6th European Conference on Computational Mechanics (ECCM 6)
7th European Conference on Computational Fluid Dynamics (ECFD 7)
11-15 June 2018, Glasgow, UK

Large eddy simulation of flow past a bluff body using immersed boundary method

Yongxin Chen[1], Kamal Djidjeli[2] and Zheng-Tong Xie[3]

[1 2 3] Faculty of Engineering and Environment, University of Southampton, SO17 1BJ, y.chen@soton.ac.uk; kkd@soton.ac.uk; z.xie@soton.ac.uk

Keywords: *Immersed Boundary Method, Turbulence, LES*

An efficient and accurate finite volume incompressible flow solver with a staggered arrangement in the Cartesian grid has been developed to simulate turbulent flow past a moving bluff body. To simulate flow around a bluff body, an efficient immersed boundary method is implemented to construct geometry in the Cartesian grid, which substantially simplifies the mesh generation for complex geometries. In this paper, flow past a stationary square cylinder at Reynolds number 21,400, based on cylinder width and free stream velocity, is studied by LES with the developed immersed boundary method. The sensitivity of resolution is tested and convergence is achieved. Both experimental and numerical results are used for validation. The global aerodynamic quantities such as lift and drag coefficients and Strouhal number are in good agreement with reference data. The turbulent statistics, in particular in the shear layer and wake regions are compared rigorously with the reference data as well. The developed solver is able to accurately predict the surface force fluctuation, which is extremely challenging from a numerical point of view.