Anisotropic mesh adaptation and numerical schemes for the LES simulation of interfaces

Keywords: Large Eddy Simulation (LES), Combustion, Two-phase flows, Atomization, Anisotropic mesh, Mesh adaptation, Numerical schemes, Unstructured mesh.

Company: SAFRAN Tech, Paris-Saclay (F)

SAFRAN supervisor: R. Mercier

Host research lab: LEGI, Grenoble (F)

LEGI supervisors: G. Ghigliotti, G. Balarac

Contract type: « CIFRE » PhD fellowship

PhD thesis beginning: Q1 2022

Position duration: 36 months

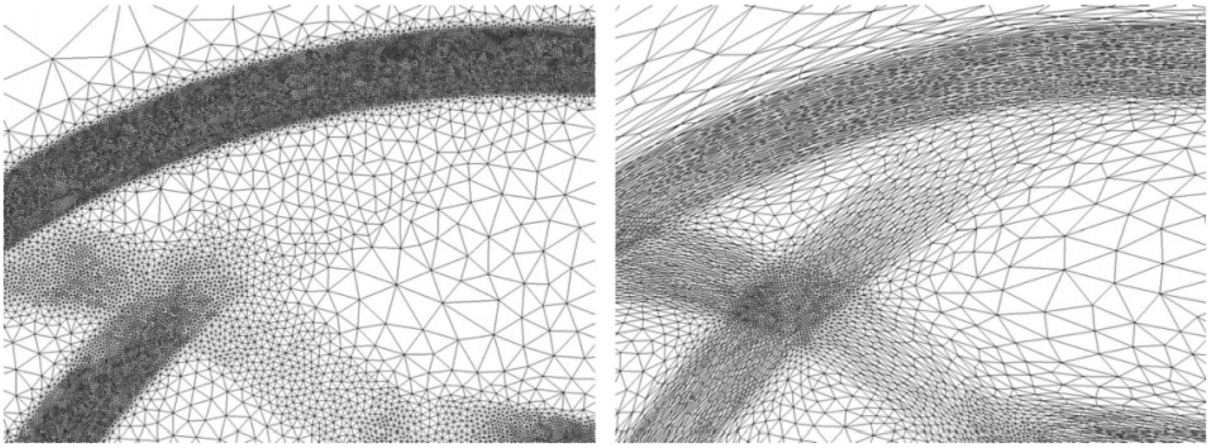


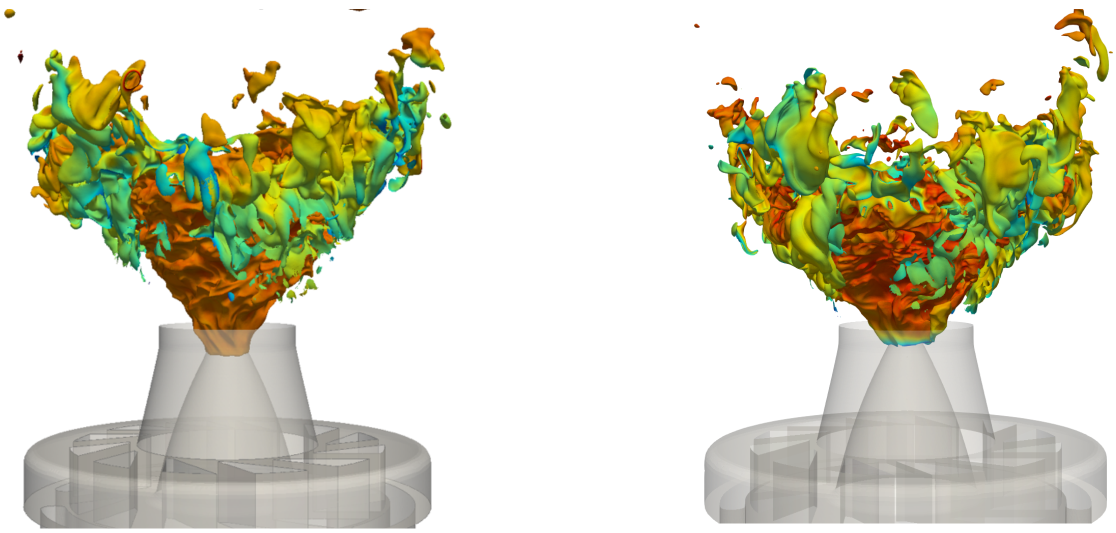
Figure 1: Example of the impact of mesh anisotropy for a triangular mesh. Left: isotropic mesh; Right: anisotropic mesh. Figure extracted from [1].

Subject description:

SAFRAN group relies more and more on high-fidelity numerical simulations for aeronautic engine design. For example, Large Eddy Simulations are used in the design offices to characterize the combustion chamber operability or for evaluating the topology of a multiphase flow at a fuel injector outlet.

In order to describe accurately the complex and multi-physical processes found in these different modules with a limited computational cost, it is possible to use mesh adaptation methods. Using those methods allows solving accurately the zones of interest (flame, mixing layer, liquid-gas interface, and so on) for the prediction of the design-relevant quantities by dynamically adjusting the spatial location of the computational effort.

Since a few years, CORIA Lab (Rouen, F), Safran Tech and LEGI Lab collaborate for the development, within the multi-physics numerical platform YALES2 [2], of an unstructured mesh adaptation method based on the MMG3D library [3], created by C. Dobrzynski at INRIA (Bordeaux, F). This adaptation method has been used in reacting flow cases representative of industrial configurations in order to finely describe the flame wrinkling and its interaction with turbulence [5]. This allowed a computational cost sensibly lower than the one needed with a homogeneous mesh (see Fig. 2). This methodology is also used to dynamically follow a liquid-gas interface and finely resolve the atomization phenomena of a fuel spray [6], or the topology of an oil flow in a gear box [7] (see Fig. 3).



Static mesh: 877 M elements / $\Delta_{\mathrm{FF}} = 150\ \mu m$ **Adapted mesh**: 260 M elements / $\Delta_{\mathrm{FF}} = 150\ \mu m$

Figure 2: Iso-contour of the progress variable @ 0,7, colored with the temperature field. Simulation of the PRECCINSTA burner presented at the ICNC conference [5].

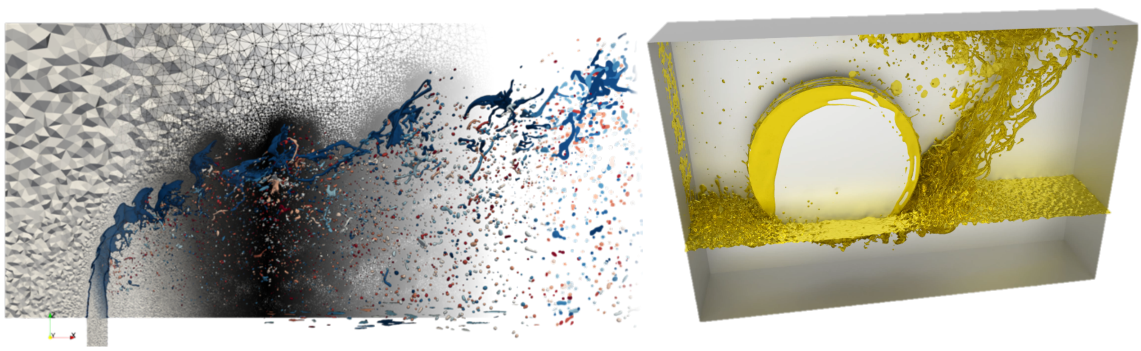


Figure 3 : (Left) Simulation of a fuel jet in a transverse air flow with dynamic mesh adaptation [6]. (Right) Simulation of the oil churning of a disk rotating in an oil bath [7].

A strong limitation of the current adaptation methodology is the imposition of a mesh size, or metrics, which needs to be homogeneous in all directions, so that remeshing is achieved with quasi-isotropic tetrahedral elements. The physical phenomena that one wants to finely describe (flame front, phase interface, boundary layer, …) being in most cases oriented in a preferential direction, substantial gains in the computational effort may be obtained by taking into account the preferential directions, i.e., by remeshing with anisotropic tetrahedral elements.

The objective of the proposed PhD thesis is then to extend the mesh adaptation capabilities to anisotropic meshes, while guaranteeing the accuracy and the robustness of the numerical simulations. So, the first challenge to address is to extend the mesh adaptation capabilities to include anisotropic mesh elements. In practice, one will need to define new mesh adaptation criteria that will consider the physics of the considered flows (see for instance [2]), but considering the different spatial dimensions, in order to be able to define an anisotropic metrics (as a tensor). The second objective is the development of new numerical schemes. Indeed, the anisotropic mesh elements may severely degrade the accuracy of the numerical schemes, and even compromise the stability of the simulation. The goal will be to propose numerical schemes having a spatial order independent of the mesh quality, and thus of its anisotropic structure. In order to attain this goal, we will try to extend the work initiated at LEGI [7] on the derivation of finite-volume numerical schemes based on the differentiation of the quantities integrated over the control volume from pointwise quantities. This approach should allow considering the shape of the control volumes, including for highly deformed and anisotropic meshes. In the case of a multiphase flow where the mesh anisotropy will be located at the phase interface, a stake will be to predict the interface curvature, needed for the computation of surface tension forces.

Project organization:

The PhD thesis will be composed of three parts:

i) The development of a formalism for the anisotropic mesh adaptation including the definition of the adaptation criteria.

ii) The development of robust and accurate numerical schemes for anisotropic meshes.

iii) The validation of these methodologies on test cases of increasing complexity, and their application on test cases representative of the industrial applications.

The PhD student will share its time between the SAFRAN Tech center (Paris-Saclay) within the *Digital Sciences & Technologies* platform (DST) in the *Multiphysics Flows Simulation Methods* (MUST) team, and the Laboratory LEGI (Grenoble). A strong collaboration with the CORIA Laboratory (Rouen) will be realized during the duration of the PhD thesis concerning the aspects of high-performance computing and model implantation. Short stays at CORIA Lab (Rouen) are expected.

Tools and Equipment:

- **Software from CORIA:** YALES2 (R&T Safran Platform, developed at CORIA)
- **Computing resources**: internal resources at SAFRAN Tech, CEA resources (TGCC), national resources through GENCI calls, and European resources through PRACE calls.

References:

[1] F. Alauzet, P.J. Frey, P.L. George, B. Mohammadi (2007) 3D transient fixed point mesh adaptation for time-dependent problems: Application to CFD simulations. Journal of Computational Physics 222, 592–623.

[2] YALES2 web site, http://www.coria-cfd.fr

[3] Benard, P., Balarac, G., Moureau, V., Dobrzynski. C., Lartigue, G. & D'angelo, Y. (2016) Mesh adaptation for large-eddy simulations in complex geometries. International Journal for Numerical Methods in Fluids, 81 (12), 719-740.

[4] Dobrzynski, C. & Frey, P. (2008) 17th international Meshing Roundtable.

[5] Mercier, R., Benard P., Lartigue, G. & Moureau V. (2019) Dynamic adaptation of tetrahedral-based meshes for the simulation of turbulent premixed flames. 17th International Conference on Numerical Combustion, Aachen, Germany.

[6] Leparoux, J., Mercier, R., Moureau, V. & Musaefendic, H. (2018) Primary atomization simulation applied to a jet in crossflow aeronautical injector with dynamic mesh adaptation. Proceedings of ICLASS (July), 22–26.

[7] Cailler, M., Mercier, R. & Moureau, V. (2019) Oil lubrication simulation using sharp interface capturing method and dynamic mesh adaptation. 10th International Conference on Multiphase Flow. Rio de Janeiro, Brazil.

[8] Bernard, M., Lartigue, G., Balarac, G., Moureau & V., Puigt, G. (2020) A framework to perform high-order deconvolution for finite-volume method on simplicial meshes. International Journal for Numerical Methods in Fluids, Vol. 92, no 11, 1551-1583.

Application: **:**

The candidate is expected to have a Master's degree with knowledge in fluid mechanics and a marked taste for numerical methods, code development, modelling and numerical simulation in a high-performance computing (HPC) environment.

To apply to this PhD position, candidates are required to send curriculum vitae, motivation letter and grade transcripts to giovanni.ghigliotti@univ-grenoble-alpes.fr and renaud-c.mercier@safrangroup.com