11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
July 20 - 25, 2014, Barcelona, Spain

On the fast transient spoiler deployment in a NACA0012 profile using LES techniques combined with AMR and IMB methods

F. Favre[1], O. Antepara[1], O. Lehmkuhl[1,2], R. Borrell[2] and A. Oliva[1]

[1] Heat and Mass Transfer Technological Center, Polytechnical University of Catalonia (UPC), Colom 11, 08222, Terrassa, Barcelona, Spain. cttc@cttc.upc.edu
[2] Termo Fluids, S.L., Avda. Jaquard, 97 1-E, 08222 Terrassa (Barcelona), Spain

Key Words: *Adaptive mesh refinement; Large Eddy Simulations; Immersed Boundary*
Abstract. Due to the high impact in final cost reduction, the development of active systems of load control for wind turbines has recently gained renewed interest. The use of these systems results in a faster and more detailed action over the blade's load than modern pitch-control systems, using aerodynamic control surfaces to locally modify the flux. Numerical simulations of these fluxes are presented in this work as a powerful tool for its understanding. A combination of large-eddy simulations (LES) techniques and the immersed-boundary method was employed in order to meet the conditions imposed by the mobile parts of the control systems embedded in the computational mesh. Use was also made of the Adaptive Mesh Refinement (AMR) method presented in [1] to minimize the amount of computational cells and therefore obtain an adequate resolution of the different length scales. A simulation of the experiment presented by Yeung et al. [2] was performed as a validation of the proposed model. The experiment consists of an aerodynamic profile deploying a control-surface with a $Re=3,5x10^5$ flux.

Mathematical and numerical model

In the present work, the turbulent flow is described by means of LES using symmetry-preserving discretizations. The spatial discretization preserves the symmetry properties of the continuous differential operator, ensure stability and conservation of the global kinetic energy on any grid [3-5].

The fractional-step method is employed to perform the time evolution of the equations. The convective and diffusive terms are explicitly treated with an Adams-Benshfort scheme. When the predictor velocity is calculated, a source term f is employed (IMB method [6,7]) only on two types of nodes: those which are inside an object (interior points) and those which are in the fluid but have an interior neighbor (forcing points). This term modifies the net moment in the cell to specify the adequate velocity value V in those nodes. In the case of interior points, V is the object's velocity at that point, whereas in the case of forcing points V is determined via a linear interpolation using the velocity of the nearest point of the object to the node and three neighbors in the fluid. In this manner f acts upon the predictor velocity, which results in a problem [6]. Therefore, the calculation of the source term of the Poisson equation was modified according to [7], i.e., it is canceled out in the interior cells and corrected having into account the faces intersected by the object in the interface cells.

Moreover, the computational grid is changed using AMR, that approach permit local mesh refinement, the algorithm is presented in [1].

Preliminary results

A numerical simulation of the experiment presented by Yeung et al. [2] has been performed to validate the proposed model. The experiment consists of the rapid deployment of a 0.1c wide superior spoiler, located at 0.7c in a NACA 0012 profile for a $Re=3.5x10^5$ based on the chord of the profile. The simulation studies the transitory response to a deployment carried out in

4.4 dimensionless time units. The evolution of the drag and lift coefficients obtained are in good agreement with the experimental results by Yeung.

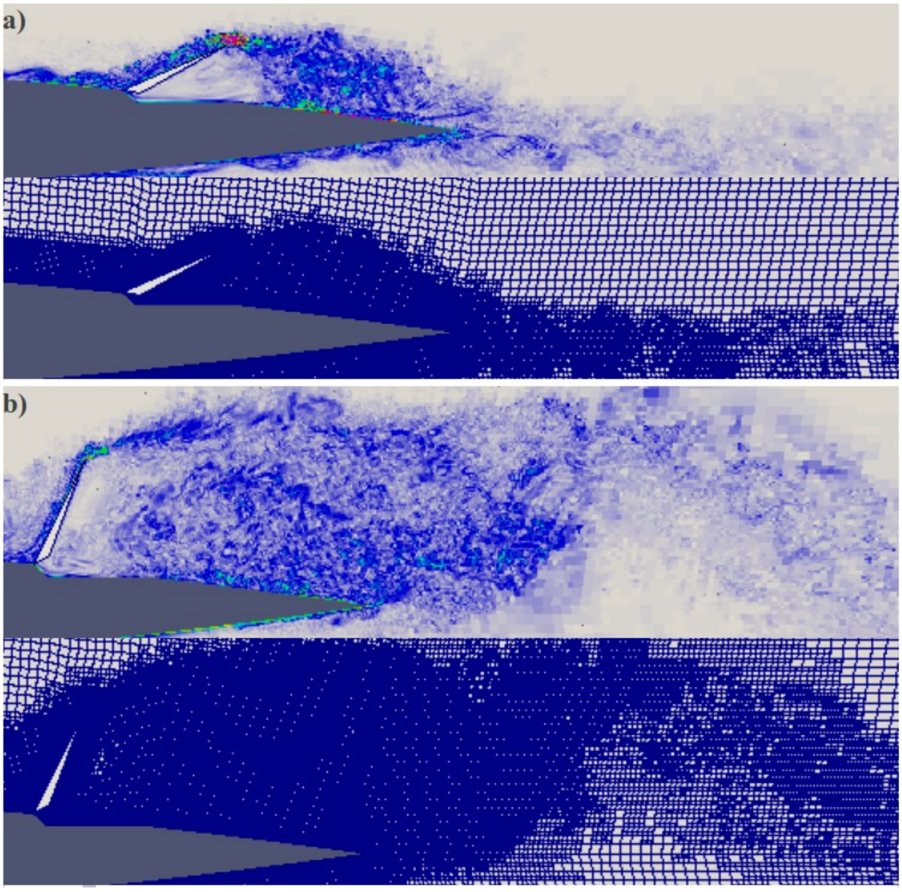

Figure 1: Instantaneous vorticity fields near the spoiler and it's corresponding mesh at: a) t*=1.6, b) t*=2.9

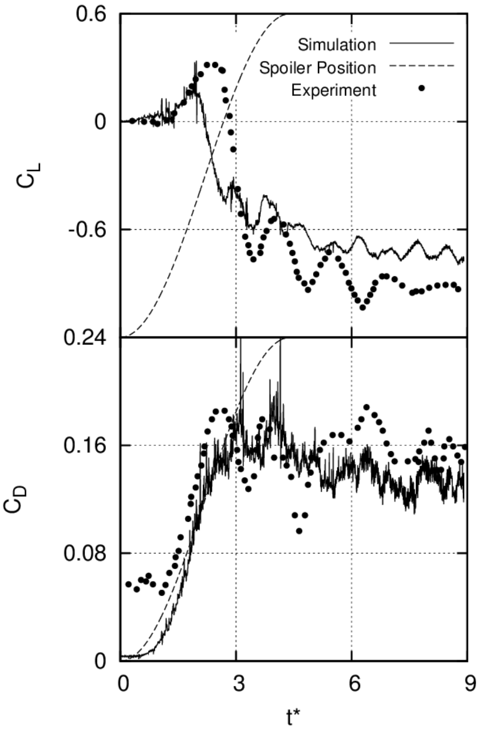

Figure 2: Comparison between computational and experimental results. Lift and Drag coefficients.

REFERENCES

[1] O. Antepara, O. Lehmkuhl, J. Chiva, R. Borrell. "Parallel Adaptive Mesh Refinement Simulation of the Flow Around a Square Cylinder at Re = 22000" 25th International Conference on Parallel Computational Fluid Dynamics. 61:246–250, 2013.

[2] Yeung W. W. H; Xu C; Gu W. "Reduction of Transient Adverse Effects of Spoilers." Journal of Aircraft, Vol 34, No. 4, July-August, pp. 478-484, 1997.

[3] O. Lehmkuhl "Direct numerical simulation and Large-Eddy simulations of the turbulent flow around a NACA-0012 airfoil" 7th International Conference on Computational Heat and Mass Transfer 1–8, 2011

[4] T.J.R. Hughes, L. Mazzei and K.E. Jansen. "Large eddy simulation and the variational multiscale method." Computing and Visualization in Science 3: 47–59, 2000.

[5] R. W. C. P. Verstappen and A. E. P. Veldman. Symmetry-Preserving Discretization of Turbulent Flow. Journal of Computational Physics 187: 343–368, 2003.

[6] Fadlun EA, Verzicco R, Orlandi P, Mohd-Yusof J. "Combined Immersed-Boundary finite-difference methods for three-dimensional complex flow simulations". J Comput Phys 161(1):35-60, 2000.

[7] Kim, Kim and Choi. "An immersed-boundary finite-volume method for simulation of flow in complex geometries." Journal Of Computational Physics 160:705-719, 2001.