RESEARCH CENTRE

Inria Saclay Center

2022

ACTIVITY REPORT

Project-Team

GAMMA

Adaptive Mesh Generation and Advanced numerical Methods

DOMAIN

Applied Mathematics, Computation and Simulation

THEME

Numerical schemes and simulations

Contents

Project-Team GAMMA

Creation of the Project-Team: 2019 June 01

Keywords

Computer sciences and digital sciences

A6.2. – Scientific computing, Numerical Analysis & Optimization

A6.2.7. – High performance computing

A6.2.8. – Computational geometry and meshes

A6.5.1. – Solid mechanics

A6.5.2. – Fluid mechanics

Other research topics and application domains

B5.2.3. – Aviation

B5.2.4. – Aerospace

B9.5.1. – Computer science

B9.5.2. – Mathematics

B9.5.3. – Physics

B9.5.5. – Mechanics

1 Team members, visitors, external collaborators

Research Scientists

- Frederic Alauzet [Team leader, INRIA, Senior Researcher, HDR]
- Paul-Louis George [Inria (pnp), Emeritus]
- David Marcum [MSU, Advanced Research Position]
- Julien Vanharen [INRIA, Senior Researcher, from Oct 2022]

PhD Students

- Sofiane Benzait [CEA]
- Francesco Clerici [INRIA]
- Eloi Guilbert [INRIA, from Sep 2022]
- Lucien Rochery [INRIA]
- Lucille Marie Tenkès [Inria, until Aug 2022]

Technical Staff

- Matthieu Maunoury [INRIA, Engineer, SED / Gamma]
- Cosimo Tarsia Morisco [INRIA, Engineer]

Administrative Assistant

- Maria Ronco [INRIA]

2 Overall objectives

Numerical simulation has been booming over the last thirty years, thanks to increasingly powerful numerical methods, computer-aided design (CAD) and the mesh generation for complex 3D geometries, and the coming of supercomputers (HPC). The discipline is now mature and has become an integral part of design in science and engineering applications. This new status has lead scientists and engineers to consider numerical simulation of problems with ever increasing geometrical and physical complexities. A simple observation of this chart

$$\text{CAD} \longrightarrow \text{Mesh} \longrightarrow \text{Solver} \longrightarrow \text{Visualization / Analysis}$$

shows: **no mesh = no simulation** along with **"bad" mesh = wrong simulation**. We have concluded that the mesh is at the core of the classical computational pipeline and a key component to significant improvements. Therefore, the requirements on meshing methods are an ever increasing need, with increased difficulty, to produce high quality meshes to enable reliable solution output predictions in an automated manner. These requirements on meshing or equivalent technologies cannot be removed and all approaches face similar issues.

In this context, `Gamma` team was created in 1996 and has focused on the development of robust automated mesh generation methods in 3D, which was clearly a bottleneck at that time when most of the numerical simulations were 2D. The team has been very successful in tetrahedral meshing with the well-known software `Ghs3d` [30, 31] which has been distributed worldwide so far and in hexahedral meshing with the software `Hexotic` [37, 38] which was the first automated full hex mesher. The team has also worked on surface meshers with `Yams` [28] and `BLSurf` [24] and visualization with `Medit`. Before `Medit`, we were unable to visualize in real time 3D meshes !

In 2010, `Gamma3` team has replaced `Gamma` with the choice to focus more on meshing for numerical simulations. The main goal was to emphasize and to strengthen the link between meshing technologies and numerical methods (flow or structure solvers). The metric-based anisotropic mesh adaptation strategy has been very successful with the development of many error estimates, the generation of highly anisotropic meshes, its application to compressible Euler and Navier-Stokes equations [20], and its extension to unsteady problems with moving geometries [23] leading to the development of several softwares `Feflo.a/AMG-Lib, Wolf, Metrix, Wolf-Interpol`. A significant accomplishment was the high-fidelity prediction of the sonic boom emitted by supersonic aircraft [21]. We were the first to compute a certified aircraft sonic boom propagation in the atmosphere, thanks to mesh adaptation. The team has started to work on parallelism with the development of the multi-thread library `LPlib` and the efficient management of memory using space filling curves, and the generation of large meshes (a billion of elements) [36]. Theoretical work on high-order meshes has been also done [29].

Today, numerical simulation is an integral part of design in engineering applications with the main goal of reducing costs and speeding up the process of creating new design. Four main issues for industry are:

- Generation of a discrete surface mesh from a continuous CAD is the last non-automated step of the design pipeline and, thus, the most human time consuming
- High-performance computing (HPC) for all tools included in the design loop
- The cost in euros of a numerical simulation
- Certification of high-fidelity numerical simulations by controlling errors and uncertainties.

Let us now discuss in more details each of these issues.

Generating a discrete surface mesh from a CAD geometry definition has been the numerical analysis Achille's heel for the last 30 years. Significant issues are far too common and range from persistent translation issues between systems that can produce ill defined geometry definitions to overwhelming complexity for full configurations with all components. A geometry definition that is ill defined often does not perfectly capture the geometry's features and leads to a bad mesh and a broken simulation. Unfortunately, CAD system design is essentially decoupled from the needs of numerical simulation and is largely driven by the those of manufacturing and other areas. As a result, this step of the numerical simulation pipeline is still labor intensive and the most time consuming. There is a need to develop alternative geometry processes and models that are more suitable for numerical simulations.

Companies working on high-tech projects with high added value (Boeing, Safran, Dassault-Aviation, Ariane Group, ...) consider their design pipeline inside a HPC framework. Indeed, they are performing complex numerical simulations on complex geometries on a daily-basis, and they aim at using this in a shape-optimization loop. Therefore, any tools added to their numerical platform should be HPC compliant. This means that all developments should consider hybrid parallelism, *i.e.,* to be compatible with distributed memory architecture (MPI) and shared memory architecture (multi-threaded), to achieve scalable parallelism.

One of the main goals of numerical simulation is to reduce the cost of creating new designs (e.g reduce the number of wind-tunnel and flight tests in the aircraft industry). The emergence of 3D printers is, in some cases, making tests easier to perform, faster and cheaper. It is thus mandatory to control the cost of the numerical simulations, in other word, it is important to use less resources to achieve the same accuracy. The cost takes into account the engineer time as well as the computing resources needed to perform the numerical simulation. The cost for one simulation can vary from 15 euros for simple models (1D-2D), to 150 euros for Reynolds-averaged Navier-Stokes (3D) stationary models, or up to 15 000 euros for unsteady models like LES or Lattice-Boltzmann [1]. It is important to know that a design loop is equivalent to performing between 100 and 1 000 numerical simulations. Consequently, the need for more efficient algorithms and processes is still a key factor.

Another crucial point is checking and certification of errors and uncertainties in high-fidelity numerical simulations. These errors can come from several sources:

---

[1] Source Valéo and Safran Tech.

i) modeling error (for example via turbulence models or initial conditions),

ii) discretization error (due to the mesh),

iii) geometry error (due to the representation of the design) and

iv) implementation errors in the considered software.

The error assessment and mesh generation procedure employed in the aerospace industry for CFD simulations relies heavily on the experience of the CFD user. The inadequacy of this practice even for geometries frequently encountered in engineering practice has been highlighted in studies of the AIAA [2] CFD Drag Prediction Workshops [39] and High-Lift Prediction Workshops [42, 43]. These studies suggest that the range of scales present in the turbulent flow cannot be adequately resolved using meshes generated following what is considered best present practices. In this regard, anisotropic mesh adaptation is considered as the future, as stated in the NASA report "CFD Vision 2030 Study: A Path to Revolutionary Computational Aerosciences" [44] and the study dedicated to mesh adaptation [40].

These preoccupations are the core of the GAMMA project scientific program. To answer the first issue, GAMMA will focus on designing and developing a geometry modeling framework specifically intended for mesh generation and numerical simulation purposes. This is a mandatory step for automated geometry-mesh and mesh adaptation processes with an integrated geometry model. To answer the last three issues, the GAMMA team will work on the development of a high-order mesh-adaptive solution platform compatible with HPC environment. To this end, GAMMA will pursue its work on advanced mesh generation methods which should fulfill the following capabilities:

i) geometrical adaptive modeling,

ii) solution adaptation,

iii) high-order,

iv) multi-elements (structured or not), and

v) using hybrid scalable parallelism.

Note that items i) to iv) are based on the well-posed metric-based theoretical framework. Moreover, GAMMA will continue to work on robust flow solvers, solving the turbulent Navier-Stokes equations from second order using Finite Volume - Finite Element numerical scheme to higher-order using Flux Reconstruction (FR) method.

The combination of adaptation - high-order - multi-elements coupled with appropriate error estimates is for the team the way to go to reduce the cost of numerical simulations while ensuring high-fidelity in a fully automated framework.

3 Research program

The main axes are:

- Geometric Modeling:
  - High-fidelity discrete CAD kernel.
  - Continuous parametric CAD kernel.
- Enhanced Generic Meshing Algorithm:
  - Adaptation (extreme anisotropy, metric-aligned, metric-orthogonal).
  - High-order (tetrahedra, hexahedra, boundary layer, adapted).
  - Larges meshes (tetrahedra, hexahedra, adapted).

---

[2] The American Institute of Aeronautics and Astronautics.

  - Moving mesh methods for moving geometries.
- Toward Certified Solutions to the Navier-Stokes Equations:
  - Flow solver and adjoints (Finite Volumes, Finite Elements, Flux Reconstruction).
  - Error estimates and correctors.
- Advanced Mesh and Solution Visualisation:
  - Pixel exact rendering (High-Order mesh, High-Order solution).
  - Pre-processing and post-processing.

4 Application domains

Our research in mesh generation, mesh adaptation and certification of the Numerical Simulation Pipeline finds applications in several different domains such as aviation and aerospace but also all fields where computation and simulation are used: fluid mechanics, solid mechanics, solving wave equations (acoustic, electromagnetism...), energy or biomedical.

5 New software and platforms

5.1 New software

5.1.1 GHS3D

Keywords: Tetrahedral mesh, Delaunay, Automatic mesher

Functional Description: GHS3D is an automatic volume mesher

URL: `http://www.meshgems.com/volume-meshing.html`

Contact: Frederic Alauzet

Participants: Paul Louis George, Adrien Loseille, Frederic Alauzet

5.1.2 HEXOTIC

Keywords: 3D, Mesh generation, Meshing, Unstructured meshes, Octree/Quadtree, Multi-threading, GPGPU, GPU

Functional Description: Input: a triangulated surface mesh and an optional size map to control the size of inner elements.

Output: a fully hexahedral mesh (no hybrid elements), valid (no negative jacobian) and conformal (no dangling nodes) whose surface matches the input geometry.

The software is a simple command line that requires no knowledge on meshing. Its arguments are an input mesh and some optional parameters to control elements sizing, curvature and subdomains as well as some features like boundary layers generation.

URL: `https://team.inria.fr/gamma/gamma-software/hexotic/`

Contact: Loic Marechal

Participant: Loic Marechal

Partner: Distene

5.1.3 FEFLOA-REMESH

Keywords: Scientific calculation, Anisotropic, Mesh adaptation

Functional Description: FEFLOA-REMESH is intended to generate adapted 2D, surface and volume meshes by using a unique cavity-based operator. The metric-aligned or metric-orthogonal approach is used to generate high quality surface and volume meshes independently of the anisotropy involved.

URL: `https://pages.saclay.inria.fr/adrien.loseille/index.php?page=softwares`

Contact: Adrien Loseille

Participants: Adrien Loseille, Frederic Alauzet, Rémi Feuillet, Lucien Rochery, Lucille-Marie Tenkes

5.1.4 Metrix

Name: Metrix: Error Estimates and Mesh Control for Anisotropic Mesh Adaptation

Keywords: Meshing, Metric, Metric fields

Functional Description: Metrix is a software that provides by various ways metric to govern the mesh generation. Generally, these metrics are constructed from error estimates (a priori or a posteriori) applied to the numerical solution. Metrix computes metric fields from scalar solutions by means of several error estimates: interpolation error, iso-lines error estimate, interface error estimate and goal oriented error estimate. It also contains several modules that handle meshes and metrics. For instance, it extracts the metric associated with a given mesh and it performs some metric operations such as: metric gradation and metric intersection.

URL: `https://pages.saclay.inria.fr/frederic.alauzet/software.html`

Contact: Frederic Alauzet

Participants: Adrien Loseille, Frederic Alauzet

5.1.5 Wolf

Keyword: Scientific calculation

Functional Description: Numerical solver for the Euler and compressible Navier-Stokes equations with turbulence modelling. ALE formulation for moving domains. Modules of interpolation, mesh optimisation and moving meshes. Wolf is written in C++, and may be later released as an opensource library. FELiScE was registered in July 2014 at the Agence pour la Protection des Programmes under the Inter Deposit Digital Number IDDN.FR.001.340034.000.S.P.2014.000.10000.

URL: `https://pages.saclay.inria.fr/frederic.alauzet/software.html`

Contact: Frederic Alauzet

Participants: Frederic Alauzet, Adrien Loseille, Rémi Feuillet, Lucille-Marie Tenkes, Francesco Clerici, Cosimo Tarsia Morisco

5.1.6 Wolf-Bloom

Keyword: Scientific calculation

Functional Description: Wolf-Bloom is a structured boundary layer mesh generator using a pushing approach. It start from an existing volume mesh and insert a structured boundary layer by pushing the volume mesh. The volume mesh deformation is solved with an elasticity analogy. Mesh-connectivity optimizations are performed to control volume mesh element quality.

URL: `https://pages.saclay.inria.fr/frederic.alauzet/software.html`

Contact: Frederic Alauzet

Participants: Adrien Loseille, David Marcum, Frederic Alauzet

5.1.7 Wolf-Elast

Keyword: Scientific calculation

Functional Description: Wolf-Elast is a linear elasticity solver using the P1 to P3 Finite-Element method. The Young and Poisson coefficient can be parametrized. The linear system is solved using the Conjugate Gradient method with the LUSGS preconditioner.

URL: `https://pages.saclay.inria.fr/frederic.alauzet/software.html`

Contact: Frederic Alauzet

Participants: Adrien Loseille, Frederic Alauzet

5.1.8 Wolf-Interpol

Keyword: Scientific calculation

Functional Description: Wolf-Interpol is a tool to transfer scalar, vector and tensor fields from one mesh to another one. Polynomial interpolation (from order 2 to 4) or conservative interpolation operators can be used. Wolf-Interpol also extract solutions along lines or surfaces.

URL: `https://pages.saclay.inria.fr/frederic.alauzet/software.html`

Contact: Frederic Alauzet

Participants: Adrien Loseille, Frederic Alauzet

5.1.9 Wolf-MovMsh

Keyword: Scientific calculation

Functional Description: Wolf-MovMsh is a moving mesh algorithm coupled with mesh-connectivity optimization. Mesh deformation is computed by means of a linear elasticity solver or a RBF interpolation. Smoothing and swapping mesh optimization are performed to maintain good mesh quality. It handles rigid bodies or deformable bodies, and also rigid or deformable regions of the domain. High-order meshes are also handled

URL: `https://pages.saclay.inria.fr/frederic.alauzet/software.html`

Contact: Paul Louis George

Participants: Adrien Loseille, Frederic Alauzet

5.1.10 Wolf-Nsc

Keyword: Scientific calculation

Functional Description: Wolf-Nsc is numerical flow solver solving steady or unsteady turbulent compressible Euler and Navier-Stokes equations. The available turbulent models are the Spalart-Almaras and the Menter SST k-omega. A mixed finite volume - finite element numerical method is used for the discretization. Second order spatial accuracy is reached thanks to MUSCL type methods. Explicit or implicit time integration are available. It also resolved dual (adjoint) problem and compute error estimate for mesh adaptation.

URL: `https://pages.saclay.inria.fr/frederic.alauzet/software.html`

Contact: Frederic Alauzet

Participants: Adrien Loseille, Frederic Alauzet

5.1.11 Wolf-Shrimp

Keywords: Scientific calculation, Domain partitionning

Scientific Description: Wolf-Shrimp is a generic mesh partitioner for parallel mesh generation and parallel computation. It can partition planar, surface (manifold and non manifold), and volume domain. Several partitioning methods are available: Hilbert-based, BFS, BFS with restart. It can work with or without weight function and can correct the partitions to have only one connected component.

Functional Description: Wolf-Shrimp is a generic mesh partitioner for parallel mesh generation and parallel computation. It can partition planar, surface (manifold and non manifold), and volume domain. Several partitioning methods are available: Hilbert-based, BFS, BFS with restart. It can work with or without weight function and can correct the partitions to have only one connected component.

URL: `https://pages.saclay.inria.fr/frederic.alauzet/software.html`

Contact: Frederic Alauzet

Participants: Adrien Loseille, Frederic Alauzet

5.1.12 Wolf-Spyder

Keyword: Scientific calculation

Functional Description: Wolf-Spyder is a metric-based high-order mesh quality optimizer using vertex smoothing and edge/face swapping.

URL: `https://pages.saclay.inria.fr/frederic.alauzet/software.html`

Contact: Frederic Alauzet

Participants: Adrien Loseille, Frederic Alauzet

5.1.13 Wolf-Xfem

Keyword: Scientific calculation

Functional Description: Wolf-Xfem is a tool providing the mesh of the intersection between a surface mesh and a volume mesh.

URL: `https://pages.saclay.inria.fr/frederic.alauzet/software.html`

Contact: Frederic Alauzet

Participants: Adrien Loseille, Frederic Alauzet

5.1.14 ViZiR4

Name: ViZiR4

Keywords: Visualization, Pixel-exact rendering, Instant rendering, High order methods

Functional Description: Its main features are: - Light, simple and interactive visualization software. - Surface and volume (tetrahedra, pyramids, prisms, hexahedra) meshes. - Pixel exact rendering of high-order solutions on straight elements. - Almost pixel exact rendering on curved elements (high-order meshes). - Post-processing tools, such as picking, isolines, clipping, capping.

URL: `https://pyamg.saclay.inria.fr/vizir4.html`

Publications: hal-01686714, hal-02950321, hal-03539257

Contact: Adrien Loseille

Participants: Adrien Loseille, Matthieu Maunoury, Frederic Alauzet

6 New results

6.1 Numerical simulations on *GPU* with the `GMlib` v3.0 library

Participants: Loïc Maréchal *(correspondant)*, Julien Vanharen.

The whole library was completely rewritten to implement an automatic finite-element shader generation that converts a simple user source code into an *OpenCL* source that is in compiled on the GPU at run time. The library handles all meshing data structures, from file reading, renumbering and vectorizing for efficient access on the GPU, and transfer to the graphic card, all automatically and transparently. With this framework, the user can focus on the calculation part of the code, known as kernel, as all the rest is taken care of by the library. The OpenCL language was chosen as it is hardware agnostic and runs on any GPU (Intel, Nvidia and AMD) and can also use the multicore and vector capacities of modern CPUs.

Julien Vanharen developed a basic heat solver using the v3.0 as a test case so we could validate the software with various boundary conditions, calculation scheme, unstructured meshes and different memory access patterns with success. Even with basic calculation which does not stress the full GPU's power, we achieved two orders of magnitude greater speed against a single CPU core and one order of magnitude compared to a multithreaded implementation.

6.2 High Order Meshing: from a straight mesh to a curved one

Participants: Loïc Maréchal *(correspondant)*.

Works continued on `P1toPk`, a software that transform any first order hybrid mesh (triangles, quads, tets, pyramids, prisms and hexes) into a second order one while respecting a prescribe surface curvature. Efforts were made on boundary layers curving, which was challenging because jacobian validity is harder to guarantee as the elements get highly stretched, and a lot effort were also made to speed up the code by optimizing mathematical operations and parallelizing them. The code is now mature enough to be sent to industrial users for real life usage and we are waiting for valuable feedback in the present year.

6.3 Pixel-exact rendering for high-order meshes and solutions

Participants: Matthieu Maunoury *(correspondant)*, Adrien Loseille.

We are developing `ViZiR 4`, a visualization software with pixel exact rendering to address the high-order visualization challenges [26, 15]. `ViZiR 4` is bundled as a light, simple and interactive high-order meshes and solutions visualization software. It is based on OpenGL 4 core graphic pipeline. The use of OpenGL Shading Language (GLSL) allows to perform pixel exact rendering of high order solutions on straight elements (without extra subdivision or ray casting) and almost pixel exact rendering on curved elements (high-order meshes). `ViZiR 4` enables the representation of high order meshes (up to degree 4) and high order solutions (up to degree 10) with pixel exact rendering. Unlike other visualization software (ParaView [34], TecPlot [46], FieldView [33], Ensight [22], Medit [27], Vizir (OpenGL legacy based version)

[35], Gmsh [32]), there is no subdivision process that is expensive nor visualization error that has to be controlled. Moreover, the subdivision of the curved entities is done on the fly on GPU which leaves the RAM memory footprint at the size of the loaded mesh. Furthermore, in comparison with standard rendering techniques based on legacy OpenGL, the use of OpenGL 4 core version improves the speed of rendering, reduces the memory footprint and increases the flexibility. Many post-processing tools, such as picking, hidding surfaces, isolines, clipping, capping, are integrated to enable on the fly the analysis of the numerical results.

6.4 Fast high-order tetrahedral mesh correction and metric-based curving for P2 cavity remeshing

Participants: Lucien Rochery *(correspondant)*, Adrien Loseille.

We aim to deal with three main topics around high-order mesh adaptation with applications to classic a posteriori curving.

One, a new method to untangle high-order Bézier meshes is introduced. It maximizes directly the minimum control coefficient over the mesh. Under some conditions, the $\max\min$ problem can be recast as a linear program, solvable exactly by specialized algorithms. The best possible position of an edge control point is obtained in as long as to compute and differentiate dependent control coefficients five times.

Two, Riemannian edge length minimization is used to prescribe metric-based mesh-interior curvature. This generalizes weakly the unit mesh definition on linear meshes, and is very fast as only edge scale problems are solved. Through metric gradation on surface metrics, surface curvature is naturally propagated to the interior by length minimization. Similarly, the mesh-intrinsic metric can be used to curve boundary layers. In realistic settings, a so-called *back mesh* holding the discrete metric field is kept unchanged throughout remeshing. This prevents anisotropy loss through repeated interpolation. It also leads to the metric at a point P within an element K being the result of two interpolations. First, at the control points of K using the back mesh. Then, at P using those. This metric at P is differentiated, in order to provide derivatives of anisotropic geometric quantities.

Three, the cavity operator if extended to P^2 meshes. It rewrites topological changes (insertions, collapses, generalized swaps) as element deletions followed by point starring. The P^2 cavity operator is modular and distinguishes between two types of curvature: prescribed and necessary. Here, CAD/P^3 surrogate projection and Riemannian length minimization are used for the surface and interior respectively.

Necessary curvature results from subsequent high-order untangling, as prescribed curvature is not expected to yield valid cavities.

Numerical results focus on a posteriori curving of difficult cases. Boundary control points are projected. Metric-based curvature is used to propagate surface curvature, curve boundary layers, and follow natural curvature of a metric field resulting from CFD adaptation. The simplex-based Jacobian smoother corrects the resulting meshes. Examples are based on 3D real-world geometries encountered in Computational Fluid Dynamics (CFD). This framework allow us to curve highly anisotropic meshes with around 10 million elements within minutes.

6.5 Unstructured anisotropic mesh adaptation for 3D RANS turbomachinery applications

Participants: Frédéric Alauzet *(correspondant)*, Adrien Loseille, Julien Vanharen.

We aim to demonstrate the viability and efficiency of unstructured anisotropic mesh adaptation techniques to turbomachinery applications. The main difficulty in turbomachinery is the periodicity of the domain that must be taken into account inthe solution mesh-adaptive process. The periodicity is strongly enforced in the flow solver using ghost cells to minimize the impact on the source code. For the mesh adaptation, the local remeshing is done in two steps. First, the inner domain is remeshed with frozen periodic frontiers, and, second, the periodic surfaces are remeshed after moving geometric entities from one side of the domain to the other. One of the main goal of this work is to demonstrate how mesh adaptation, thanks to its automation, is able to generate meshes that are extremely difficult to envision and almost impossible to generate manually. This study only considers feature-based error estimate based on the standard multi-scale Lp-interpolation error estimate. We presents all the specific modifications that have been introduced in the adaptive process to deal with periodic simulations used for turbomachinery applications. The periodic mesh adaptation strategy is then tested and validated on the LS89 high pressure axial turbine vane and the NASA Rotor 37 test cases.

6.6 Mixed-element mesh adaptation for CFD simulations

Participants: Frédéric Alauzet *(correspondant)*, Julien Vanharen, Adrien Loseille, Cosimo Tarsia Morisco.

Due to their various nature, physical phenomena that we seek to capture in CFD simulations may have specific specific mesh requirements. For example, to solve the boundary layer, some numerical schemes favor structured meshes respecting alignment with the boundary of the domain, while these constraints are not necessary elsewhere. Our approach is to use the techniques of metric-based mesh adaptation to generate a mixed-element mesh that can fulfill these different mesh requirements. This approach is based on the *metric-orthogonal* point-placement, creating some structured parts from the intrinsic directional information bore by the metric-field. Some unstructured areas may remain where structure is not needed. The main goals of this work are to improve the orthogonality of the output mesh and its alignment with the metric field. This work has three main axes. First, we have improved the preprocessing gradation step to smooth the metric field and improve the orthogonality of the final mesh. Then, we have studied two methods to obtain quadrilaterals: one using an a priori quadrilaterals recombination, the other detecting straightforwardly the orthogonal patterns during the re-meshing step. Finally, the work on the solver Wolf has been carried on and corrected to perform robust and accurate simulations on mixed-element meshes. These three developments were embodied in a mixed-element adaptation loop. The first two topics are detailed in what follows.

Enhanced metric gradation correction The previously described generation method highly relies on the metric field. However, a metric field computed from a solution during the adaptation process is most of the time quite messy and shows abrupt size variations. In standard mesh adaptation, it leads to low-quality elements. In orthogonal mesh adaptation, it additionally breaks the alignment and the structure of the output mesh. An additional step to smooth the input metric field is therefore required. In the context of mixed-element mesh adaptation, this gradation correction process has been modified to improve the number and the quality of the quadrilaterals in the final mesh. Further developments have been considered on this topic, in particular to increase the robustness of the method. Results have been published in [47].

***A posteriori* and *a priori* mesh generation** Metric-orthogonal point-placement is currently used to generate quasi-structured meshes with right-angled triangles where the metric is the most anisotropic and unit triangles elsewhere. The aim of this work is to recover some quadrilaterals in the structure. To do so, two approaches can be considered: an *a posteriori* quadrilateral recombination based on geometrical criteria, and an *a priori* quadrilateral detection. The latter is more straightforward because it uses directly the point-placement information. A framework was established to set up this method. Developments and preliminary results were presented in [48].

In order to obtain a correct metric field on hybrid meshes, a robust hybrid solver is mandatory. When dealing with 2D (3D) elements different from triangles (tetrahedra), the most tricky aspect is the gradient formulation. This is due to the fact that within a Finite Elements interpolation framework, the gradient on an element with more than three nodes (i.e. not simplicial complex) is not element-wise constant. This brings many added difficulties to the flux balance computation. A first attempt at performing inviscid and laminar simulations on hybrid meshes was to approximate gradients on quadrilaterals with its iso-barycenter values [48]. The extension of this formulation to turbulent flows, however highlighted a lack of efficiency and robustness. For this reason, a APFE (APproximated Finite-Element) method [41] has been implemented and as well extended to a implicit time integration scheme. This approach turns out to be very efficient and robust in many fully-structured mesh verification cases. The extension to 3D cases (prisms and pyramids) is ongoing. Details can be found in [17].

An extension of the Vertex-Centered Mixed-Element-Volume MUSCL scheme to 3D mixed-element meshes is proposed for the convective fluxes. This scheme involves a clever exploitation of the FE gradients, which can be efficiently implemented as P^1 FE gradients of particular sub-tetrahedra inside prismatic and/or pyramidal elements. Conversely, diffusive fluxes are discretized using an original extension of the APproximated Finite Element (APFE) method to 3D elements. This method could fit for any element, as long as a robust FE gradient and a dual volume with planar dual facets is provided.

The combination of the proposed discretization strategies for convective and diffusive fluxes shows a certain robustness for regular and not regular mixed element meshes as well interesting results for turbulent flows. The two dimensional analysis highlighted the influence of gradient formulation in source terms for highly anisotropic meshes. This aspect is under investigation in 3D. On the whole, the methodology proposed seems to be a promising candidate to tackle mixed-element adaptation in some future works. More details in [45].

6.7 Coupled flow and adjoint solver

Participants: Francesco Clerici *(correspondant)*, Frédéric Alauzet.

When solving the RANS equations, usually one decouples the equations relative to the mean-flow and the equations relative to turbulence. This division provides two separated systems to be solved at each time step, one relative to the mean-flow and the other relative to the turbulence. This presents two main drawbacks: in the flow solver, the Jacobian of the system lacks of the terms bounding the mean-flow and the turbulence, and this can slow down the residual convergence. The second drawback regards the adjoint problem, which consists into a linear system assembled with the transpose of the Jacobian matrix of the residuals and, on the right-hand side, the derivative of an aeronautical coefficient with respect to the flow variables. A Jacobian missing the coupling terms between the mean-flow and the turbulence provides a null adjoint turbulent viscosity, and this is a limitation in the development of more complex discretization error estimates. We have therefore developed a 2D version of the coupled flow and adjoint solver which includes in the Jacobian the coupling terms between the mean-flow and the turbulence. When have tested this method on the 2D geometry provided for the 4^{th} CFD AIAA High Lift Prediction Workshop, and the result provided several features to emerge, such as a high mesh refinement inside the boundary layer of the leading edges, and inside the regions of high turbulence destruction. The work is pursued in collaboration with Philippe Spalart (Boeing), and is presented in [25].

6.8 Turbulent error estimate

Participants: Francesco Clerici *(correspondant)*, Frédéric Alauzet.

Goal-oriented mesh adaptation is a methodology used to adapt the mesh in order to minimize the discretization error commited on a functional depending on the solution. As an intermediate step, one finds an upper bound to such discretization error taking the form of a weighted sum of the interpolation errors of the solution, and such upper bound is called *error estimate*. Regarding Wolf and the RANS equations, up to now we have focused only on the mean-flow part of such an error estimate, meaning that the terms coming from turbulence have been neglected. The scope of this work is to enrich the error estimate with the information coming from the turbulence. In particular, the methodology has been tested on the 2D geometry provided for the 4^{th} CFD AIAA High Lift Prediction Workshop, providing high mesh refinements on the boundaries of the turbulent regions. Also this work is pursued in collaboration with Philippe Spalart (Boeing), and is presented in [25].

7 Bilateral contracts and grants with industry

7.1 Bilateral contracts with industry

Participants: Frédéric Alauzet *(correspondant)*, Adrien Loseille.

- Safran Tech
- Ariane Group
- Lemma

8 Partnerships and cooperations

8.1 European initiatives

8.1.1 Horizon Europe

NEXTAIR

Participants: Julien Vanharen *(correspondant)*, Frédéric Alauzet.

NEXTAIR project on cordis.europa.eu

Title: NEXTAIR - multi-disciplinary digital - enablers for NEXT-generation AIRcraft design and operations

Duration: From September 1, 2022 to August 31, 2025

Partners:

- INSTITUT NATIONAL DE RECHERCHE EN INFORMATIQUE ET AUTOMATIQUE (INRIA), France
- THE UNIVERSITY OF SHEFFIELD (USFD), United Kingdom
- IMPERIAL COLLEGE OF SCIENCE TECHNOLOGY AND MEDICINE (Imperial), United Kingdom
- AIRBUS OPERATIONS SAS (AIRBUS OPERATIONS), France
- ETHNICON METSOVION POLYTECHNION (NATIONAL TECHNICAL UNIVERSITY OF ATHENS - NTUA), Greece
- SAFRAN SA, France
- UNIVERSITA DEGLI STUDI DI CAGLIARI (UNICA), Italy
- OFFICE NATIONAL D'ETUDES ET DE RECHERCHES AEROSPATIALES (ONERA), France
- DEUTSCHES ZENTRUM FUR LUFT - UND RAUMFAHRT EV (DLR), Germany
- FUNDACION CENTRO DE TECNOLOGIAS DE INTERACCION VISUAL Y COMUNICACIONES VICOMTECH (VICOM), Spain

- DASSAULT AVIATION, France
- ASOUTI V & SIA OE, Greece
- OPTIMAD ENGINEERING SRL (Optimad srl), Italy
- IRT ANTOINE DE SAINT EXUPERY, France
- ERDYN CONSULTANTS SARL, France
- ROLLS-ROYCE PLC, United Kingdom

Inria contact: Pietro Congedo

Coordinator:

Summary: Radical changes in aircraft configurations and operations are required to meet the target of climate-neutral aviation. To foster this transformation, innovative digital methodologies are of utmost importance to enable the optimisation of aircraft performances.

NEXTAIR will develop and demonstrate innovative design methodologies, data-fusion techniques and smart health-assessment tools enabling the digital transformation of aircraft design, manufacturing and maintenance. NEXTAIR proposes digital enablers covering the whole aircraft life-cycle devoted to ease breakthrough technology maturation, their flawless entry into service and smart health assessment. They will be demonstrated in 8 industrial test cases, representative of multi-physics industrial design, maintenance problems and environmental challenges and interest for aircraft and engines manufacturers.

NEXTAIR will increase high-fidelity modelling and simulation capabilities to accelerate and derisk new disruptive configurations and breakthrough technologies design. NEXTAIR will also improve the efficiency of uncertainty quantification and robust optimisation techniques to effectively account for manufacturing uncertainty and operational variability in the industrial multi-disciplinary design of aircraft and engine components. Finally, NEXTAIR will extend the usability of machine learning-driven methodologies to contribute to aircraft and engine components' digital twinning for smart prototyping and maintenance.

NEXTAIR brings together 16 partners from 6 countries specialised in various disciplines: digital tools, advanced modelling and simulation, artificial intelligence, machine learning, aerospace design, and innovative manufacturing. The consortium includes 9 research organisations, 4 leading aeronautical industries providing digital-physical scaled demonstrator aircraft and engines and 2 high-Tech SME providing expertise in industrial scientific computing and data intelligence.

9 Scientific production

9.1 Major publications

[1] F. Alauzet and A. Loseille. 'High Order Sonic Boom Modeling by Adaptive Methods'. In: *Journal Of Computational Physics* 229 (2010), pp. 561–593. URL: `http://dx.doi.org/10.1016/j.jcp.2009.09.020`.

[2] T. Amari, A. Canou, J.-J. Aly, F. Delyon and F. Alauzet. 'Magnetic cage and rope as the key for solar eruptions'. In: *Nature* 554.7691 (Feb. 2018), pp. 211–215. DOI: `10.1038/nature24671`. URL: `https://hal-cea.archives-ouvertes.fr/cea-01872913`.

[3] A. Belme, F. Alauzet and A. Dervieux. 'An a priori anisotropic Goal-Oriented Error Estimate for Viscous Compressible Flow and Application to Mesh Adaptation'. working paper or preprint. Nov. 2018. URL: `https://hal.inria.fr/hal-01927113`.

[4] H. Borouchaki and P.-L. George. *Meshing, Geometric Modeling and Numerical Simulation 1*. John Wiley & Sons, Inc., 30th Oct. 2017. DOI: `10.1002/9781119384335`. URL: `https://hal-utt.archives-ouvertes.fr/hal-02562289`.

[5] S. Chaillat, S. P. Groth and A. Loseille. 'Metric-based anisotropic mesh adaptation for 3D acoustic boundary element methods'. In: *Journal of Computational Physics* 372 (Nov. 2018), pp. 473–499. DOI: `10.1016/j.jcp.2018.06.048`. URL: `https://hal.archives-ouvertes.fr/hal-01895636`.

[6] R. Feuillet, M. Maunoury and A. Loseille. 'On pixel-exact rendering for high-order mesh and solution'. In: *Journal of Computational Physics* 424 (Jan. 2021), p. 109860. DOI: `10.1016/j.jcp.2020.109860`. URL: `https://hal.inria.fr/hal-02950321`.

[7] P. L. George, F. Alauzet, A. Loseille and L. Maréchal. *Meshing, Geometric Modeling and Numerical Simulation 3*. 1. Wiley; Wiley, 30th Nov. 2020. DOI: `10.1002/9781119788072.index`. URL: `https://hal.inria.fr/hal-03146790`.

[8] P.-L. George, H. Borouchaki, P.-J. Frey, P. Laug and E. Saltel. 'Mesh Generation and Mesh Adaptivity: Theories and Techniques'. In: *Encyclopedia of Computational Mechanics, Volume 1: Fundamentals*. ISBN 0-470-84699-2, E. Stein, R. de Borst and T.J.R. Hughes ed., 2nd edition 2008. Wiley InterScience, 2004. Chap. 17, pp. 497–523.

[9] A. Loseille and F. Alauzet. 'Optimal 3D Highly Anisotropic Mesh Adaptation based on the Continuous Mesh Framework'. In: *18th International meshing roundtable*. Salt-Lake City, UT, USA: Springer, 2009. URL: `http://dx.doi.org/10.1007/978-3-642-04319-2%5C_20`.

[10] A. Loseille, A. Dervieux and F. Alauzet. 'Fully anisotropic goal-oriented mesh adaptation for 3D steady Euler equations'. In: *Journal Of Computational Physics* 229 (2010), pp. 2866–2897. URL: `http://dx.doi.org/10.1016/j.jcp.2009.12.021`.

[11] A. Loseille and R. Feuillet. 'Vizir: High-order mesh and solution visualization using OpenGL 4.0 graphic pipeline'. In: *2018 - AIAA Aerospace Sciences Meeting, AIAA SciTech Forum*. kissimmee, United States, Jan. 2018, pp. 1–13. DOI: `10.2514/6.2018-1174`. URL: `https://hal.inria.fr/hal-01686714`.

[12] A. Loseille and R. Löhner. 'Robust Boundary Layer Mesh Generation'. Anglais. In: *Proceedings of the 21st International Meshing Roundtable*. Ed. by X. Jiao and J.-C. Weill. Springer Berlin Heidelberg, 2012, pp. 493–511. DOI: `10.1007/978-3-642-33573-0_29`. URL: `http://hal.inria.fr/hal-00935315`.

[13] E. Montreuil, W. Ghedhaifi, V. Chmielarski, V. François, F. Gand and A. Loseille. 'Numerical Simulation of contrail formation on the Common Research Model wing/body/engine configuration'. In: *AIAA Aviation and Aeronautics Forum and Exposition 2018*. Atlanta, United States, June 2018. URL: `https://hal.archives-ouvertes.fr/hal-01961143`.

9.2 Publications of the year

International peer-reviewed conferences

[14] F. Alauzet, F. Clerici, A. Loseille, C. Tarsia-Morisco and J. Vanharen. 'Some progress on CFD high lift prediction using metric-based anisotropic mesh adaptation'. In: *AIAA SCITECH 2022 Forum*. AIAA SCITECH 2022 - Forum. San Diego, United States: American Institute of Aeronautics and Astronautics, 3rd Jan. 2022, p. 0388. DOI: `10.2514/6.2022-0388`. URL: `https://hal.archives-ouvertes.fr/hal-03542592`.

[15] M. Maunoury, R. Feuillet and A. Loseille. 'Using ViZiR 4 to analyze the 4th AIAA CFD High Lift Prediction Workshop Simulations'. In: AIAA SciTech 2022 Forum. San Diego / Virtual, United States: American Institute of Aeronautics and Astronautics, 3rd Jan. 2022. DOI: `10.2514/6.2022-2246`. URL: `https://hal.inria.fr/hal-03539257`.

[16] F. Miralles, B. Sauvage, S. Wornom, F. Alauzet, B. Koobus and A. Dervieux. 'Simulation of massively separated flows using hybrid turbulence models ans mesh adaptation'. In: DLES13 2022 - Direct and Large-Eddy Simulation 13. Udine, Italy, 26th Oct. 2022. URL: `https://hal.inria.fr/hal-03929769`.

[17] L.-M. Tenkes, A. Loseille and F. Alauzet. 'Quasi-structured anisotropic quad-dominant mesh adaptation using metric-orthogonal approach'. In: SCITECH 2022 - AIAA SciTech Forum. San Diego, United States: American Institute of Aeronautics and Astronautics, 3rd Jan. 2022. DOI: `10.2514/6.2022-1246`. URL: `https://hal.inria.fr/hal-03536970`.

Scientific books

[18] A. Dervieux, F. Alauzet, A. Loseille and B. Koobus. *Mesh Adaptation for Computational Fluid Dynamics 1: Continuous Riemannian Metrics and Feature-based Adaptation*. Numerical methods in engineering series. Wiley-Iste, 2022. URL: `https://hal.inria.fr/hal-03930979`.

Reports & preprints

[19] F. M. Gerosa, D. Corti, F. Alauzet and M. A. Fernández. *3D Nitsche-XFEM method for fluid-structure interaction with immersed thin-walled solids*. 30th Dec. 2022. URL: `https://hal.inria.fr/hal-03916638`.

9.3 Cited publications

[20] F. Alauzet and A. Loseille. 'A decade of progress on anisotropic mesh adaptation for Computational Fluid Dynamics'. In: *Comput. Aided Des.* 72 (2016), pp. 13–39.

[21] F. Alauzet and A. Loseille. 'High Order Sonic Boom Modeling by Adaptive Methods'. In: *J. Comp. Phys.* 229 (2010), pp. 561–593.

[22] Ansys Inc. *Ensight*. `https://www.ansys.com/en/products/platform/ansys-ensight`.

[23] N. Barral, G. Olivier and F. Alauzet. 'Metric-based anisotropic mesh adaptation for three-dimensional time-dependent problems involving moving geometries'. In: *J. Comp. Phys.* 331 (2017), pp. 157–187.

[24] H. Borouchaki, P. Laug and P. George. 'Parametric surface meshing using a combined advancing-front – generalized-Delaunay approach'. In: *International Journal for Numerical Methods in Engineering* 49.1-2 (2000), pp. 233–259.

[25] F. Clerici, P. Spalart and F. Alauzet. 'Coupled adjoint solver and turbulent error estimate for anisotropic mesh adaptation in high-fidelity RANS simulations.' In: *AIAA J.* (2022).

[26] R. Feuillet, M. Maunoury and A. Loseille. 'On pixel-exact rendering for high-order mesh and solution'. In: *Journal of Computational Physics* 424 (Jan. 2021), p. 109860. DOI: `10.1016/j.jcp.2020.109860`. URL: `https://hal.inria.fr/hal-02950321`.

[27] P. J. Frey. *Medit: An interactive mesh visualization software, INRIA Technical Report RT0253*. 2001.

[28] P. Frey. 'About surface remeshing'. In: *Proceedings of the 9th international meshing roundtable*. New Orleans, LO, USA, 2000, pp. 123–136.

[29] P. L. George and H. Borouchaki. 'Construction of tetrahedral meshes of degree two'. In: *International Journal for Numerical Methods in Engineering* 90.9 (2012), pp. 1156, 1182.

[30] P. George and H. Borouchaki. '"Ultimate" robustness in meshing an arbitrary polyhedron'. In: *Int. J. Numer. Meth. Engng* 58.7 (2003), pp. 1061–1089.

[31] P. George, F. Hecht and E. Saltel. 'Automatic mesh generator with specified boundary'. In: *Comput. Methods Appl. Mech. Engrg.* 92 (1991), pp. 269–288.

[32] C. Geuzaine and J.-F. Remacle. 'Gmsh: A 3-D finite element mesh generator with built-in pre- and post-processing facilities'. In: *International Journal for Numerical Methods in Engineering* 79.11 (2009), pp. 1309–1331.

[33] Intelligent Light. *FieldView*. `http://www.ilight.com/en/products/fieldview-18`.

[34] KitWare Inc. *ParaView*. `https://www.paraview.org/`.

[35] A. Loseille, H. Guillard and A. Loyer. *An introduction to Vizir: an interactive mesh visualization and modification software*. EOCOE, Rome, Italy. 2016.

[36] A. Loseille, F. Alauzet and V. Menier. 'Unique cavity-based operator and hierarchical domain partitioning for fast parallel generation of anisotropic meshes'. In: *Computer-Aided Design* 85 (2017), pp. 53–67.

[37] L. Maréchal. 'A new approach to octree-based hexahedral meshing'. In: 2001, pp. 209–221.

[38] L. Maréchal. ‘Advances in Octree-Based All-Hexahedral Mesh Generation: Handling Sharp Features’. In: *Proceedings of the 18th International Meshing Roundtable*. Salt Lake City, UT, USA, 2009, pp. 65–84.

[39] D. Mavriplis. ‘Results from the 3rd Drag Prediction Workshop using NSU3D unstructured mesh solver’. In: *45th AIAA Aerospace Sciences Meeting*. AIAA-2007-0256, Reno, NV, USA, Jan. 2007.

[40] M. Park, J. Krakos, T. Michal, A. Loseille and J. Alonso. ‘Unstructured Grid Adaptation: Status, Potential Impacts, and Recommended Investments Toward CFD Vision 2030’. In: *46th AIAA Fluid Dynamics Conference*. AIAA Paper 2016-3323, Washington, D.C., USA, 2016.

[41] G. Puigt, V. Auffray and J.-D. Müller. ‘Discretisation of diffusive fluxes on hybrid grids’. In: *Journal of Computational Physics* 229.5 (2010), pp. 1425–1447.

[42] C. Rumsey and J. Slotnick. ‘Overview and Summary of the Second AIAA High Lift Prediction Workshop’. In: *Journal of Aircraft* 52.4 (2015), pp. 1006–1025.

[43] C. Rumsey, J. Slotnick, M. Long, R. Stuever and T. Wayman. ‘Summary of the first AIAA CFD High-Lift Prediction Workshop’. In: *Journal of Aircraft* 48.6 (2011), pp. 2068–2079.

[44] J. Slotnick, A. Khodadoust, J. Alonso, D. Darmofal, W. Gropp, E. Lurie and D. Mavriplis. *CFD Vision 2030 Study: A path to revolutionary computational aerosciences*. CR-2014-218178. NASA, Mar. 2014.

[45] C. Tarsia Morisco, L.-M. Tenkes and F. Alauzet. ‘Extension of the Vertex-Centered Mixed-Element-Volume MUSCL scheme to mixed-element meshes’. In: *AIAA SCITECH 2023 Forum*. 2023, p. 0832. DOI: `10.2514/6.2023-0832`.

[46] TecPlot Inc. *TecPlot*. `https://www.tecplot.com/`.

[47] L.-M. Tenkes and F. Alauzet. ‘Size Gradation Control for Anisotropic Hybrid Meshes’. In: *Numerical Geometry, Grid Generation and Scientific Computing*. Vol. 143. Lecture Notes in Computational Science and Engineering. Springer International Publishing, May 2021, pp. 127–139. DOI: `10.1007/978-3-030-76798-3\_8`. URL: `https://hal.inria.fr/hal-03536960`.

[48] L.-M. Tenkes, F. Alauzet and J. Vanharen. ‘Hybrid anisotropic mesh adaptation using metric-orthogonal approach’. In: *AIAA Scitech 2021 Forum*. Virtual, United States: American Institute of Aeronautics and Astronautics, Jan. 2021. DOI: `10.2514/6.2021-1779`. URL: `https://hal.inria.fr/hal-03163869`.