August 27-31, 2023
FLORENCE - ITALY
ICWE 16
16th International Conference on Wind Engineering

Development of wind tunnel and numerical simulations for the flow around a circular cylinder over a large range of Reynolds numbers

Sophie Breitkopf[1] and Christian Hartz[1]

[1]*Chair of Structural Design, TU Dortmund University, Dortmund, Germany,* Email address

SUMMARY:
For the design of high-rise buildings or other slender structures wind effects such as wind loading and dynamic behaviour become relevant for the design. Wind tunnel or numerical simulations are the two methods currently available to model the behaviour. For the occurring very high Reynolds numbers both methods are limited and the models are usually scaled down. While this is acceptable for sharp edged shapes the prediction of the flow behaviour for rounded shapes still poses a challenge. As numerical simulations improve with increasingly available computational power they might close this gap in the future. In order to judge both developments an extensive literature review for the basic circular cylinder is undertaken. For wind tunnel and numerical simulations a large range of results is documented for the critical transition of the flow, where the behaviour is highly dependent on the surface roughness. To go beyond this range the lack of existing reference data makes a validation even more complex. This paper focusses on the comparison of wind tunnel and numerical simulations for the flow around the circular cylinder in the scientific research community.

Keywords: high-rise buildings, CFD, slender structure, circular cylinder, surface roughness, accuracy

1. INTRODUCTION

For safe and sustainable slender structures it is important to accurately predict the wind loads and the dynamic behaviour in turbulent flow. While constructions with sharp edges can be scaled, the flow around high-rise buildings with rounded corners, chimneys or wind-turbine towers is highly dependent on the Reynolds number. As very high Reynolds numbers (up to 10^7 and 10^8) can occur, the state of the art wind tunnel approach with scaled models reaches a limit for rounded building-shapes. So far advancing Computational Fluid Dynamics (CFD) applications have also been limited to smaller Reynolds numbers as the required computational power increases with higher numbers. However, with gradually increasing computational power, the boundaries and accuracy of CFD are developing with technological advances. The questions arise to which extend the accuracy increases and what data to use for validation.

To judge the accuracy of models, it is essential to ensure a correct depiction of the physical behaviour. Due to its dependence on the turbulence intensity, mixing length, dimension of the object, surface roughness and the Reynolds number, the flow around rounded shapes is complex. The basic circular cylinder has been studied intensively throughout different disciplines and methods for around a century creating a large data base of published results. This makes the circular cylinder particularly suitable for the overview regarding the development of wind tunnel tests and CFD simulations over time and Reynolds number. The current research also includes CFD simulations to further study the limits and influencing factors of the simulations throughout the different Reynolds numbers.

2. METHODOLOGY

Existing literature on the circular cylinder from the beginning of wind tunnel testing until now is summarized. Comparable values such as the drag and lift coefficient, the Strouhal number and the pressure coefficients over the surface are compiled as a reference. In OpenFOAM the air flow ($\rho = 1$ kg/m^3 and $\nu = 1.5 \cdot 10^{-5}$ m^2/s) around a circular building cross-section of 30 m diameter is simulated for gradually increasing velocities up to 50 m/s (Mach number <0.2, incompressible, Re=10^8). For the URANS (unsteady Reynolds averaged Navier Stokes) simulations the k-ε and k-ω SST turbulence models are applied. A domain size of 600 m (20 d) towards the inlet, top and bottom and 1800 m (60 d) towards the outlet is modelled with a block mesh for which mesh independence is ensured. The cylinder surface is smooth, the turbulence intensity 5% and the mixing length / diameter ratio 10%. The simulation time is set to ensure that the fluid flows through the domain at least 3 times and the Courant number is below <1.

3. RESULTS

In figure 1 some of the results for the mean drag coefficient are displayed over the Reynolds number. The critical transition range is zoomed in the lower subfigure. Recently published results from literature are displayed with an increasing visual impact: wind tunnel simulations (grey), CFD simulations (blue), full-scale measurements (purple), DIN EN 1991-1-4, 2010 (black). When the surface roughness is stated and altered the results are grouped into the categories of the DIN EN 1991-1-4, 2010 and indicated with the equivalent line. Own simulation results according to section 2 are marked in red.

It can be seen that CFD simulations develop towards higher Reynolds numbers and can reproduce the drop in the drag coefficient around the critical flow transition to a certain extend. While wind tunnel and numerical simulations align rather well for the Reynolds numbers up to the Kármán vortex street, the range of results in the transition phase, with the high dependency on the boundary conditions, is large. Published results often miss precise information regarding those influencing factors, which is essential for validation. A lack of reliable data for very high Reynolds numbers becomes evident. The simulations of this study confirm the high accuracy for Reynolds numbers towards the subcritical range and the need for further studies for the transition phase.

4. CONCLUSION

Until the transition range of the flow around a circular cylinder wind tunnel experiments and numerical simulations mostly align. In order to judge the accuracy in the transition range the research has to be extended regarding an evaluation of specific conditions for published results and own studies. Even though numerical simulation progress to higher Reynolds numbers, significantly more studies are required for the transcritical Reynolds numbers in order to be able to validate the results. One approach is the bluff body benchmark by Breitkopf et al., 2022 that targets this range.

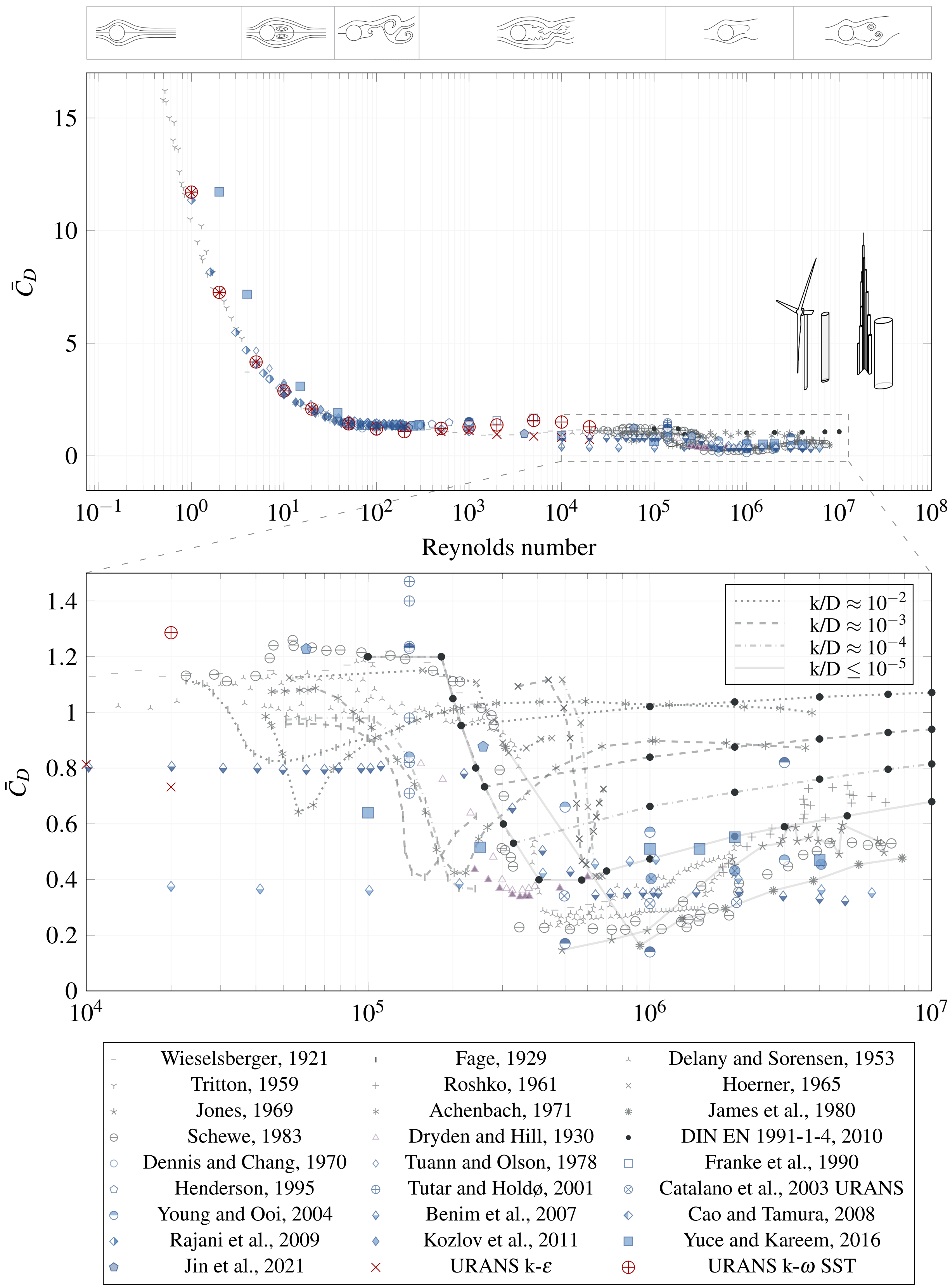

Figure 1. drag coefficient for circular cylinder over different Reynolds number obtained from CFD simulations or wind tunnel experiments between 1920 and today (flow characterisation at the top according to Schlichting and Gersten, 2006)

REFERENCES

Achenbach, E., 1971. Influence of surface roughness on the cross-flow around a circular cylinder. Journal of fluid mechanics 46, 321–335.

Benim, A., Cagan, M., Nahavandi, A., and Pasqualotto, E., 2007. RANS predictions of turbulent flow past a circular cylinder over the critical regime. Proceedings of Proc. 5th IASME/WSEAS International Conference on Fluid Mechanics and Aerodynamics, Athens, Greece, 232–237.

Breitkopf, S., Sander, A., and Hartz, C., 2022. Development of Bluff Body Benchmark at high Reynolds numbers. Proceedings of 18th EAWE PhD Seminar on Wind Energy. EAWE. Bruegge, 307–310.

Cao, S. and Tamura, Y., 2008. Flow around a circular cylinder in linear shear flows at subcritical Reynolds number. Journal of wind engineering and industrial aerodynamics 96, 1961–1973.

Catalano, P., Wang, M., Iaccarino, G., and Moin, P., 2003. Numerical simulation of the flow around a circular cylinder at high Reynolds numbers. International journal of heat and fluid flow 24, 463–469.

Delany, N. K. and Sorensen, N. E., 1953. *Low-speed drag of cylinders of various shapes*. Tech. rep.

Dennis, S. and Chang, G.-Z., 1970. Numerical solutions for steady flow past a circular cylinder at Reynolds numbers up to 100. Journal of Fluid Mechanics 42, 471–489.

DIN EN 1991-1-4, Dec. 2010. *Eurocode 1: Actions on strucutres - Part 1-4: General actions - wind actions*. Norm.

Dryden, H. L. and Hill, G. C., 1930. Wind pressure on circular cylinders and chimneys. Bureau of Standards Journal of Research 5, 653–693.

Fage, A., 1929. The effects of turbulence and surface roughness on the drag of a circular cylinder. Rep, Memo. 1.

Franke, R., Rodi, W., and Schönung, B., 1990. Numerical calculation of laminar vortex-shedding flow past cylinders. Journal of Wind Engineering and Industrial Aerodynamics 35, 237–257.

Henderson, R. D., 1995. Details of the drag curve near the onset of vortex shedding. Physics of Fluids 7, 2102–2104.

Hoerner, S. F., 1965. Fluid-Dynamic Drag. Theoretical, experimental and statistical information. Copyright by: SF Hoerner Fluid Dynamics, Vancouver, Printed in the USA, Card Number 64-19666.

James, W., Paris, S., and Malcolm, G., 1980. Study of viscous crossflow effects on circular cylinders at high Reynolds numbers. Aiaa Journal 18, 1066–1072.

Jin, D., Wu, Z., and Choi, H., 2021. A predictive model of the drag coefficient of a circular cylinder. Physics of Fluids 33, 111702.

Jones, G. W., 1969. Aerodynamic forces on a stationary and oscillating circular cylinder at high Reynolds numbers. Vol. 300. National Aeronautics and Space Administration.

Kozlov, I. M., Dobrego, K. V., and Gnesdilov, N. N., 2011. Application of RES methods for computation of hydrodynamic flows by an example of a 2D flow past a circular cylinder for Re= 5–200. International Journal of Heat and Mass Transfer 54, 887–893.

Rajani, B., Kandasamy, A., and Majumdar, S., 2009. Numerical simulation of laminar flow past a circular cylinder. Applied Mathematical Modelling 33, 1228–1247.

Roshko, A., 1961. Experiments on the flow past a circular cylinder at very high Reynolds number. Journal of fluid mechanics 10, 345–356.

Schewe, G., 1983. On the force fluctuations acting on a circular cylinder in crossflow from subcritical up to transcritical Reynolds numbers. Journal of fluid mechanics 133, 265–285.

Schlichting, H. and Gersten, K., 2006. Grenzschicht-Theorie. Springer-Verlag.

Tritton, D. J., 1959. Experiments on the flow past a circular cylinder at low Reynolds numbers. Journal of Fluid Mechanics 6, 547–567.

Tuann, S.-y. and Olson, M. D., 1978. Numerical studies of the flow around a circular cylinder by a finite element method. Computers & Fluids 6, 219–240.

Tutar, M. and Holdø, A., 2001. Computational modelling of flow around a circular cylinder in sub-critical flow regime with various turbulence models. International journal for numerical methods in fluids 35, 763–784.

Wieselsberger, C. v., 1921. Neuere feststellungen uber die gesetze des flussigkeits und luftwiderstands. Phys. Z. 22, 321.

Young, M. and Ooi, A., 2004. Turbulence models and boundary conditions for bluff body flow. Proceedings of Proc. 15th Australian Fluid Mechanics Conf.

Yuce, M. I. and Kareem, D. A., 2016. A numerical analysis of fluid flow around circular and square cylinders. Journal-American Water Works Association 108, E546–E554.