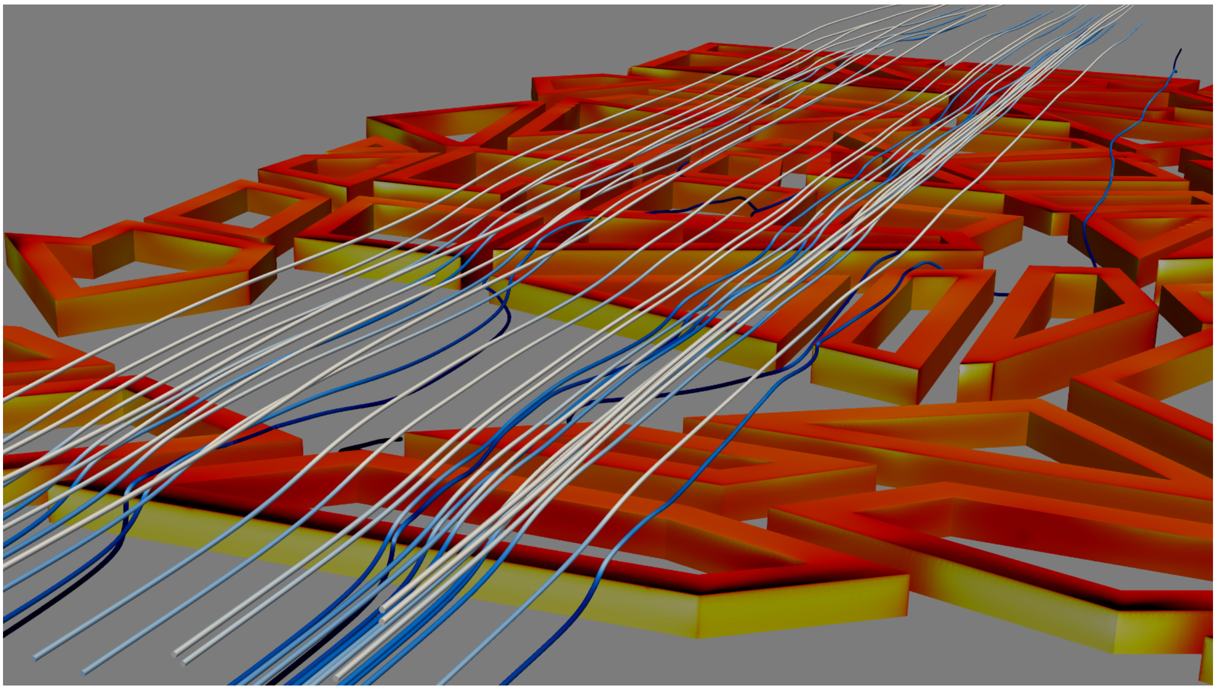
Validation of flow field prediction within a realistic but idealized urban environment with OpenFOAM

Dr.-Ing. Ulf Winkelmann (uw@kalwin-engineering.com)

Dr.-Ing. Cornelia Kalender (ck@kalwin-engineering.com)

October 01, 2022



KalWin Engineering GbR
Bergerstr. 10
58452 Witten
Germany

Validation of flow field prediction within a realistic but idealized urban environment with OpenFOAM

The accurate prediction of wind fields in the urban environment is important in many wind engineering applications such as the assessment of wind comfort, determination of ideal locations for small-scale wind turbines, or urban planning considering city ventilation and pollution dispersion. Investigations of these fields all aim to improve the life of individuals that live in cities. Wind comfort analyses are focused on determining the mechanical impact of wind on individuals in the built environment to avoid, for example, highly accelerated flows on ground level that negatively influence the subjective perception or potentially pose a risk. Contrary, accelerated flows are desired for small-scale wind turbines, often positioned on roofs, to increase the generation of green energy. The fields of city ventilation and the analysis of pollution dispersion aim to yield the best environments for individuals in cities, trying to mitigate high temperatures and air pollution.

It is clear, that the analysis of the wind field in the built environment is a subject for environmental engineers. As climate continues to change, today's challenges become even more pronounced. For instance, the cooling of overheated cities during long heat waves and droughts in the summer will increase in significance. To reduce the emission of greenhouse gases, potentially the most important aspect for humanity to fight climate change, energy production by fossil fuels needs to decrease, hence for this aim, efficient positioning of small-scale vertical wind turbine in cities is paramount.

The CEDVAL-LES database (EWTL 2022) offers a variety of wind tunnel experiments of different geometries that are specifically intended for validation of wind field predictions by CFD simulations. The most complex geometry offered at the time of writing is the case BL3-3 "Michel-Stadt, a"realistic but still idealized urban geometry" (EWTL 2022) in an atmospheric boundary layer. The geometry and results from 1838 measurement points are adopted in VDI (2017), a standard that guides developers and users to evaluate prognostic microscale wind field models. All measurement points are shown in Figure 2. In this standard, the experimental results of the wind field around the geometry "Michel-Stadt" (test case "c5") is provided for validation purposes. The geometry with a geometrical scaling factor of 1/225 inside the "WOTAN" wind tunnel of the Environmental Wind Tunnel Laboratory (EWTL) of the University of Hamburg of the is shown in Figure 1.

Figure 1: Geometry of the test case "Michel-Stadt" inside the "WOTAN" wind tunnel of EWTL of the University of Hamburg. Flow direction downwards. Figure from (EWTL 2022).

To analyze the wind field in the built environment, numerical simulations in the field of Computational Fluid Dynamics (CFD) are an excellent tool for flow predictions as these simulations can yield detailed flow information. Even in complex situations with highly separated and accelerated flows, complex wake flows, flow reattachment, and interference of buildings, CFD simulations can predict to accurate results. However, to achieve a good representation of real wind flows by numerical models, expertise to handle these complex software is required. It is important to thoughtfully chose an appropriate numerical domain, mesh, boundary conditions, turbulence model, discretization schemes, and solution procedures to obtain accurate results and to avoid and reduce errors. Disregarding any of these characteristics can be detrimental towards an accurate prediction. Besides the correct handling of the CFD software, the computational wind engineer also requires expertise and experience with computational mathematics, theoretical fluid dynamics, and the physics of boundary layer wind to achieve high quality

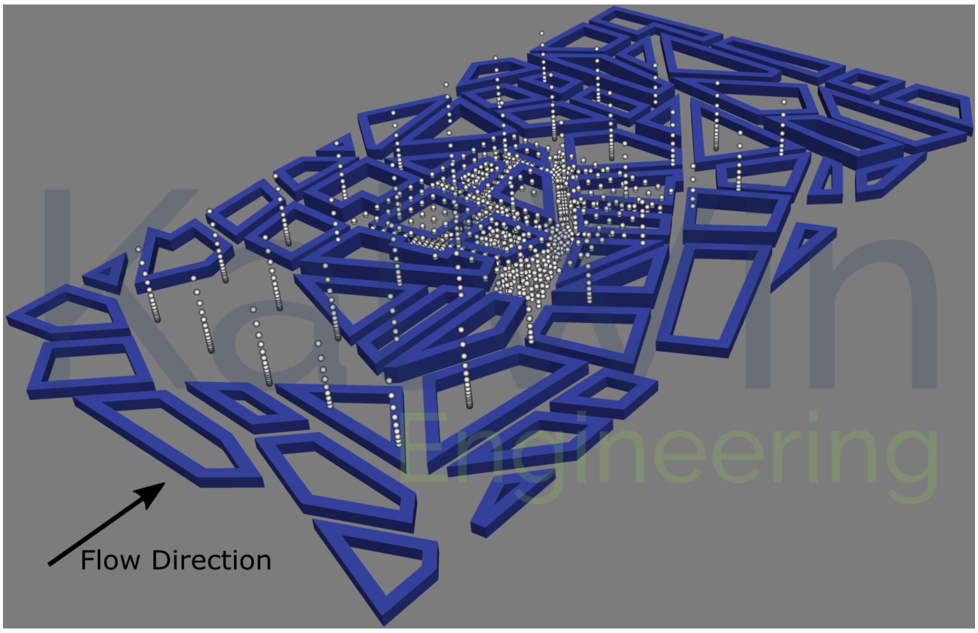


Figure 2: 1838 measurement points in "Michel-Stadt" as adopted from VDI (2017).

simulations and interpret the limitations of the modeled flow.

> **!** It is important to thoughtfully chose an appropriate numerical domain, mesh, boundary conditions, turbulence model, discretization schemes, and solution procedures to obtain accurate results and to avoid and reduce errors. Disregarding any of these characteristics can be detrimental for an accurate numerical prediction.

To ensure the accuracy of the numerical simulations, validation is important. Validation methods quantify the difference of the applied mathematical model to reality AIAA (1998). In other words, validation answers the question if the CFD simulation is "solving the right equations" (Roache 1998). Validation, in simple terms and most frequently conducted for wind engineering applications, is the comparison of CFD results to wind tunnel experiments. In this regard, VDI (2017) proposes the use of the hit rate q to assess CFD results for the prediction of wind flows at locations:

$$q = \frac{n}{N} = \frac{1}{N} \cdot \sum_{i=1}^{N} n_i.$$

Here, n/N is the ratio of correctly predicted measurement points to the total number of measurement points N. A correct prediction at one measurement point n_i is achieved, when either the relative difference between prediction by CFD P_i to wind tunnel measurement O_i or the absolute difference is small:

$$n_i = \begin{cases} 1 \text{ if } \left|\frac{P_i - O_i}{O_i}\right| \leq D \text{ or } |P_i - O_i| \leq W \\ 0 \text{ otherwise} \end{cases} .$$

The thresholds that define the allowed deviations between CFD and wind tunnel are D that describes a relative or "permitted" (VDI 2017) difference , and W that accounts for the absolute allowed difference. The values for D and W are related to the reproducibility of the wind tunnel measurement and besides other factors, repeatability and limits of measurement equipment, respectively (VDI 2017) . For the case of "Michel-Stadt" (test case c5 in (VDI 2017)), the following values are recommended: $D = 0.25$, $W = 0.08$, and $q = 0.66$ for one component of the velocity vector and $q = 0.5$ for the combined analysis of longitudinal and lateral components (u and v).

For all defined 1838 measurement locations in (VDI 2017), the hit rates q for the normalized longitudinal velocity component u/u_{ref} (parallel to the ABL inflow), the normalized lateral velocity component v/u_{ref}, and a combination of both are shown in Table 1. Here, u_{ref} is the longitudinal velocity component at the normalization point according to (VDI 2017). Additionally, the allowed thresholds from (VDI 2017) are given in the last row. Clearly, the results of the CFD simulations are all well below the minimum required hit rates. It should be noted that the wind tunnel data from (VDI 2017) comprises velocity components with the value of zero. In these situations, only the absolute criterion of the hit rate is applied. If the absolute criterion is not met, the relative criterion is discarded and a non-hit is noted for this particular measurement point.

Table 1: Hit rate for all 1838 measurement locations from (VDI 2017), for the normalized longitudinal velocity component u/u_{ref}, the normalized lateral velocity component v/u_{ref}, and a combination of both.

	u/u_{ref}	v/u_{ref}	u/u_{ref} and v/u_{ref}
q [%] CFD	77.6	82.8	67.7
q [%] VDI Threshold	66.0	66.0	50.0

Figure 3 shows scatter plots of the normalized longitudinal and lateral velocity components in a) and b), respectively. This visual comparison of the predicted CFD results and wind tunnel data for all measurement points shows that most predictions are well within the allowed thresholds. From the plots of the longitudinal and lateral velocity components in Figure 3 a) and b), respectively, some trends are observable:

- highest longitudinal velocities with significant acceleration $u/u_{ref} > 1$ are underestimated by CFD.
- average positive velocity velocities $0.1 < u/u_{ref} < 0.5$ are mostly underestimated by CFD.
- lowest longitudinal velocities (with a negative sign) $u/u_{ref} < -0.25$ are mostly overestimated by CFD.
- similar trends for the lateral velocity are not observable.

It remains to be analyzed to understand regions around the bluff bodies of the simplified buildings where the aforementioned trends appear.

Figure 4 depicts all measurement points where the numerical prediction is outside the hit rate criteria. While few measurement points are highlighted in courtyards and the open place on the bottom left of Figure 4 a), most mispredictions appear in the street canyons and the place in the inner part of the “Michel-Stadt” (see Figure 4 b)) where most measurement points are located as depicted in Figure 2.

Figure 5 shows the qualitative distribution of the magnitude of the velocity field at a height of 1.75/225 m. Depicting the figure it becomes apparent that may regions in the idealized city part show rather low velocities as indicated by the dark colors.

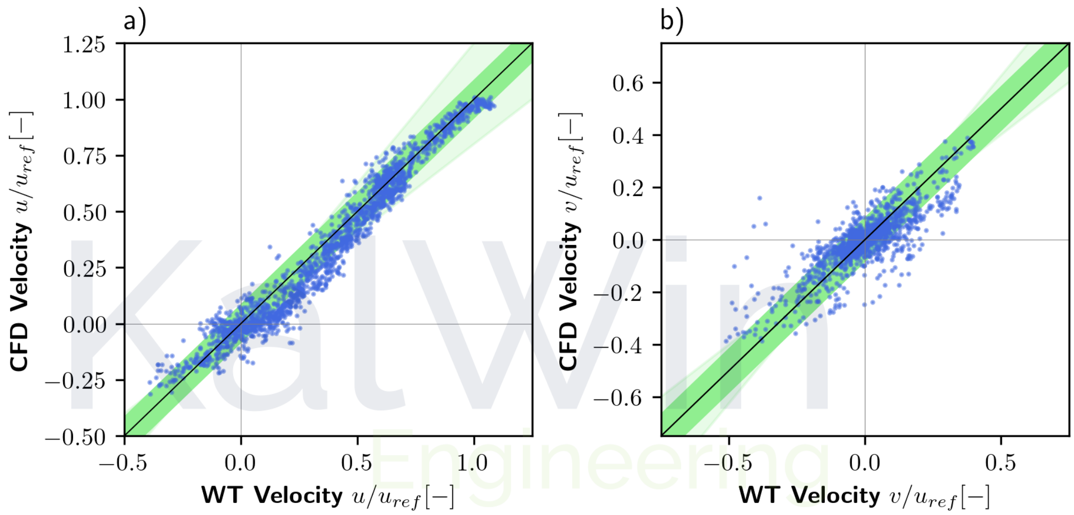


Figure 3: Scatter plots of normalized longitudinal velocity component in a) and normalized lateral velocity component in b). Vertical axes relate to the CFD results; horizontal axes relate to the wind tunnel results. Black diagonal line represents a perfect match between CFD and wind tunnel. The green region represents the absolute allowed difference; the light green region represents the allowed relative difference

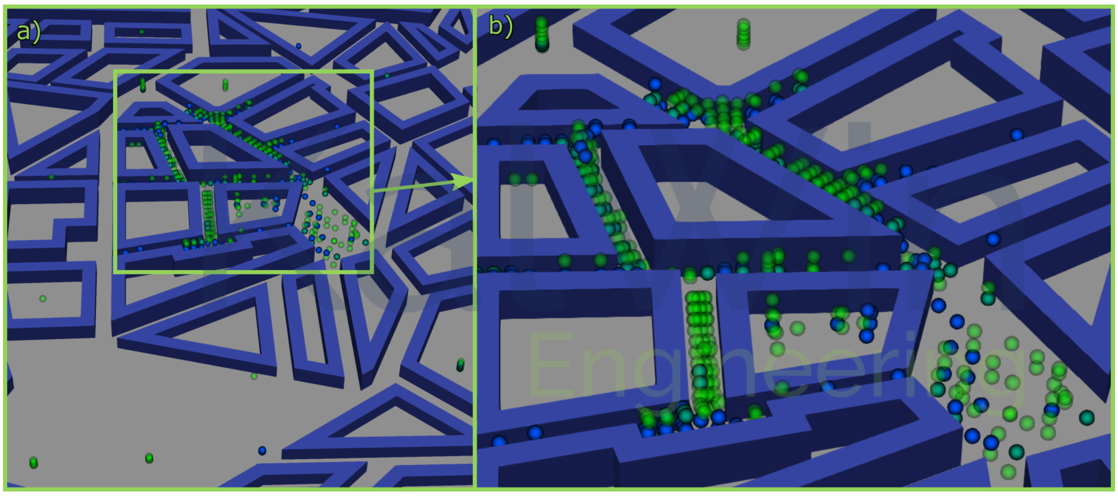


Figure 4: Visualization of all measurement points where the numerical prediction is outside the hit rate criteria. Non-hits for the lateral velocity component are visualized in green; non-hits for the longitudinal velocity component are visualized in blue. a) Shows all non-hits; b) is a close up of the missed predictions in the center of the geometry. Main flow direction upwards

However, in certain street canyons, public squares, and within blocks of houses, significant flow acceleration is observable.

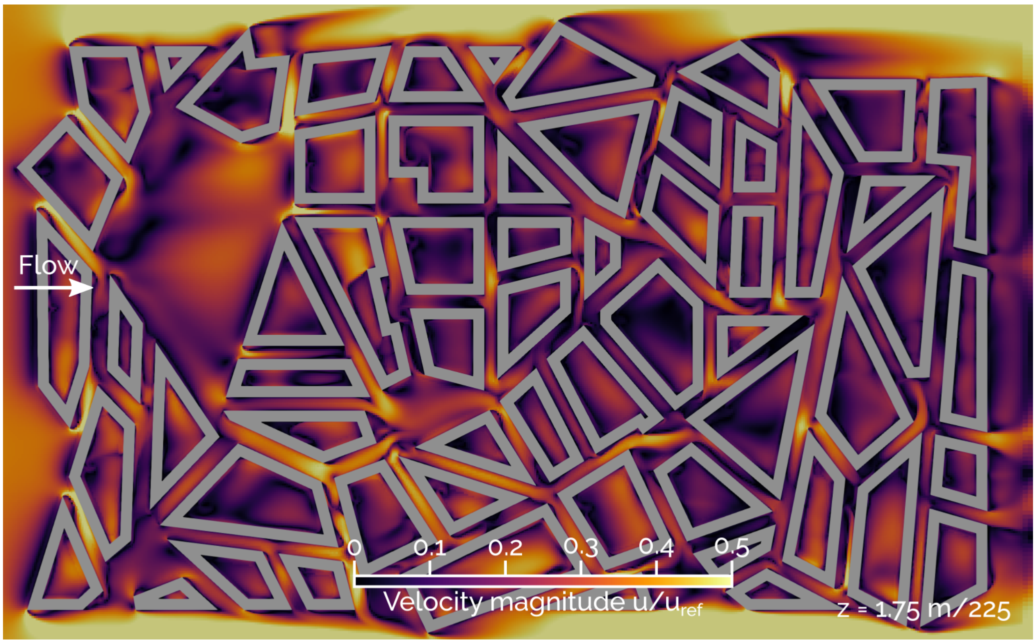


Figure 5: Birds eye view of the velocity field at a height of 1.75/225 m

The report shows that numerical CFD simulations conducted with OpenFOAM predict satisfactory results within the permitted deviations for the calculation of wind flow in a realistic but idealized urban environment when compared to wind tunnel results. The applied hit rate validation metric for longitudinal and lateral velocity components are well within the proposed thresholds in VDI (2017). Based on the presented results, the wind field prediction with OpenFOAM is suitable for further investigations such as wind comfort analyses and utilization in efficient positioning of small-scale VAWT in the urban environment. Here, even more emphasis should be put on the spatial discretization in the areas of interest, i.e., the first 1.75 m above ground (B. Blocken, Janssen, and Hooff 2012) and on the roofs, respectively. Such a mesh refinement could potentially further increase the accuracy of the simulations.

Details of numerical simulations

RANS Simulations are conducted using OpenFOAM (Weller and Tabor 1998) v.2112 with the realizable $k{-}\epsilon$ turbulence model (Shih et al. 1995) and the SIMPLEC solution algorithm (Van Doormaal and Raithby 1984) with a Finite Volume Method (Versteeg and Malalasekera 2007) discretization, employing the second order upwind scheme for the convective term of the momentum equations. The simulations resemble the published wind tunnel experiment of the EWTL of the University of Hamburg (EWTL 2022) with a geometrical scaling factor of 1/225. The simulations are conducted under the assumption of incompressibility.

The distance between geometry and the numerical domain's boundaries are $5\,H$ upstream, $15\,H$ downstream, and $5\,H$ to the top, where H is the maximum height of the buildings. The distance between the geometry and the front and back of the domain is enlarged so that the directional blockage ratio is less than 17% (Bert Blocken 2015).

The atmospheric boundary layer the the inlet of the numerical domain is based on the wind tunnel experiment and described by the log-law with a roughness length $z_0 = 1.53/225\,m$. Inlet conditions of turbulent kinetic energy k and ϵ are based on the proposal of (Richards and Hoxey 1993). The outlet is modeled as a constant pressure outlet. Walls are modeled as smooth walls. The top, front, and back of the domain feature a symmetry boundary condition.

The simulation is assumed to be statistically converged after all flow variables remain constant with increased iterations. The steady state solution at all 1838 measurement point for the two components of the velocity vector are shown in Figure 1 a) and b). Here, constant velocity components are observed after approximately 2000 iterations as highlighted in grey region.

The final mesh after careful mesh sensitivity study, comprises 8 836 111 cells is shown in Figure 2. Here, the smallest cells are located in the vicinity of the geometry,

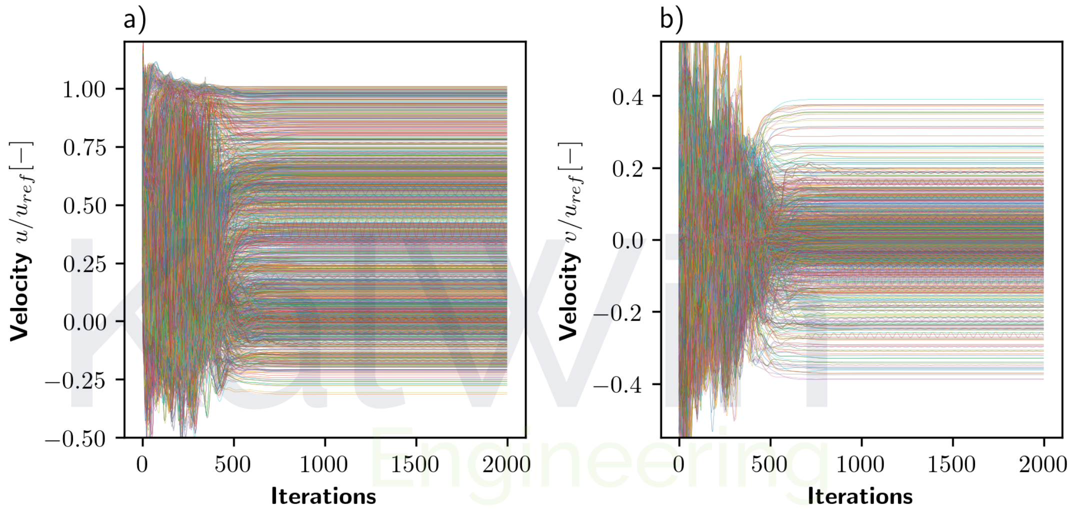


Figure 1: Normalized longitudinal and lateral velocity components u/u_{ref} and v/u_{ref} in a) and b), respectively, over 2000 iterations highlighting the converged steady state results at the end of the simulation. The figure shows all 1838 measurement points from (VDI 2017).

especially on the edges of the buildings. Several fields in the shape of cuboids surround the geometry.

To understand the influence of the mesh and obtain an accurate but also economically viable refinement of the spatial discretization, a mesh sensitivity study has been conducted. A significantly finer mesh with 36 004 524 did not lead to a relevant difference of the flow prediction with regards to the purpose of this analysis. The comparison for the prediction of velocity components between the reference mesh as described above and the fine mesh is shown in Figure 3. For the vast majority of analyzed measurement locations, the difference between the numerical prediction on the different meshes is well below $\Delta = \pm 0.1$ as highlighted by the green area.

Additionally, a coarser mesh consisting of 2 610 571 has been analyzed. Here, the discrepancy to the reference mesh was larger especially for the lateral velocity component u/u_{ref} than the differences shown in Figure 3 (not shown here). Hence, the reference mesh has been deemed appropriate.

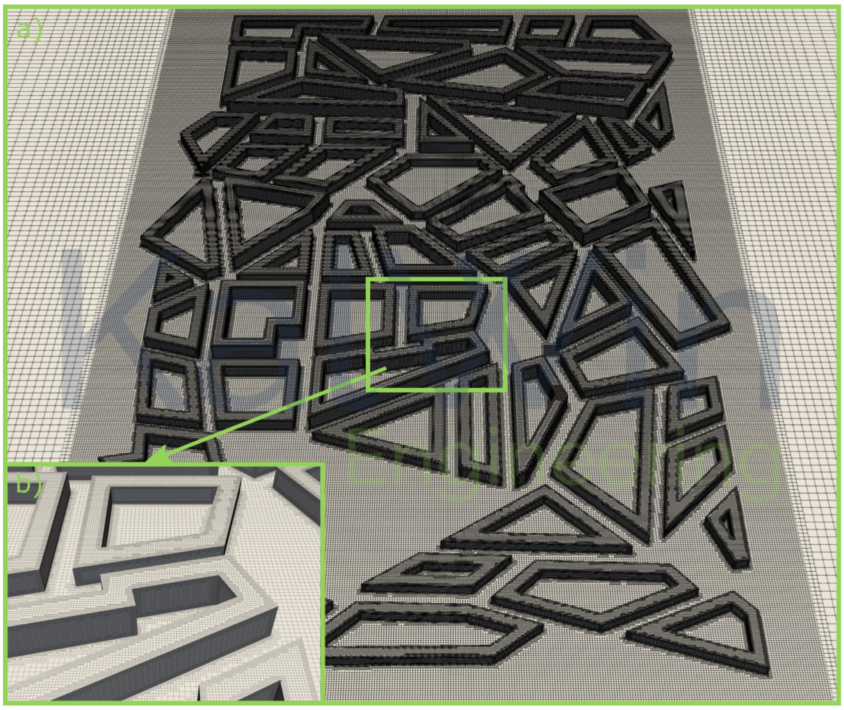


Figure 2: Surface mesh in the vicinity of the geometry with a close-up in b). Wind direction is upwards.

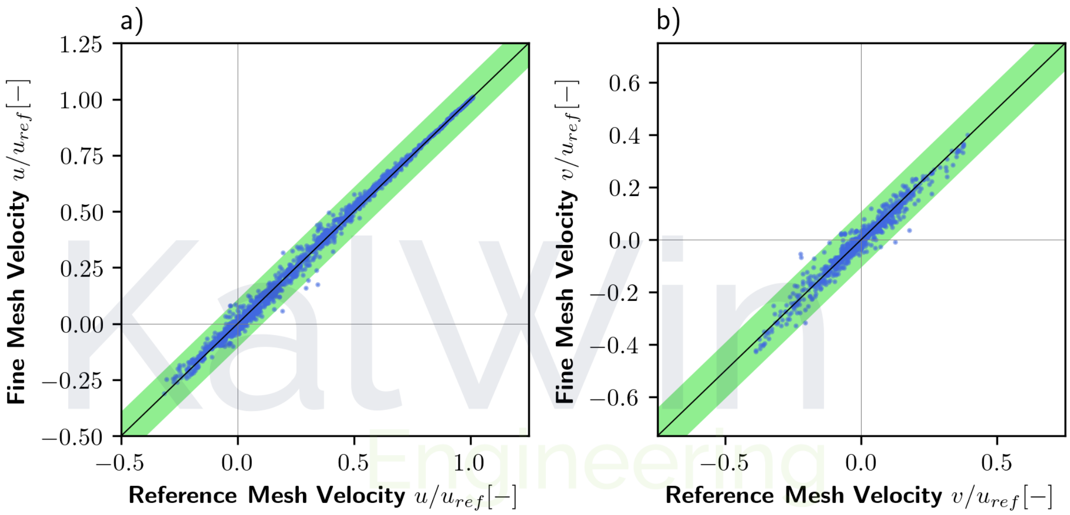


Figure 3: Surface mesh in the vicinity of the geometry with a close-ups in b). Wind direction is upwards.

References

AIAA. 1998. "Guide for the Verification and Validation of Computational Fluid Dynamics Simulations (AIAA G-077-1998(2002))." In. Washington, DC. https://doi.org/10.2514/4.472855.001.

Blocken, Bert. 2015. "Computational Fluid Dynamics for Urban Physics: Importance, Scales, Possibilities, Limitations and Ten Tips and Tricks Towards Accurate and Reliable Simulations." *Building and Environment* 91: 219–45. https://doi.org/10.1016/j.buildenv.2015.02.015.

Blocken, B., W. D. Janssen, and T. van Hooff. 2012. "CFD Simulation for Pedestrian Wind Comfort and Wind Safety in Urban Areas: General Decision Framework and Case Study for the Eindhoven University Campus." *Environmental Modelling & Software* 30 (April): 15–34. https://doi.org/10.1016/j.envsoft.2011.11.009.

EWTL. 2022. "Compilation of Experimental Data for Validation of Microscale Dispersion Models (CEDVAL-LES)." http://www.mi.uni-hamburg.de/cedval-les Last accessed: 23.08.2022.

Richards, P. J, and R. P Hoxey. 1993. "Appropriate Boundary Conditions for Computational Wind Engineering Models Using the k- Turbulence Model." *Journal of Wind Engineering and Industrial Aerodynamics* 46-47: 145153. https://doi.org/10.1016/0167-6105(93)90124-7.

Roache, Patrick J. 1998. *Verification and Validation in Computational Science and Engineering*. Albuquerque, N.M: Hermosa Publishers.

Shih, Tsan-Hsing, William W. Liou, Aamir Shabbir, Zhigang Yang, and Jiang Zhu. 1995. "A New k-Epsilon Eddy Viscosity Model for High Reynolds Number Turbulent Flows." *Computers & Fluids* 24 (3): 227–38. https://doi.org/10.1016/0045-7930(94)00032-T.

Van Doormaal, J. P., and G. D. Raithby. 1984. "ENHANCEMENTS OF THE SIMPLE METHOD FOR PREDICTING INCOMPRESSIBLE FLUID FLOWS." *Numerical Heat Transfer* 7 (2): 147–63. https://doi.org/10.1080/01495728408961817.

VDI. 2017. "VDI 3783 Blatt 9: VDI 3793 Blatt 9 - Environmental Meteorology - Prognostic Microscale Wind Field Models - Evaluation for Flow Around Buildings and Obstacles." https://www.vdi.de/richtlinien/programme/inhalte-zu-richtlinien/vdi-3783-blatt-9.

Versteeg, H. K., and W. Malalasekera. 2007. *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*. 2nd ed. Harlow; England; New York: Pearson Education Ltd.

Weller, H G, and G Tabor. 1998. "A Tensorial Approach to Computational Continuum Mechanics Using Object-Oriented Techniques." *Computers in Physics* 12 (6): 620–31. https://doi.org/10.1063/1.168744.