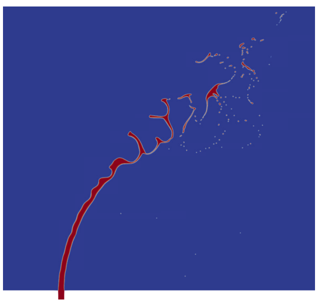
# Anisotropic mesh adaptation for multiphase flows

Robin Barbera[*,†], Giovanni Ghigliotti[‡], Roxane Letournel[†], Guillaume Balarac[*],

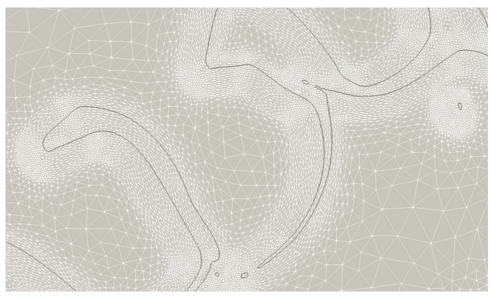
In recent years, the increase in computational power along with the development of algorithms, encouraged the use of Computational Fluid Dynamic (CFD) for industrial applications. In this context, complex configurations are feasible thanks to the use of adaptive mesh refinement techniques, that allow to focus the computational effort in the regions of interest[1]. Nowadays, these methods are relatively mature when isotropic meshes are used, but the use of anisotropic meshes is still hampered by several challenges linked to computational accuracy and mass conservation. Nevertheless, their use can be an important tool to further reduce computational cost, by reducing the number of required mesh elements[2]. This idea seems promising for multiphase flows, as anisotropy is relevant near phase interfaces.

In this presentation, we propose a new anisotropic dynamic remeshing strategy for multiphase flows. This strategy has been implemented in YALES2, a massively parallel, finite-volume, unstructured grid CFD library allowing for both DNS and LES[3]. The phase interface is resolved through the accurate conservative level-set method[4]. The proposed strategy is tested on several configurations, both academic (droplet convection, dam break) and complex (jet in crossflow). A particular focus is made on mass conservation issues during the remeshing step, arising from the need to interpolate the nodal quantities to the new mesh. This issue has already been widely tackled in the literature, but the proposed solutions are not satisfactory for complex configurations, as they are either costly in terms of computing time[5] or in terms of memory allocation and challenging to implement in a massively parallel context[6].

To cope with these limitations, we propose a new interpolation procedure, based on interface curvature, and aiming at the improvement of mass conservation and controlling the numerical diffusion during remeshing. This procedure is coupled with an automatic detection of regions where anisotropic meshes are relevant. Another point of interest lies into the complex coupling between the liquid phase and turbulence in the gas phase. Similarly to static mesh adaptation of turbulent flows[7] highly Anisotropic meshes at the base of the jet can lead to underresolved turbulence, so a particular attention needs to be paid to the turbulent remeshing criterion in order to accurately resolve the structures triggering hydrodynamic instabilities. This framework is tested on several configurations (Fig. 1) with both high and low interface curvature.



(a) Global view



(b) Close up view around the interface (in black)

Figure 1: Jet in crossflow test case. Level-set function showing the liquid phase (in red) and the gazeous phase (blue).

[*]Université Grenoble Alpes, CNRS, Grenoble-INP, LEGI, 38000 Grenoble, France
[†]Safran Tech, Modelling and Simulation, 78772 Magny-les-Hameaux, France
[‡]Université Grenoble Alpes, CNRS, Grenoble-INP, LPSC, 38000 Grenoble, France

[1]R. Janodet et al., *Journal of Computational Physics* **458**, 111075 (2022)
[2]F. Alauzet et al., *Computer-Aided Design* **72**, 13-39 (2016)
[3]YALES2 Wiki, Retrieved January 22, 2025 from https://www.coria-cfd.fr/index.php/YALES2
[4]O. Desjardins et al., *Journal of Computational Physics* **227**, 8395-8416 (2008)
[5]C. Bahbah et al., *Computers & Fluids* **191**, 104223 (2019)
[6]F. Alauzet, *Computer Methods in Applied Mechanics and Engineering* **299**, 116-142 (2016)
[7]R. Barbera et al., *AIAA SCITECH 2025 Forum*, 1378 (2025)