Evaluation of OpenFOAM with the German VDI guideline 3783/9 for environmental meteorology

Michael Sturm, michael.sturm@uni-siegen.de **Jörg Franke**, joerg.franke@uni-siegen.de	University of Siegen, 57076 Siegen, Germany

Abstract

In the field of wind engineering and urban flow the use of Computational Fluid Dynamics (CFD) becomes more and more important. Consequently, the CFD codes have to be validated for that kind of applications. In Germany the Association of German Engineers (Verein Deutscher Ingenieure, VDI) published an evaluation guideline for prognostic microscale wind field models, and hence also for general purpose CFD codes [1]. The guideline contains a general evaluation, a scientific evaluation and five validation test cases for the flow around buildings and obstacles, including a 2D beam, a wall mounted cube, a wall mounted cube rotated by 45° against the approach flow, a cuboid and an array of 21 cuboids. For all cases mean velocities are available from experiments for comparison with CFD results.

The five validation test cases were simulated with OpenFOAM 1.6.x. To calculate the statistically steady turbulent flow around the buildings the Reynolds Averaged Navier-Stokes equations are solved together with the standard k-ε turbulence model. The purely hexahedral meshes used were generated by Franke [2] for the evaluation of FLUENT 6.2 and also used by Wevers et al. [3] for the evaluation of CFX 11.0. The grids' structure and spacing complies with the requirements of the guideline. The approach flow is prescribed by analytical profiles for an equilibrium atmospheric boundary layer over a rough wall [4]. To achieve a homogeneous atmospheric boundary layer the ground of the simulation domain is also rough (with the same roughness as at the inlet), using OpenFOAM's rough wall function. For the outlet a constant pressure is defined and the lateral boundaries as well as the upper boundary are symmetry planes. The density is constant and the viscosity is that of air. Three different divergence schemes are used, namely the upwind, linearUpwind and SFCD-scheme.

The computed mean velocity components are compared with the corresponding experimental results. For the validation metric a hit rate is defined, that takes both absolute and relative differences between the computed and measured velocity components into account. The comparison between the numerical and experimental data has to be carried out for all data points (up to 10^3) as well as for a subset close to the building, in the so called near field. A hit rate of 66% for each velocity component is required for a successful evaluation.

The simulations are showing that OpenFOAM 1.6.x achieves the required hit rates for all five setups, i.e. the evaluation has been successfully. Moreover, a comparison with other commercial CFD codes shows that the achieved hit rates are in the same range. From the present study, it can be concluded that OpenFOAM 1.6.x is well suited for applications in the field of wind engineering and urban flow.

Key words: urban flow, OpenFOAM, evaluation, wind engineering, VDI 3783/9, RANS, standard k-ε model

References

[1] VDI: *Environmental meteorology - Prognostic microscale wind field models - Evaluation for flow around buildings and obstacles* , VDI guideline 3783, Part 9. Beuth Verlag, Berlin 2005.

[2] J. Franke: *Evaluating FLUENT V6.2 with the German VDI guideline for obstacle resolving micro-scale models. Proceedings of the Fourth International Symposium on Computational Wind Engineering (CWE2006)*, 793 - 796. Yokohama, 2006.

[3] C. Wevers, J. Franke, R. Höfler: *Validierung von CFD-Simulationsprogrammen anhand der VDI Richtlinie 3783*, Bauingenieur, 82:S2 - S5, 2008. Published in German.

[4] P. J. Richards, R.P. Hoxey: *Appropriate boundary conditions for computational wind engineering models using k-epsilon model*, Journal of Wind Engineering and Industrial Aerodynamics 46 & 47, 145-153, 1993