Abstract Submitted
for the DFD15 Meeting of
The American Physical Society

Massive separation around bluff bodies: comparisons among different cfd solvers and turbulence models[1] VINCENZO ARMENIO, AHMAD FAKHARI, Univ of Trieste - Trieste, ANDREA PETRONIO, IEFLUIDS s.r.l., ROBERTA PADOVAN, CHIARA PITTALUGA, GIOVANNI CAPRINO, Cetena, Italy — Massive flow separation is ubiquitous in industrial applications, ruling drag and hydrodynamic noise. In spite of considerable efforts, its numerical prediction still represents a challenge for CFD models in use in engineering. Aside commercial software, over the latter years the opensource software OpenFOAM^R (OF) has emerged as a valid tool for prediction of complex industrial flows. In the present work, we simulate two flows representative of a class of situations occurring in industrial problems: the flow around sphere and that around a wall-mounted square cylinder at $Re = 10000$. We compare the performance two different tools, namely OF and ANSYS CFX 15.0 (CFX) using different unstructured grids and turbulence models. The grids have been generated using SNAPPYHEXMESH and ANSYS ICEM CFD 15.0 with different near wall resolutions. The codes have been run in a RANS mode using $k-\epsilon$ model (OF) and $SST-k-\omega$ (CFX) with and without wall-layer models. OF has been also used in LES, WMLES and DES mode. Regarding the sphere, RANS models were not able to catch separation, while good prediction of separation and distribution of stresses over the surface were obtained using LES, WMLES and DES. Results for the second test case are currently under analysis.

[1]Financial support from COSMO "cfd open source per opera mortta" PAR FSC 2007-2013, Friuli Venezia Giulia

Vincenzo Armenio
Univ of Trieste - Trieste

Date submitted: 31 Jul 2015 Electronic form version 1.4