Vortex dynamics in the wake of wall-mounted cylinders: experiment and simulation

Thomas Uffinger¹, Stefan Becker², Irfan Ali³

1: Institute of Process Machinery and Systems Engineering, University of Erlangen, Germany, uf@ipat.uni-erlangen.de
2: Institute of Process Machinery and Systems Engineering, University of Erlangen, Germany, sb@ipat.uni-erlangen.de
3: Institute of Fluid Mechanics, University of Erlangen, Germany, irfan@lstm.uni-erlangen.de

Keywords: wake, LDA, numerical simulation, CFD

Investigations of the flow field around wall-mounted cylinders in a cross flow as shown in Figure 1 can help to better understand bluff body aerodynamics. Furthermore, cylinder-like structures appear in many technical applications such as antennas and attaching parts of vehicles, high-rise buildings or supports in internal and external flows.

In literature much attention has been paid to the topic. Nevertheless there are remaining questions. This work will focus on the main flow structures depending on the geometry of the cylinder and the unsteady behavior of the flow.

In the present paper the three-dimensional flow fields around three different wall-mounted cylinder stump geometries in a cross-flow are evaluated. The basic configuration is a square cylinder with a length \( L \) of 120 mm and an edge-length \( D \) of its cross area of 20 mm. Besides this basic configuration a second one with an elliptical afterbody mounted behind the cylinder and a third one with a wedge in front of the cylinder are investigated. With a free stream velocity \( U_∞ \) of 10 m/s a Reynolds number \( Re \) of about 12,500 is achieved.

A combined approach which uses experimental as well as numerical investigations is applied to provide an extensive data base. For the experimental investigations laser-Doppler anemometry is used to determine the average velocity field and the RMS-values of the velocities. Besides average values the frequency spectra of the flow velocities based on the measured data are analyzed. Concerning computational fluid dynamics three different simulation techniques are applied. A shear stress transport (SST) turbulence model represents RANS approaches. A hybrid simulation technique combining Reynolds averaged Navier-Stokes (RANS) and large eddy simulation (LES) features is realized by the scale adaptive simulation (SAS) model. The third type of numerical simulation is a LES that is able to capture the unsteady flow field in the entire computation domain.

One major finding of the measurements is that the geometry of the cylinder can significantly vary the influence of the roof vortex system on the average flow field in the wake. For the configuration with the elliptical afterbody the roof vortex reattaches on the topside of the cylinder and the flow field behind the cylinder barely interacts with the roof vortex system resulting in an undisturbed Kármán vortex street evolving behind the cylinder. In case of the basic square cylinder the influence of the roof vortices is stronger and the Kármán vortex street cannot develop along the full height of the cylinder (see Figure 1). For the configuration with the wedge in front a three-dimensional flow field in the wake is achieved.

Furthermore the unsteady behavior of the flow is of special interest. Besides the shedding frequency of the Kármán vortex street a much lower frequency of about 3 Hz can be detected in the wake (see Figure 2) which can be explained as follows: the region behind the cylinder is filled with fluid with a large amount of momentum over time. After some time the momentum in the wake is large enough and the fluid is washed out of the wake, which in consequence produces higher values of the unsteady velocity. After a while the velocity values drop back to a lower level and the wake is filled with fluid of high momentum again.

The numerical results confirm the above findings concerning the unsteady behavior of the flow in the wake of the cylinder. As in the experiments a low frequency of around 3 Hz can be detected besides the shedding frequency of the Kármán vortex street.

Another aim of the present investigations is the comparison of different numerical approaches to compute the flow field behind wall-mounted cylindrical geometries. The average velocity field of all three numerical approaches fits the experimental results satisfactorily, although LES-based velocity fields show the best results. Analyzing turbulence quantities LES gives the best consistency with experimental results. Therefore, the higher computational effort of LES compared to the RANS based approaches is justified in case of wall mounted cylinder geometries as better results can be achieved.

4.1.4.