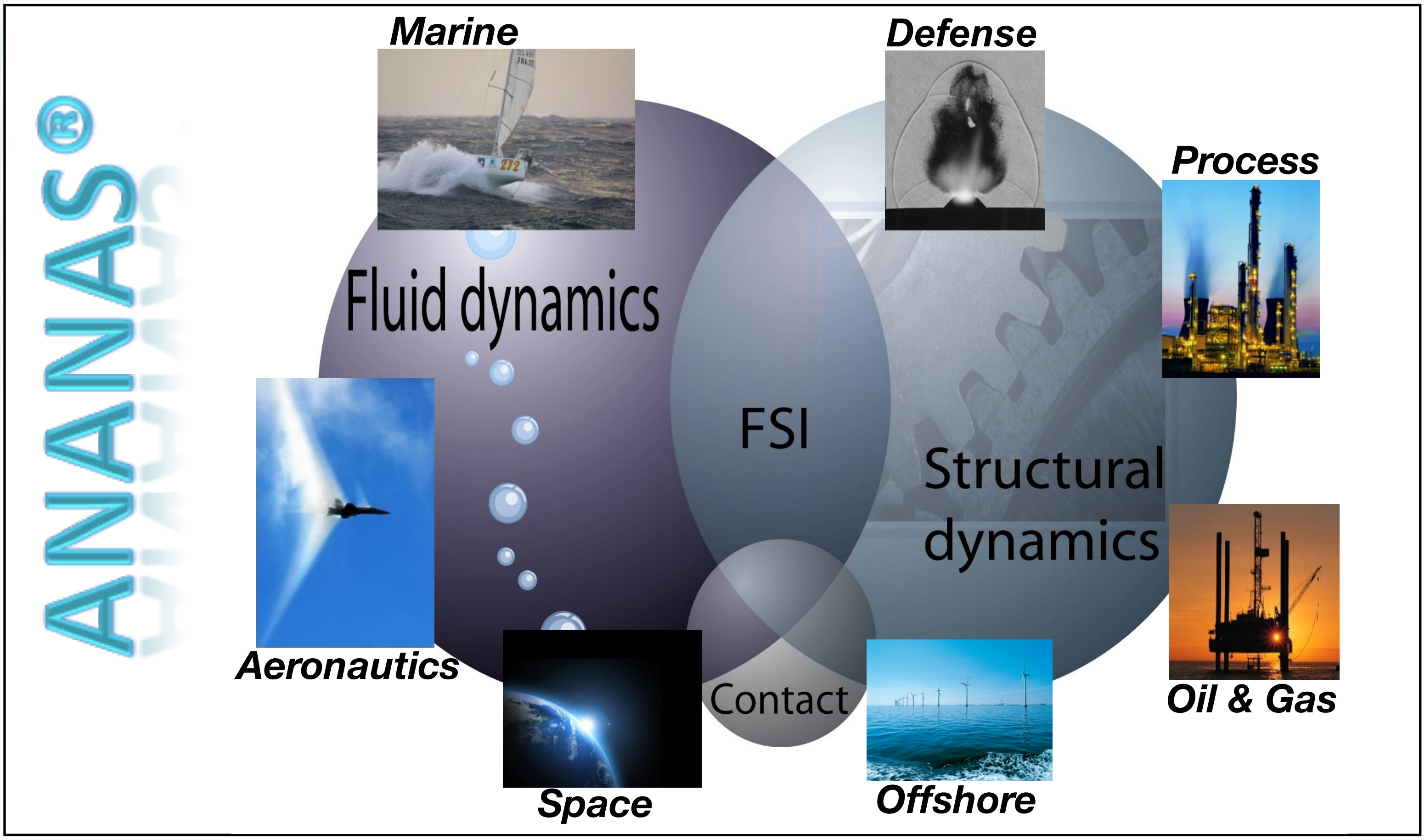
ANANAS®

An innovative tool for your CFD, Structure and Fluid Structure Interaction challenges

Mindset

At LEMMA, we believe that the complexity of reality deserves the most advanced numerical resolution. Thanks to the contributions of experts in applied mathematics, fluid and structural mechanics, we were able to build our software around cutting-edge methods optimised for parallel computing. Now, we can deliver accurate solutions to challenges in a wide variety of fields, thanks to our integrated multi-physics software **ANANAS®**.



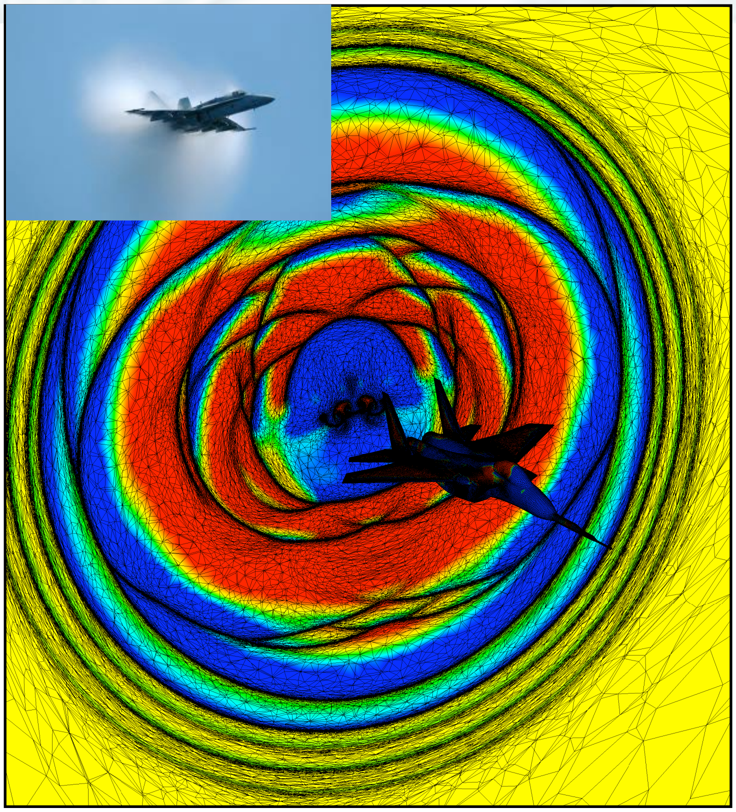
LEMMA Sophia-Antipolis 2000 route des Lucioles, Les Algorithmes-Thales A, 06480, Biot, FR phone: (+33) 493905390

LEMMA Paris Domaine de Voluceau BP 105, 78153, Le Chesnay, FR phone: (+33) 139635387

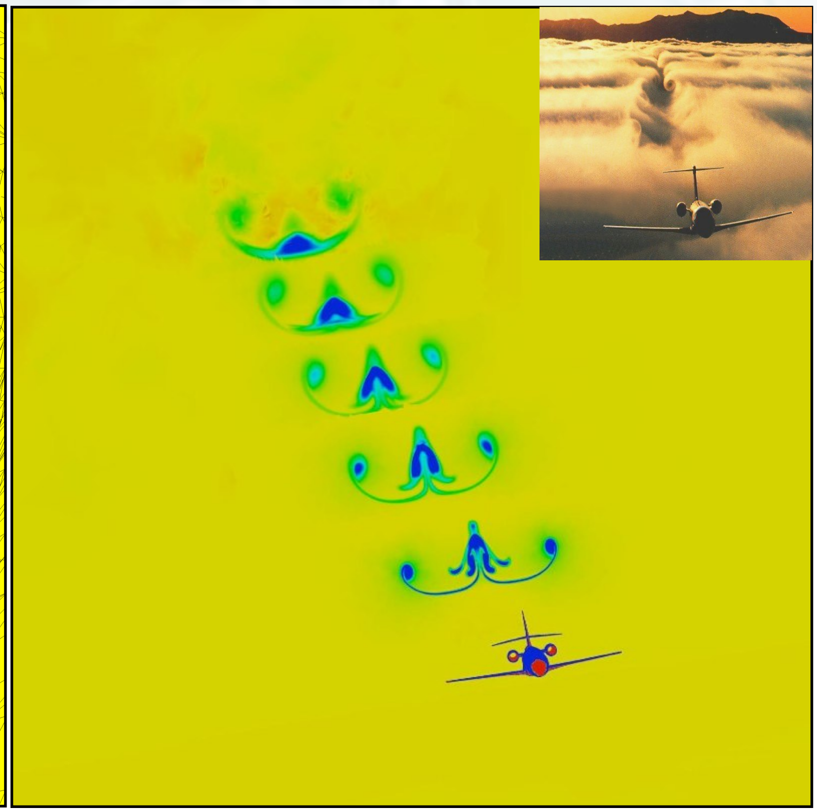
LEMMA USA 1235 North Loop West suite 716, 77008, Houston, TX, USA phone: (+1) 7133339940

www.lemma-ing.com

«Seize the dimensions of reality»



Tri-dimensionnal structure of the shock and pressure field around a supersonic fighter jet



Decay of the wingtip vortices hundreds of meters downstream a business aircraft

What are the common points between the tri-dimensionnal multi-phase flow in a kilometric pipe, the sea-keeping of a full scale boat during three hours, the shock pattern kilometers around a supersonic fighter jet and the dynamics of wingtip vortices several kilometers behind a business aircraft?

- All these full-scale problems involve a wide range of dimensions that cannot be properly captured using traditional numerical approaches without a prohibitive CPU cost.
- Yet, all these problems have been solved with **ANANAS®**.

At **LEMMA**, we believe that the best way to get a trustworthy numerical solution to a challenging industrial problem is to properly handle three key aspects of the simulation process - namely the mesh, the physics, and the numerics.

First, industrial designs often exhibit stringent geometrical features (complex blueprints, moving parts) that are difficult to properly reproduce without an adequate meshing technology.
Second, industrial flows involve a combination of physical phenomena related to a wide spectrum of domains (multi-phase flow, turbulence, structural mechanics, heat transfer...) which need to be properly modeled.
Finally, the numerical methods need to be robust enough to dampen the numerical instabilities and, at the same time, be precise enough to capture the physical ones .

Mesh
- industrial designs
- local phenomena

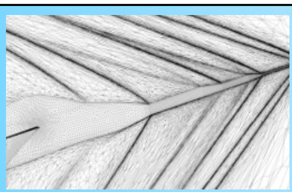
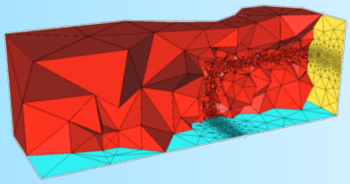
Physics
- multi-physics
- multi-scale

Numerics
- precision
- robustness

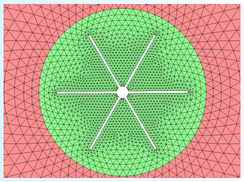
By combining cutting-edge technologies to cope with these three aspects in its fully integrated **ANANAS®** solver, **LEMMA** will provide the best solution possible to your numerical challenges.

Technology overview

Mesh

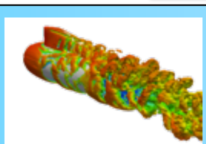



2D/3D Fully unstructured and intelligent mesh

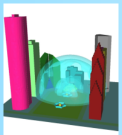


Moving mesh

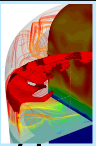
Physical modeling



Turbulence



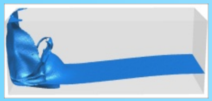
(In)compressible flows



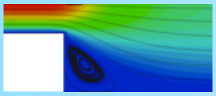
Heat transfer

Small and large structure deformation

Fluid - Structure Interaction



Free surface flows



Non newtonian flows

Bubbles / Droplets

Emulsion / sand flow

Erosion

Oil / polymer rheology

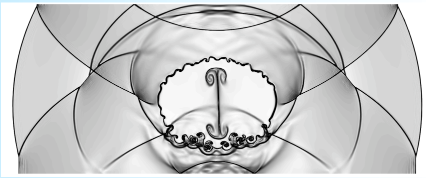
Contact / damaging

Numerics

High order conservative schemes at high anisotropy

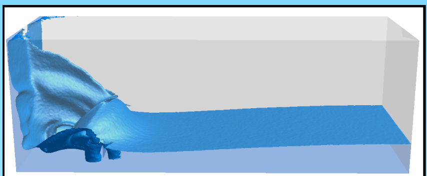
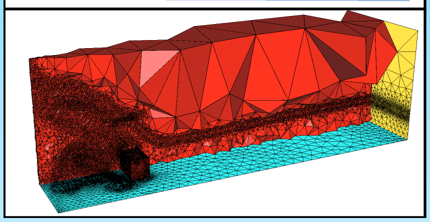
Up to 6th order in space and 3rd order in time

Mixed Finite Element and Volume methods

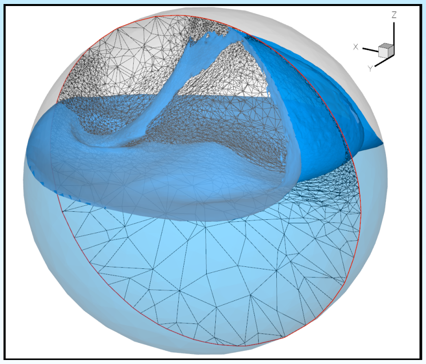


Focus on cutting-edge technologies

Industrial problems

Breaking dam splashing (i.mesh)



Sloshing in a cryogenic tank

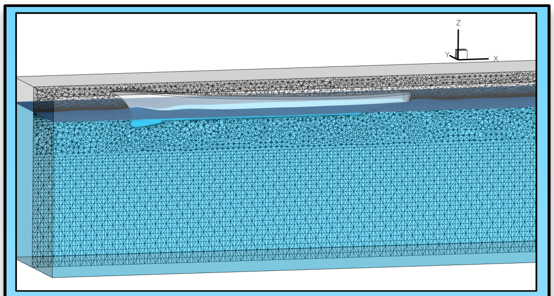
Intelligent mesh

The **intelligent mesh** concept is a powerful tool to reduce the cost of numerical simulations while preserving a desired level of accuracy. The main idea is that, at a fixed number of vertices, all meshes are not equivalent in terms of numerical efficiency. Indeed, some meshes will lead to smaller numerical errors than others.

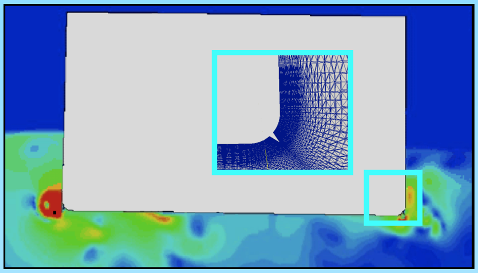
In **ANANAS®**, contrarily to most commercial softwares, our mesh adaptation process is not based on a simple refinement/coarsening strategy depending on a purely local error estimation. Our strategy controls local as well as global numerical errors. Moreover, it does not only adapt the mesh in terms of number of vertices but also in terms of elements' shape and orientation, generally depending on the physical phenomena at stake. Locally, elements' size and orientation are adjusted to equally distribute numerical errors in all directions. Globally, an optimal distribution of grid points in the computational domain is guaranteed.

The intelligent mesh method has been successfully applied to a wide range of flows such as:

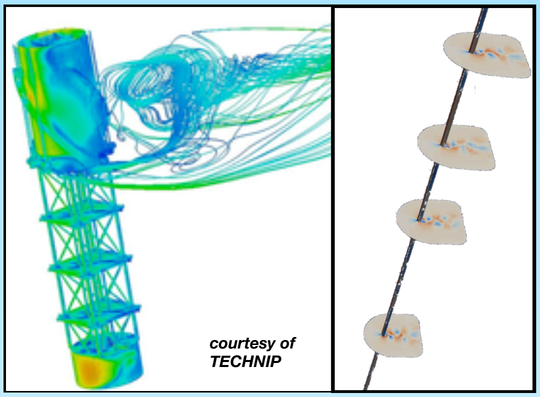
- the sloshing in ship and aerospace tanks under gravity or in micro-gravity
- the three-phase separation in a FPSO tank
- sea-keeping problems combined with the numerical basin
- splashing of a breaking dam (*Guegan et al. Int. J. Num. Meth. Eng. 2010*)
- compressible two-phase flows
- propagation of acoustic waves



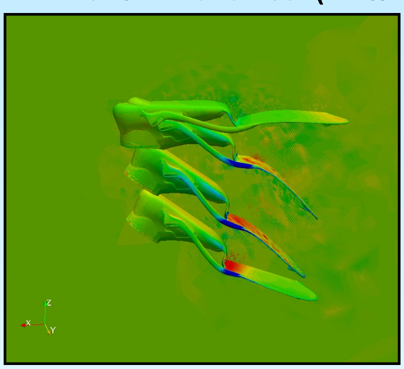
Numerical basin technology



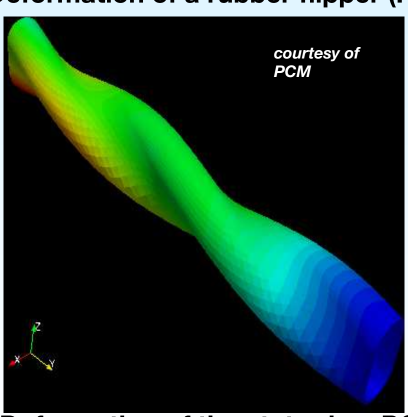
Sea-keeping of a FPSO (VMS/FS)



VIV-VIM of SPAR and riser (VMS/FSI)



Deformation of a rubber flipper (FSI)



Deformation of the stator in a PCP

Free-surface flows and numerical basin

Free-surface flows are encountered in many offshore and hydrodynamic applications and their precise handling can lead to a high CPU cost as the conditions simulated approach full scale.

In **ANANAS®**, free surface flows are modeled with a level-set approach which can reach a higher order of numerical precision than regular Volume of Fluid methods, thus conveniently reducing the number of grid nodes.

A dedicated numerical basin that includes all these technologies, along with a wave and current toolbox to simulate regular/irregular waves and potential flows, allows the user to reproduce real wave conditions on detailed geometries with high fidelity.

This approach has proven its efficiency for several problems such as:
- the sea-keeping of a FPSO with bilge keels
- the maneuvrability of a DTMB ship
- the Vortex Induced Motion of a flat buoy (*Vivet et al. OMAE 2011)*

Turbulence

Most industrial flows are turbulent because they involve large scales and high velocity. Hence, the modeling of turbulence is of prime importance for a CFD software. A predictive numerical simulation of turbulent flows gets more difficult to obtain as the Reynolds number increases, requiring more precise approaches in terms of modeling and numerics.

In **ANANAS®**, we have put a strong emphasis on high fidelity turbulence modeling. Our flagship model Large Eddy Simulation - Variatonal Multi-Scale (LES-VMS) is a very powerful tool to capture the dynamics of the turbulent structures as small as the grid size. As a consequence, most of our simulations of turbulent flows are run using this approach. The use of such high fidelity models are only made possible thanks to high order numerical schemes for unstructured grids (up to order 6 in space and 3 in time) which give access to precise unsteady computations.

However, we are aware that such computations are very CPU demanding; therefore, we also offer a Reynolds Averaged Navier Stokes (RANS) approach and hybrid RANS-VMS modeling which can conveniently replace LES-VMS for high Reynolds number attached flows.

Our turbulent models have successfully been able to simulate industrial problems such as:
- the turbulent flow in smooth and corrugated pipes (LES-VMS)
- turbulent far wakes behind dragged or propelled bodies (LES-VMS)
- the VIM of a detailed SPAR (LES-VMS) (*Sirnivas et al. OMAE 2006*)
- slender buoy VIM + riser VIV (RANS-VMS) (*Minguez et al. OMAE 2011)*

Fluid Structure Interaction

In several industrial applications, the flow has a strong interaction on the structure of the device which needs to be accurately predicted.

In **ANANAS®**, we offer a combination of cutting-edge meshing technics and a strong coupling between the fluid and the structure which makes possible the simulation of FSI problems. Moreover, the code is scalable for both fluids and structure enabling highly parallel computation.

FSI strongly coupled simulations have been satisfyingly achieved for:
- the rotor-stator interaction in a PCP pump (*Berton et al. SPE 2011*)
- the prediction of the dynamics of a rubber flipper
- the prediction of ice loads (combined with damaging)