

LARGE EDDY SIMULATION OF NON-CARTESIAN BODIES IN CARTESIAN GRIDS USING IMMERSED BOUNDARY TECHNIQUE

Julien Bodart, France

Supervisors: J.-M. Buchlin, C. Benocci, P. Rambaud, R. Giammanco

The purpose of this project has been the implementation of an Immersed Boundary capability, in MiOma, the new Large Eddy Simulation code developed in the EA department at the Von Karman Institute. This technique (Verzicco, Balaras, 2000), allows to solve the flow around complex geometry within a cartesian grid.

This approach is completely different from the classical body fitted grid one, requiring either curvilinear structure or unstructured mesh. By giving one of three different attributes to each cell (fluid, solid, or boundary cell), a forcing term will be applied in the Navier Stokes equation to take into account the solid location (forcing velocity to zero in the solid for example) and can be implemented in a fully cartesian solver.

Along this project, a general procedure has been implemented in the preprocessor, in order to import generic geometries from a CAD software. The so-called "direct forcing" immersed boundary technique, with linear boundary reconstruction for the velocities is then applied in the solver, keeping the parallel and multidomain philosophy of MiOma.

While the correct working of the technique has been ascertained for fully 3D bodies (i.e. torus or sphere), two test cases periodic in spanwise direction have been particularly investigated:

- Half of a circular cylinder at $Re=150$, using symmetry condition, keeping in this way a steady laminar flow, with a typical "anti cartesian" geometry.
- Full circular cylinder at $Re=300$, as in this range of Reynolds number, the flow is expected to be fully 3D

Finally, a turbulent flow around a circular cylinder at $Re=3900$ has been reached to a developed state, using a newly implemented subgrid scale WALE model to investigate structures coming out from the body.

Side effects of this technique are pressure wiggles apparition and oscillating behavior in the pressure field. Improvement might be needed like special boundary condition for the pressure or grid refinement procedure to get better accuracy. However this technique has been shown to be reliable and very powerful, as a 3D geometry can be now easily investigated, requiring only few minutes of appropriated meshing and CAD manipulation.

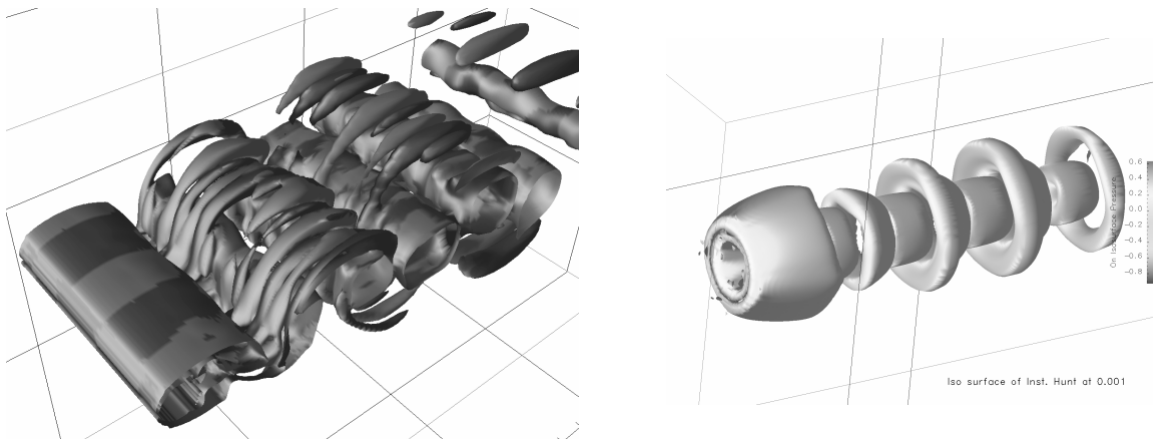


Figure 1: Coherent structures developing around a cylinder and a torus, at $Re=300$