Abstract Submitted
for the DFD05 Meeting of
The American Physical Society

Computing Bluff Body Flows Using Commercial CFD Software

SUJIT KIRPEKAR, DAVID BOGY, University of California, Berkeley — Commercial CFD codes are increasingly being used to simulate complex engineering flows. Three commercial codes: CFD-ACE v2004, Fluent 6.2.16 and CFX 5.7.1 are examined for their ability to compute the separated flow over a square cylinder. Large Eddy Simulation (LES) results are presented using four SGS models implemented in these commercial codes: the Smagorinsky's model, the dynamic model (Germano et al., 1991), the localized dynamic model (Kim and Menon, 1995) and the WALE model (Nicoud and Ducros, 1999). Global simulation results, time averaged quantities and phase averaged quantities are benchmarked against the experimental results of Lyn and Rodi (Journal of Fluid Mechanics, 1994). All simulations predict the Strouhal number accurately, and simulations employing the dynamic model are excellent in predicting the mean recirculation length and the r.m.s. of the lift coefficient on the cylinder. In terms of flow fluctuations, all simulations over-predict the streamwise component, but under-predict the vertical component. Velocity fluctuations in the wake correlate well with the fluctuation of forces on the cylinder. An examination of the streamlines of the flow indicates that CFD-ACE and Fluent's implementation of the dynamic model offers the best prediction of the vertical displacement of the wake and the size of the shed vortex. Finally, the addition of 10% upwind differencing to the convective terms is also investigated.

Sujit Kirpekar
University of California, Berkeley

Date submitted: 04 Aug 2005

Electronic form version 1.4