VALIDATION OF TWO RANS SOLVERS WITH FLOW DATA OF THE FLAT ROOF *MICHELSTADT* CASE

A. Rakai (1), J. Franke (2)

(1) Budapest University of Technology and Economics, Department of Fluid Dynamics, 1111 Budapest, Hungary
(2) University of Siegen, Institute of Fluid- and Thermodynamics, 56068 Siegen, Germany

Presenting author email: rakai@ara.bme.hu

ABSTRACT

To further increase the confidence in the results of numerical simulations of flow and dispersion phenomena in urban environments thorough verification and validation of the numerical models is required. Wind tunnel measurements of an idealized Central-European city, *Michelstadt*, which are publicly available in the CEDVAL-LES database were used here to validate the open source code OpenFOAM 1.7.1 and the commercial code Ansys Fluent 13. The results of the flow field computations were compared graphically and with the metric hit rate to the experimental data. Unstructured tetrahedral and polyhedral meshes were used with different resolutions to investigate the advantages and shortcomings of the polyhedral meshing. With both codes and mesh types the qualitative agreement between numerical simulation and experiment is good for the mean velocities, with a minimum hit rate of 0.64 for the coarse polyhedral mesh. The Reynolds stresses on the other hand are consistently under-predicted with both codes on all meshes. Validation of the two codes with this realistic urban geometry shows the capabilities and shortcomings of RANS CFD modelling for regulatory purposes in urban air quality modelling.

1. INTRODUCTION

Quality assurance of Computational Wind Engineering can be obtained by proper verification and validation of the codes used. OpenFOAM 1.7.1 has already been validated against the VDI guideline (Franke et al. 2011 ICWE) and the Mock Urban Setting Test (MUST) case (Rakai et al. 2010). In the framework COST 732 (Schatzmann et al. 2010) several Computational Fluid Dynamics (CFD) codes were tested against the MUST case and a real city, Oklahoma's wind tunnel measurement data, but not OpenFOAM. The two test cases proved to be either too simple or too complex, so a new test case in between was investigated in the Environmental Wind Tunnel Laboratory of the University of Hamburg (Fischer et al. 2010) which is used here for further validation of OpenFOAM 1.7.1 and Ansys Fluent 13.

2. VALIDATION EXPERIMENT MICHELSTADT

The used dataset is part of CEDVAL-LES, a collection of data for validation of Large Eddy Simulation (LES) models (http://www.mi.uni-hamburg.de/Data-Sets.6339.0.html). For the Michelstadt case both flow and dispersion measurements were done, but here only the flow results will be discussed. The geometry is an idealized Central-European city centre placed in the Atmospheric Boundary Layer (ABL) modelled by roughness elements. Two component velocity data time series were collected with Laser Doppler Velocimetry (LDV) in 40 vertical profiles, 2 horizontal planes and 3 street canyon planes (see Figure 1). For the approach flow 3 component measurements were carried out.

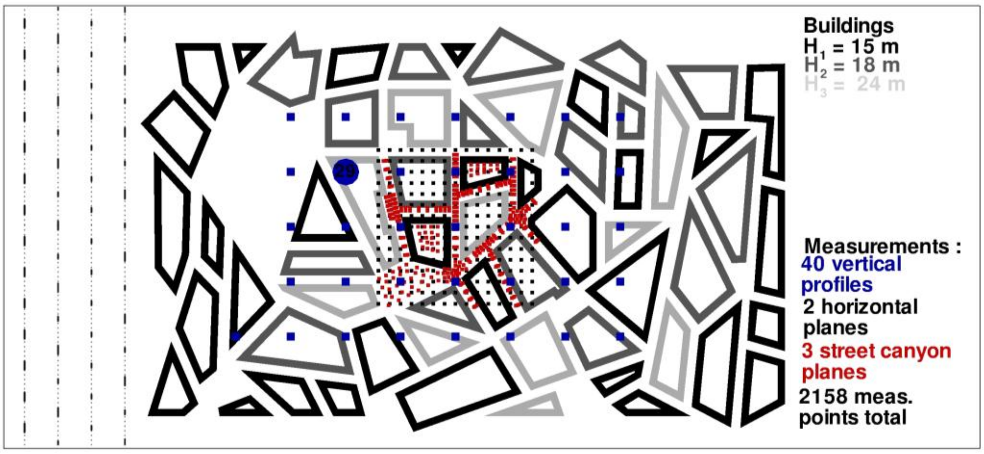

Figure 1: Computational domain with roughness elements, buildings and measurement positions.

3. COMPUTATIONAL SETUP

The computational domain was defined to correspond with the COST 732 Best Practice Guideline (Franke et al. 2010 IJEP) (Figure 1), which resulted in a 1575x900x168 m domain, with a distance of the buildings of $11H_3$ from the inflow, $9.4H_3$ from the outflow and at least $6H_3$ from the top boundaries, where H_3 = 24 m is the highest buildings' height. The computations were done in full scale while the experiment was done at a scale of 1:225. As can also be seen in Figure 1, four lines of the wind tunnel's roughness elements were included in the mesh as the first buildings are in the wake of them. The height of the roughness elements is 9 and 18 m in full scale, so they are relatively high compared to the buildings and the corresponding aerodynamic roughness height z_0 = 1.53 m is unattainable with reasonable meshes.

Unstructured Delaunay tetrahedral grids were generated in Ansys Icem CFD, with three different resolutions, fine, medium and coarse. To investigate the influence of the mesh type, a polyhedral mesh was generated from each tetrahedral mesh with Ansys Fluent 13. The number of cells of each of the six meshes can be seen in Table 1. It must be noted that the polyhedral mesh decreases the number of cells approximately with a ratio of 4, but the number of cell faces does not decrease that much.

	coarse	medium	fine
polyhedral	$1.73\cdot10^6$ (P3)	$3.21\cdot10^6$ (P2)	$6.17\cdot10^6$ (P1)
tetrahedral	$6.65\cdot10^6$ (T3)	$13.17\cdot10^6$ (T2)	$26.79\cdot10^6$ (T1)

Table 1: Number of cells of the six meshes (and their abbreviations).

As inflow boundary condition, a power law profile (exponent 0.27, with a reference velocity U_{ref} = 6.11 m/s defined at z_{ref} = 100 m) fitted to the measured velocity values was given. The turbulent kinetic energy and its dissipation profiles were calculated from the measured approach flow values by their definition and equilibrium assumption. At the top of the domain the measurement values corresponding to that height were fixed. The lateral boundaries were treated as smooth solid walls, as the domain's extension is the same as the wind tunnel width. The floor, roughness elements and buildings were also defined as smooth walls. Standard wall functions were used.

The Reynolds Averaged Navier Stokes Equations were solved with standard k-ε turbulence model and the SIMPLE method was used for pressure-velocity coupling. For OpenFOAM convective terms were first discretized with first order upwind schemes to help convergence, then the momentum divergence terms were changed to the cell limited linear upwind scheme (OpenFOAM 2010). With second order discretization for all variables not all of the computations were stable, so those are not included in this investigation. When the residuals became unstable the equations were substantially underrelaxed, which was already necessary for the first order solutions on the polyhedral meshes. The tetra meshes were more robust, in their case the default underrelaxation factors (0.7 for all quantities except for 0.3 for pressure) were sufficient.

With Ansys Fluent a similar approach was followed, using the second order upwind method (Fluent 2009) for pressure and momentum and the first order upwind method for the turbulence quantities in the final simulations. With second order approximations for all solution variables the simulations became unstable as with OpenFOAM. No attempt was made to stabilise the solution with lowered under relaxation factors. These partial second order solutions could only be achieved when for gradients the Green-Gauss cell based approximation was used with the polyhedral meshes and the Green-Gauss node based approximation with the tetrahedral meshes (Fluent 2009). Contrary to OpenFOAM, the Fluent simulations with polyhedral grids were more stable than the ones with tetrahedral grids. When using the least square approximation the tetrahedral meshes could only be used with first order upwind, but all polyhedral meshes with the combination of 1^{st} and 2^{nd} order approximations as described above.

4. RESULTS AND DISCUSSION

The simulation data are obtained at the measurement locations using linear interpolation between cell centre values. The results are presented graphically for selected vertical profiles (Figure 1). In addition hit rates (VDI) are calculated as validation metric for the streamwise (U_{mean}) and lateral (V_{mean}) mean velocity components. The corresponding Reynolds stress components (U_{RMS}, V_{RMS}, U'V') are compared to measurements by scatter plots as information about the experimental confidence intervals is missing for them. The Reynolds stress components of the numerical simulations were obtained from the Boussinesq hypothesis.

The profiles for the mean velocities show a very good agreement with the measured values in most cases as can be seen for a typical profile in Figure 2, where the inlet reference velocity and height given in section 3.2 are used for non-dimensionalisation. No significant difference can be observed between the two used numerical codes. Between the polyhedral and tetrahedral meshes differences occur close to the ground for the lateral velocity profile. The tetrahedral mesh results are slightly closer to the measurements but still qualitatively wrong with negative velocity components. For the streamwise velocity component the over-prediction at higher elevations is not yet fully understood. Whether the speed-up is due to blockage effects or due the well known errors in the flow prediction above the roofs by the standard k-ε model is currently investigated in a domain with height $11H_3$.

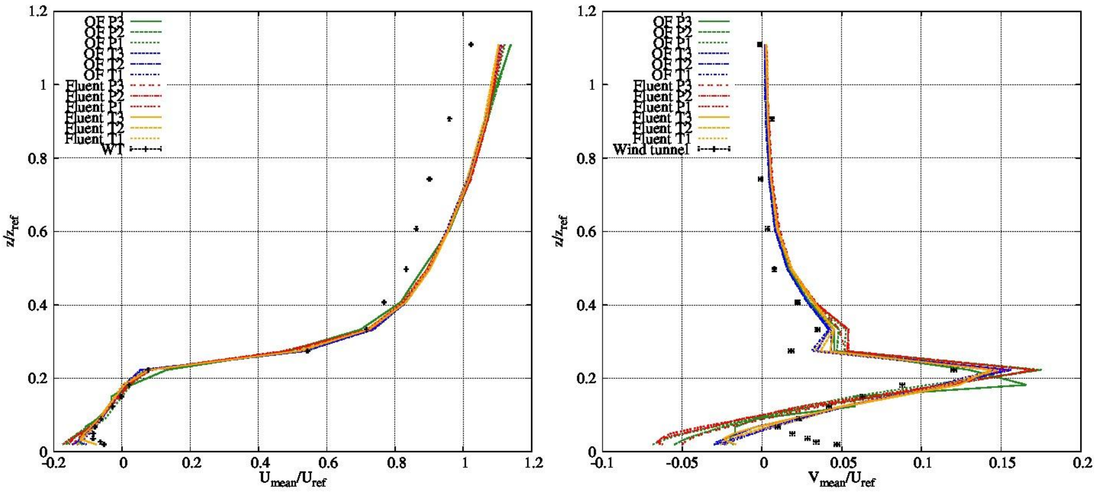

Figure 2. Streamwise and lateral velocity profiles at location 29 (see Figure 1).

Hit rates are used for the quantitative comparison between the simulations and experiments. The hit rate is defined as the ratio of hits to the total number of measurement locations (VDI 2005). A hit is obtained if the absolute or relative difference between simulation result and measurement is smaller than a given value. Here 25% is the allowed relative difference and 0.033 for U_{mean}/U_{ref} and 0.0576 for V_{mean}/U_{ref} are the allowed absolute differences (Efthimiou et al. 2011). From the hit rate results in Table 2 it can be seen that the simulations are not fully grid independent. There is no significant difference between the two numerical codes, but the tetrahedral meshes have higher hit rates, indication that the finer resolution of these yields more accurate results, especially for U_{mean}/U_{ref}. Between the medium and fine meshes the change in hit rate is however only very small. On these meshes all presented values are acceptable according to the VDI guideline (VDI 2005), which has 0.66 as lower limit for successful validation. These results are similar to the ones obtained by Efthimiou et al. (Efthimiou et al. 2011) with the commercial code StarCD and in-house code ADREA.

Hit rates	U_{mean}/U_{ref}			V_{mean}/U_{ref}		
OpenFOAM/Fluent	coarse	medium	fine	coarse	medium	fine
polyhedral	0.66/ 0.64	0.68/ 0.68	0.69/ 0.69	0.78/ 0.78	0.79/ 0.78	0.78/ 0.78
tetrahedral	0.72/ 0.69	0.76/ 0.75	0.76/ 0.75	0.82/ 0.80	0.82/ 0.82	0.83/ 0.82

Table 2. Hit rates of mean velocity components.

The turbulent quantities are compared to the experiment at all 2158 measurement positions with scatter plots of the Reynolds stress components (Figure 3). It can be clearly seen that the measured values are always under-predicted. No systematic difference can be observed for the different codes or different meshes. The under-prediction can be explained by the dying out of turbulence in the free stream due to viscous effects where no further source of turbulence exists, apart from the velocity gradients. The turbulent kinetic energy k therefore approaches the equilibrium value, which is smaller than the measured values prescribed at the inlet boundary. The higher experimental k can be attributed to the vortex generators used in the wind tunnel.

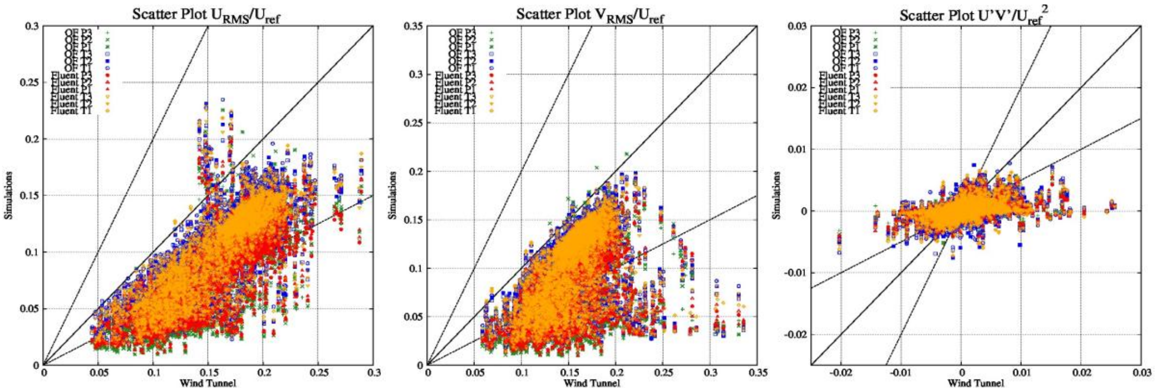

Figure 3. Scatter plots of the Reynolds stress components at all 2158 measurement positions.

5. CONCLUSIONS

Validation of OpenFOAM and Fluent against wind tunnel flow measurements in the model city Michelstadt has revealed that the mean velocities are equally well predicted by both codes. Except for the coarse polyhedral mesh all hit rates are above 0.66, with the highest values obtained on tetrahedral grids with better resolution. The Reynolds stress components are under-predicted by both codes, due to the inconsistency of the equilibrium turbulence modelling in numerical simulations and the high approach flow turbulence in boundary layer wind tunnels. How these discrepancies affect the dispersion of a passive scalar will be investigated in the future by validation against measured mean concentrations.

In addition the speed-up of the flow at high elevations will be analysed by increasing the domain height, and it will be tested whether the results on the two grid triples can be used for the quantification of the numerical discretization uncertainty.

6. REFERENCES

Efthimiou G.C., Hertwig D., Fischer R., Harms F., Bastigkeit I., Koutsourakis N., Theodoridis A., Bartzis J.G., Leitl B., 2011. Wind flow validation for individual exposure studies. Proceedings of the 13th International Conference on Wind Engineering (ICWE13), Amsterdam, The Netherlands.

Fischer R., Bastigkeit I., Leitl B., Schatzmann M., 2010. Generation of spatio-temporally high resolved datasets for the validation of LES-models simulating flow and dispersion phenomena within the lower atmospheric boundary layer. Proceedings of The Fifth International Symposium on Computational Wind Engineering (CWE2010), Chapel Hill, North Carolina, USA.

FLUENT 2009. ANSYS FLUENT 12.0 Theory Guide. Canonsburg, Pennsylvania, Ansys Inc.

Franke J., Hellsten, A., Schlunzen, K.H., Carissimo, B. 2011 The COST 732 Best Practice Guideline for CFD simulation of flows in the urban environment: a summary, International Journal of Environment and Pollution 2011 - Vol. 44, No.1/2/3/4 pp. 419 – 427 DOI: 10.1504/IJEP.2011.038443

Franke, J., Sturm, M., Siebel, C. - Validation of OpenFOAM v1.6.x with the German VDI guideline for obstacle resolving micro-scale models. In Proceedings of the 13th International Conference on Wind Engineering - ICWE13, Amsterdam, The Netherlands, 2011

OpenFOAM 2010 OpenFOAM User Guide version 1.7.1 OpenCFD Ltd.

Rakai, A., Kristof, G. - CFD Simulation of Flow over a Mock Urban Setting Using OpenFOAM. Gépészet 2010: Proceedings of the Seventh Conference on Mechanical Engineering. Budapest, Hungary, 25/05/2010-26/05/2010. Budapest: Budapest University of Technology and Economics, pp. 1-7. Paper 032. (ISBN: 978-963-313-007-0)

Schatzmann. M. ,Olesen, H., Franke, J., Hrsg. COST 732 Model Evaluation Case Studies: Approach and Results. COST office, Brussels, 2010.

VDI 2005: Environmental Meteorology – Prognostic microscale wind field models – Evaluation for flow around buildings and obstacles, VDI guideline 3783, Part 9, Beuth Verlag, Berlin, 2005