Recent Advances on Immersed Boundary Methods at the Italian Aerospace Research Center

Francesco Capizzano¹, Emiliano Iuliano¹

¹ Fluid Dynamics Laboratory, Italian Aerospace Research Center - CIRA, Via Maiorise, 81043 Capua (CE), Italy

ABSTRACT

An Immersed Boundary (IB) technique has been developed at CIRA in the last seven years and a solid background is reached in the automatic generation of locally refined Cartesian meshes. Experience was matured in wall modelling for compressible two- and three-dimensional high Reynolds flows mainly during the activities of the JT1 – GRA and JT1 – GRC European projects. An Eulerian IB-Method for Water Droplet Impingement was developed in the framework of the EXTICE European project towards ice accretion simulation. A validation campaign was carried out whose results are collected in Ref.[3]. In the last year an eXtra Large Eddy (X-LES) module[5] was added to the pre-existent tool to meet the hybrid RANS-LES topic of the Garteur AG-49 research group[7].

Basically a fully unstructured data management is adopted to deal with Cartesian meshes. A cell-centered finite volume method is used to solve the Reynolds Averaged Navier-Stokes (RANS) equations. The convective and diffusive fluxes are approximated by using a 2nd order accurate in space Central Difference Scheme (CDS). The amount of added artificial diffusion is controlled in each cell by a matrix linked to the Jacobian of the convective fluxes[9] (matrix artificial diffusion MATD). A multistage explicit Runge-Kutta pseudo-time integration is carried out to find a steady state solution to the RANS and the turbulence model equations which are solved simultaneously. Time accurate integration is required for unsteady RANS or X-LES computations. In both cases a dual-time-stepping technique is adopted based on an implicit 2nd order backward discretization for the time derivative. The basic scheme is modified near the wall by means of an IB approach. It consists of a discrete forcing in the momentum equation by means of a direct BC imposition. Proper fluxes are forced at the near-wall cells to satisfy Dirichlet/Neumann conditions at the wall. A turbulent wall model is able to reconstruct non linear quantities at high Reynolds numbers. Numerical schemes and algorithms are optimized to run on vectorial and parallel architectures to make time-affordable three-dimensional numerical experiments. An overview of the RANS solver along with implementation details can be found in Ref. [2]. Examples of current activities are shown in the following sections.

Turbulent flow around a reduced landing gear configuration

The reduced landing gear (RLG) is a simplified four-wheel truck recently proposed as test-case geometry in the Workshop on Benchmark problems for Airframe Noise Computations (BANC-I)[8]. Sponsored by the Aero-Acoustics and Fluid Dynamics Technical Committees (AA/T/FDTC), the Workshop was launched in June 2010 with the aim of asses simulation-based noise-prediction tools in the context of airframe noise. An experimental campaign was carried out at the 1.5m low speed wind tunnel of the National Aerospace Laboratories (NAL) in Bangalore, India (see fig. 1-a). The RLG configuration was placed in a numerical wind tunnel having the same test-section dimensions and flow conditions of the NAL experiment. Starting from a cell root dimension of \( \Delta_{\text{cell}} / D = 5.27 \cdot 10^{-1} \), seven levels of geometry-based refinements were carried out to obtain a near wall cell dimension of \( \Delta_{\text{wall}} / D = 4.11 \cdot 10^{-3} \) and a total number of 8,364,529 cells. Note that, the computational time required to generate automatically the Cartesian mesh was less than half an hour on a single core, Intel Xeon running at 2.8GHz. A low-subsonic turbulent flow around the geometry at \( Re_{\infty} = 1.0 \times 10^6 \), \( M_{\infty} = 0.12 \) (\( U_{\infty} = 40 \text{ m/s} \)) and 0° angle of incidence was simulated. Note that the Reynolds number refers to the RLG wheel diameter. The

Figure 1: RLG experimental and numerical wind-tunnel set-up.
tunnel walls were treated as inviscid as recommended by the BANC-I organizers. A fully turbulent computation was carried out. The approximation of a fully turbulent flow is realistic due to sharp squared axels, beam and post components along with wheel trips on the tire shoulders. A time-accurate numerical experiment was carried out in X-LES mode. In the dual-time the artificial matrix dissipation and residual smoothing were switched on. The solid surface was taken into account by means of an IB technique based on a turbulent wall model[2]. The OpenMP version of the code was used to run on 4 to 8 CPU of the CIRA NEC SX6 vector machine. A non-dimensional time-step of $\Delta t = \Delta t \times U_\infty / D = 6.94 \times 10^{-3}$ was adopted and a total of 55 convective time units ($CTU = t \cdot U_\infty / D$) were simulated. Averaged data were extracted from the time-history by considering only the last five CTUs and compared with the experimental data available from the BANC-I Workshop. A global view of the cutting-plane $z/D = 0.0$ colored by the averaged pressure coefficient is shown in fig. 1-b and a contour map of the vorticity magnitude is shown in fig. 1-c. The numerical solution is compared to experiments in terms of pressure coefficient at the same quote $z/D = 0.0$ (see fig. in 2-a). The expansion peaks and the pressure plateau zones are well captured by the numerical solution. On the whole numerical and experimental data are aligned each other as on the front wheel and on the rear one. The time histories of the global loads $C_L$, $C_D$ and $C_3$ are shown in fig.2-b. After a transient period of near 10 CTUs the side force oscillates around a mean value of $O(10^{-5})$. This suggests that the time sampling of 55 CTUs represent a sufficient database for future aeroacoustic analyses.

Water droplet impingement on a NACA 64A008 wing tip

The three-dimensional test case is a full-scale reflection plane tail model consisting of the outboard portion of a general aviation business jet tail. The tail tip consists of a semi-cylindrical cap. The tail airfoil is a symmetric 8% thick NACA 64A008 section and it is kept constant from root to tip. The location of maximum thickness for this airfoil section is at jet tail. The tail tip consists of a semi-cylindrical cap. The tail airfoil is a symmetric 8% thick NACA 64A008 section and it is kept constant from root to tip. The location of maximum thickness for this airfoil section is at jet tail.

The research leading to part of these results has received funding from the European Community’s Seventh Framework Programme (FP7/2007-2013) for the Clean Sky Joint Technology Initiative under grant agreement N° CSJU – GAM – GRA – 2008 – 01.
Figure 3: NACA64A008 tail: $\alpha = 0^\circ$.

Figure 4: NACA64A008 tail: $\alpha = 0^\circ$. Section $y/b = 0.75$. Present method (solid), body-conforming reference method (dashed) and experimental data (symbols).

REFERENCES


