PhD defense of Manon Deville, *Modélisation de l'électroporation et de la transfection de gènes à l'échelle du tissu. Aspects théorique et numérique*. " Institut de Mathématiques de Bordeaux, université de Bordeaux, novembre 2017.
Angelo Iollo has participated as president to the PhD defense of Agathe Peretti *Quantification de l hétérogénéité tumorale à partir de l imagerie médicale. Application à la classification de tumeurs rénales.* Institut de Mathématiques de Bordeaux, université de Bordeaux, décembre 2017.

10.3. Popularization

Lisl Weynans has co-organized the *Journée Filles et Maths, une équation lumineuse*, April 2017.
Afaf Bouharguane and Lisl Weynans have co-organized the *Journée Emploi Maths de l' Unité de Formation Mathématiques et Interaction*, November 2017.

11. Bibliography

Publications of the year

Doctoral Dissertations and Habilitation Theses

[1] A. RAELI. *Solution Of The Variable Coefficient Poisson Equation On Cartesian Hierarchical Meshes In Parallel: Applications To Phase Changing Materials*, IMB - Institut de Mathématiques de Bordeaux, October 2017, https://hal.inria.fr/tel-01666340

Articles in International Peer-Reviewed Journals

[2] 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

[3] 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", January 2018, vol. 352, pp. 301 - 325 [*DOI :* 10.1016/J.JCP.2017.10.001], https://hal.inria.fr/hal-01668546

[4] F. BERNARD, A. IOLLO, G. PUPPO. *Simulation of particle dynamics for rarefied flows: Backflow in thruster plumes*, in "European Journal of Mechanics - B/Fluids", 2017, vol. 63, 14 p. , https://hal.inria.fr/hal-01467275

[5] A. DE BRAUER, A. IOLLO, T. MILCENT. *A Cartesian Scheme for Compressible Multimaterial Hyperelastic Models with Plasticity*, in "Communications in Computational Physics", September 2017, vol. 22, n$^{\text{o}}$ 5, pp. 1362-1384 [*DOI :* 10.4208/CICP.OA-2017-0018], https://hal.inria.fr/hal-01586837

[6] A. RAELI, M. BERGMANN, A. IOLLO. *A Finite-Difference Method for the Variable Coefficient Poisson Equation on Hierarchical Cartesian Meshes*, in "Journal of Computational Physics", November 2017, vol. 355, pp. 59-77 [*DOI :* 10.1016/J.JCP.2017.11.007], https://hal.inria.fr/hal-01662050

Invited Conferences

[7] M. BERGMANN. *A hybrid DNS/ROM approach for wind and ocean wave energy converters*, in "MECASIF Workshop 2017 - MECASIF on reduced order methods for wind and marine current power", Sophia Antipolis, France, February 2017, https://hal.inria.fr/hal-01668467

[8] M. BERGMANN, A. FERRERO, A. IOLLO. *A Zonal POD Approach for Gust Computations*, in "Conference on Classical and Geophysical Fluid Dynamics: Modeling, Reduction and Simulation", Blacksburg, United States, June 2017, https://hal.inria.fr/hal-01668420

[9] A. IOLLO, M. BERGMANN, A. FERRERO, H. TELIB. *Model reduction and advection modes*, in "Conference on Reduced Order Modeling for classical and geophycis fluid flows", Blacksburg , United States, June 2017, https://hal.inria.fr/hal-01668543

[10] A. IOLLO. *Compressible multimaterial numerical models*, in "IperPV 2017 - XVII Italian meeting on hyperbolic equations", Pavia, Italy, September 2017, https://hal.inria.fr/hal-01668533

International Conferences with Proceedings

[11] E. ABBATE, A. IOLLO, G. PUPPO. *A relaxation scheme for the simulation of low Mach number flows*, in "FVCA 2017 - International Conference on Finite Volumes for Complex Applications 8", Lille, France, FVCA 2017: Finite Volumes for Complex Applications VIII - Hyperbolic, Elliptic and Parabolic Problems, June 2017, vol. 200, pp. 227-235 [*DOI :* 10.1007/978-3-319-57394-6_25], https://hal.inria.fr/hal-01666248

Conferences without Proceedings

[12] E. ABBATE, A. IOLLO, G. PUPPO. *An all speed relaxation scheme for compressible materials*, in "NUMHYP 2017 - Numerical methods for hyperbolic problems", Monte Verità, Switzerland, May 2017, https://hal.inria.fr/hal-01666500

[13] M. BERGMANN, A. FERRERO, A. IOLLO. *POD-assisted computation of incompressible gust flows*, in "MORTech 2017 : Reduced Basis, POD and PGD Model Reduction Techniques", Séville, Spain, November 2017, https://hal.inria.fr/hal-01668445

[14] A. IOLLO, M. BERGMANN, A. FERRERO. *A multiscale reduced-order model for NS equations from residual minimization* , in "SIAM 2017 - Conference on Computational Science and Engineering", Atlanta, United States, February 2017, https://hal.inria.fr/hal-01668516

[15] A. IOLLO, M. BERGMANN, A. FERRERO, H. TELIB. *Advection modes and model reduction* , in "MORTech 2017 - 4th International Workshop on Reduced Basis, POD and PGD Model Reduction Techniques", Seville, Spain, November 2017, https://hal.inria.fr/hal-01668550

Research Reports

[16] M. Bergmann, L. Weynans. *A sharp Cartesian method for incompressible flows with large density ratios*, Inria Bordeaux, March 2017, n^{o} RR-8926, 23 p. , https://hal.inria.fr/hal-01331234

[17] B. Lambert, L. Weynans, M. Bergmann. *Local Lubrication Model for Spherical Particles within an Incompressible Navier-Stokes Flow*, Inria Bordeaux, équipe MEMPHIS ; Université Bordeaux, September 2017, n^{o} RR-9093, pp. 1-33, https://hal.inria.fr/hal-01585066

[18] L. Weynans. *Convergence of a cartesian method for elliptic problems with immersed interfaces*, Inria Bordeaux ; Univ. Bordeaux, September 2017, n^{o} RR-8872, 24 p. , https://hal.inria.fr/hal-01280283

[19] L. Weynans. *Super-convergence in maximum norm of the gradient for the Shortley-Weller method*, Inria Bordeaux ; Inria, August 2017, n^{o} RR-8757, 16 p. , https://hal.inria.fr/hal-01176994

Other Publications

[20] A. Bouharguane, B. Melinand. *A Splitting Method for Deep Water with Bathymetry*, February 2017, https://arxiv.org/abs/1606.01630 - working paper or preprint, https://hal.archives-ouvertes.fr/hal-01326794

[21] A. Bouharguane, N. Seloula. *Finite Element Method for a Space-fractional Anti-diffusive Equation*, September 2017, working paper or preprint, https://hal.archives-ouvertes.fr/hal-01592569

References in notes

[22] P. Angot, C. Bruneau, P. Fabrie. *A penalization method to take into account obstacles in a incompressible flow*, in "Num. Math.", 1999, vol. 81, n^{o} 4, pp. 497-520

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

[24] 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

[25] M. Bergmann, C. Bruneau, A. Iollo. *Enablers for robust POD models*, in "J. Comput. Phys.", 2009, vol. 228, n^{o} 2, pp. 516–538

[26] 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

[27] 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

[28] 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

[29] 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

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

[31] 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]

[32] J. L. LUMLEY. *Atmospheric Turbulence and Wave Propagation. The structure of inhomogeneous turbulence*, A.M. Yaglom & V.I. Tatarski, 1967, pp. 166-178

[33] 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]

[34] 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

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

[36] 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]

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

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

[39] 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

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

[41] 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 [*DOI :* 10.1016/J.JCP.2016.02.032], http://www.sciencedirect.com/science/article/pii/S0021999116000966