An Investigation of Numerical and Modeling Issues Regarding CFD-Predictions of Velocity in a Stirred Tank with the RANS k-w Turbulence Model

P. Farber, K. Farber, S. Minnerup, M. Märtin, J. Gräbel, M. Motovilov, F. Bator, M. Wieschalla, C. Lambertz IMH - Institute of Modelling and High-Performance Computing, Hochschule Niederrhein, University of Applied Sciences, Krefeld, Germany

Abstract

In industrial applications of Computational Fluid Dynamics (CFD) the consideration of best practice is of increasing interest. Additionally the rising availability of parallel compute clusters makes the minimization of numerical errors in CFD simulations possible letting the modeling errors become obvious. The current work describes a systematic investigation of different modeling strategies for the velocity in a stirred tank in CFD simulations. We discuss the iteration error as well as mesh convergence and compare the results with physical experiments. For all system models the RANS k-w turbulence model is used. On the basis of the experiments of Costes and Couderc [1] simulation experiments with several combinations of different submodels like multiple reference frame, sliding mesh, one-phase pure water and two-phase (air and water) volume of fluid (VOF) were performed. The iteration error for each simulation experiment is demonstrated and discussed. Meshes of different size (e.g. 2 mill. cells, 5 mill. cells, 10 mill cells) for each set of combinations of submodels were utilized and the mesh convergence explored. In comparison with the physical experimental data the modeling error is displayed and disputed. At last parallel performance on compute clusters are presented.

[1] J. Costes, J.P. Couderc, 1988: "Study by laser Doppler anemometry of the turbulent flow induced by a Rushton turbine in a stirred tank"; 6th European Conference on Mixing