Determination of the separation points for the flow around blunt bodies: experimental and numerical studies

N O Tănase¹*, Ş M Simionescu¹, M M Păduroiu¹ and C Bălan¹

¹REOROM Group, Department of Hydraulics, Hydraulic Machinery and Environmental Engineering, Faculty of Power Engineering, University Politehnica of Bucharest, Romania.
	octavia.tanase@upb.ro

Abstract. The paper is concerned with experimental studies and numerical modelling of the air flow around blunt bodies (cylinder, aerodynamic profile) inside a wind tunnel. The range of the investigated Reynolds numbers is 12000<Re<35000. The main goal of the work is to establish a CFD procedure to analyse the flow pattern in the vicinity of the bodies and to predict the position of the boundary layer detachment. The computations are performed with several turbulence models implemented in the ANSYS Fluent code. The direct visualizations and the numerical results are corroborated, with the aim to investigate the positions of the separation points and the flow spectrum developed downstream the bodies.

1. Introduction

The flow over circular cylindrical structures have been a benchmark problem in the field of wind engineering. Previous studies have mainly focused on the Reynolds number effect of the drag coefficients of blunt bodies, especially of the circular cylinders [1÷8].

Yao J. et al. [1] have studied the characteristic parameters of mean wind pressure coefficient distribution on cylindrical surface, drag coefficient and lift coefficient for different Reynolds numbers and turbulence intensities.

A comparative study based on the 3D computational simulations of the flow around a circular cylinder fitted with vortex generators, benefiting from a series of dedicated model tests conducted in a large circulating water tunnel was investigated by Ünal and Gören [2]. Significant drag reduction was observed in both experimental and computational study due to the application of the vortex generators.

Zhang [3] analysed the turbulent flow past a finite circular cylinder (AR=1) is simulated at a relatively large Reynolds number (Re = 20000). The focus is to examine the influence of different turbulence models on several aspects of the flow: the mean drag coefficient, the mean surface-pressure coefficient, the time averaged velocity and pressure fields, and the mean bed-shear-stress amplification.

Ravi et al. [4] investigated numerically the viscous incompressible flow around NACA 4412 subsonic airfoil using FLUENT code at a free stream Reynolds number of 3 million. The analysis is carried out with Spalart Allmaras and $k - \omega$ SST turbulence models with transition capabilities. Lift and drag coefficients obtained with CFD analyses are compared with the wind tunnel test data available in open literatures. It is concluded that $k - \omega$ SST turbulence model with transition capabilities gives close prediction of lift and drag coefficients both in pre stall and post stall region.
The flow was investigated both numerically with large eddy simulation and experimentally with hot-wire anemometry and particle image velocimetry by Parnaudeau et al. [5]. The numerical simulation was performed using high-order schemes and a specific immersed boundary method. The present study focuses on turbulence statistics and power spectra in the near wake up to ten diameters.

The main goal of the paper is to investigate the flow around blunt bodies positioned in a wind tunnel. The study is based on the corroboration between experiments and numerical simulations performed with the ANSYS Fluent code. The three turbulence models: standard $k - \varepsilon$, $k - \omega$ SST and Detached Eddy Simulation (DES) with SST $k - \omega$ RANS model were used.

The investigations are concerned with the following aspects:
(i) the location of the separation points on bodies;
(ii) the flow spectrum of the wake downstream the bodies;
(iii) the distributions of the wall shear stress on the cylinder wall and the velocity magnitude on some lines of interest downstream the cylinder (see paragraph 2.3).

2. Experimental study of the flow around blunt bodies: cylinder and aerodynamic profile

2.1. Wind tunnel
The experiments are performed in an open circuit wind tunnel, with an axial fan with adjustable speed at the downstream end.

The tunnel (figure 1) consists of:
• inlet section (square section, 90 × 60 cm) with honeycomb for reducing the turbulence and perturbations in the flow;
• two transparent test sections with a length of 50 cm, height 30 cm and width 10 cm, where various bodies can be mounted and the velocities can be measured. These visualization and measurement areas are provided with windows for direct viewing of the flow. In the test chamber the flow is considered uniform. The bodies used were placed in the second viewing area. Measurements of the velocity distribution were made in the first viewing section and upstream of the bodies.
• output section (circular, with the diameter of 65 cm), where the axial fan is mounted. The characteristics of the fan are: flowrate 11900 m³/h, electric motor power 750 W, maximum 1360 rot/min.

Figure 1. The open circuit wind tunnel used for experiments
2.2. Experimental procedure

The experiment was performed by injecting fume with a smoke machine. The speed of the fan was set at three different values. For each fan speed a set of direct visualizations of the flow spectrum downstream the bodies was realized. The smoke was made visible in a plane generated by a continuous laser with 532 nm wavelength. The development of the studied phenomena was captured with a slow-motion camera, focused on the plane made visible by the laser beam.

The two blunt bodies used for tests in the wind tunnel are a cylinder with a diameter of 62 mm and a span of 100 mm (equal to the width of the test section) and an S814 airfoil with a chord length of 180 mm and a span of 100 mm.

The following measuring devices were used in this experiment:
- a stroboscope of type HELIO STROB for measuring the speed of the axial fan,
- a hot wire anemometer of type AN500, produced by Extech for measuring the local air velocities in the test sections.

Some examples of direct flow visualizations acquired during the experiments are presented in figures 2 and 3.

![Figure 2](image2.png)

**Figure 2.** Flow spectrum around the cylinder for flows at $Re = 9800$ (a), $Re = 10700$ (b) and $Re = 16800$ (c).

![Figure 3](image3.png)

**Figure 3.** Flow spectrum around S814 airfoil for flows at $Re = 31200$, angle of incidence: a) 0°, b) 10°, c) 20° and d) 30°.
2.3. Experimental results

The velocities were measured in the first transparent section on the line at a distance of $x = 1.9$ m from the entrance section and in the second section, downstream of the bodies, on three lines using the devices described above. The three lines (L1, L2, L3) are positioned at distances equal to 5 cm downstream the bodies and each other.

The velocities downstream the two bodies were measured for the first speed of the fan, in the case of the cylinder, and for the second speed of the fan in the case of the profile. On these lines, several points were considered at a distance of 3 cm to top and bottom walls, starting from 9 to 39 cm on the tunnel height, in which the velocities were measured. The velocity distributions on lines L1, L2 and L3 are shown in figure 4. The first graph shows a drop of the velocity in the centre of the distribution, due to the presence of the cylinder, whereas the turbulence downstream the airfoil is highlighted on the second graph by the irregular shape of the velocity distributions downstream the airfoil.

![Velocity distribution](image)

**Figure 4.** Velocity distribution on different lines downstream the cylinder for $n_1 = 173$ rot/min and downstream the S814 airfoil for $n_2 = 185$ rot/min.

The characteristic Reynolds number of the flow is calculated as

$$Re = \frac{\rho v L}{\eta}$$

where $\rho = 1.204$ kg/m$^3$ is the air density, $\eta = 1.825 \times 10^{-5}$ Pa·s is dynamic viscosity at $T = 20$ °C and $L$ is the characteristic length: in the case of the cylinder $L = D = 62$ mm - the diameter of the cylinder, and in the case of the profile $L = c = 180$ mm - the chord length of the airfoil.

The Reynolds number for the flow around the cylinder was calculated for three different fan speeds. The mean velocity in the first transparent section and the characteristic Reynolds numbers are presented in table 1.

| $n$ [rot/min] | 173 | 185 | 260 |
|---------------|-----|-----|-----|
| $v$ [m/s]     | 2.4 | 2.63| 4.12|
| $Re$          | 9800| 10700| 16800|

The Reynolds number for the flow around the S814 airfoil at the speed of the fan $n_2 = 185$ rot/min is $Re = 31200$.

The angles where the separation points occur ($\theta_{D_1}$ and $\theta_{D_2}$) were measured from the center of the leading edge of the cylinder, see figure 5. The flow begins to separate from the solid surface at different angles depending on the Reynolds number – see table 2.
Table 2. Experimental values of the angles at which the detachment points appear for the cylinder

| Re  | $\theta_{D1}$ | $\theta_{D2}$ |
|-----|---------------|---------------|
| 9800| 105°          | 254.4°        |
| 10700| 103.6°        | 256°          |
| 16800| 102°          | 259°          |

3. Numerical study

3.1. Flow geometry and boundary conditions

The 3D numerical simulations were performed with unsteady solver and turbulent models implemented in the numerical code ANSYS Fluent, [6÷8]. The geometry and mesh are presented in figure 7. The computational mesh consisted of hexahedral finite volumes, using 2 million cells.

The boundary conditions for the numerical simulations are: inlet: average velocity $v_0 = 0.21$ m/s; outlet: constant atmospheric pressure, $p = p_0$, walls and obstacle: stationary solid, the fluid adheres to the solid surface $v = 0$, the other lines that appear in the geometry have been built to achieve a finer discretization: inside, it does not influence the flow.

3.2. Numerical results

The turbulent models used for 3D numerical simulations are standard $k - \varepsilon$, $k - \omega$ SST and Detached Eddy Simulation (DES) with SST $k - \omega$ RANS model.
Figure 7. Comparison between pathlines distributions for the three turbulent models: a) standard $k - \varepsilon$, $k - \omega$ SST, c) DES SST $k - \omega$ and d) experimental flow spectrum.

In figure 7 the results for the computed flow spectrums are presented, in comparison with the direct visualizations. The wake structure downstream the cylinder – in particular, the size and length of this wake – is well reproduced qualitatively by the DES SST $k - \omega$ model.

Figure 8. Distribution of wall shear stress on the extrados of the cylinder for: a) $k - \varepsilon$ Standard, b) $k - \omega$ SST and c) DES SST $k - \omega$. 

$k - \omega$ SST and c) DES SST $k - \omega$. 

$k - \omega$ SST and c) DES SST $k - \omega$. 

$k - \omega$ SST and c) DES SST $k - \omega$. 

$k - \omega$ SST and c) DES SST $k - \omega$.
The numerical solutions give an insight of all flow quantities, one of the most important being the wall shear stress (WSS) distribution on the cylinder, see figure 8. By plotting the components of WSS as function of the $\theta^\circ$ - angle, one can determine the location of critical points. The values of the angle at which the detachment point $D_1$ appear for the cylinder obtained numerically and experimentally are presented in table 3.

A quantitative confirmation that the solution DES SST $k - \omega$ is the most indicated for the analysis is obtained by the comparison of the measured and computed velocity profiles downstream the cylinder, figure 9. The velocity profiles correspond to three lines (normal to the main flow direction) positioned at distances equal to 5 cm downstream the cylinder and from each other.

The qualitative and quantitative comparisons between the experimental and the numerical results recommend the DES SST $k - \omega$ model as the most indicated for performing the modelling and analysis of the flow around bodies positioned in a wind tunnel.

### Table 3. Comparison between numerical and experimental values of the angles at which the detachment points appear for the cylinder

| Turbulence models | $k - \varepsilon$ standard | $k - \omega$ SST | DES SST $k - \omega$ |
|-------------------|---------------------------|------------------|----------------------|
| $\theta_{D_1}$    | 136°                      | 136°             | 107°                 |
| Experimental      |                            |                  | 105°                 |

**Figure 9.** Velocity distribution on different lines downstream the cylinder - DES SST $k - \omega$ model vs. experiment.

### 4. Conclusions

The main objective of the research was the investigation of the flow around blunt bodies (cylinder and S814 airfoil) positioned in a wind tunnel. The study was based on the comparison between experimental and the corresponding numerical results. The three turbulence models: standard $k - \varepsilon$, $k - \omega$ SST and Detached Eddy Simulation (DES) with SST $k - \omega$ RANS model were used.

**Figure 10.** Comparison between the flow spectrum experimental and numerical around the profile S814.
The validation of the numerical results was performed starting from the experimental data measured. A qualitative comparison of the flow spectra showed a good prediction for the numerical results (see also figure 10), but the most important was the quantitative validation, presented in paragraph 3.2. Following the validation, the CFD studies were extended and additional relevant quantities were determined: flow pathlines around the studied bodies and distributions of the wall shear stress. From the three turbulence models used for the simulations, the best results were obtained with Detached Eddy Simulation (DES) model.

The present research has treated the evolution of the air flow around two different types of blunt bodies, involving different kinematic parameters. The knowledge of the various phenomena and the understanding of the operating mechanisms gained from this study can be directly transposed into higher complexity industrial applications.

Acknowledgments
The work has been funded by the Operational Programme Human Capital of the Ministry of European Funds through the Financial Agreement 51675/09.07.2019, SMIS code 125125.

References
[1] Yao J, Lou W, Shen G, Guo Y and Xing Y 2019 Appl. Sci. 9.
[2] Ünal U O and Gören Ö 2011 Eng. Appl. of Comput. Fluid Mech. 5 1.
[3] Zhang D 2017 Conf. Series: J. of Physics 910.
[4] Ravi H C, Madhukeshwara N and Kumarappa S 2013 I. J. of Innov. Res. in Science, Eng. And Techn. 2 7.
[5] Philippe Parnaudeau Ph, Johan Carlier J, Dominique Heitz D and Eric Lamballais E 2008 Physics of Fluids 20.
[6] Bailly C and Comte-Bellot G 2015 Turbulence (Springer New York and London).
[7] Swirydezk J 1990 A visualization study of the interaction of a free vortex with the wake behind an airfoil (Experiments in Fluids) (Springer Verlag).
[8] ***ANSYS Fluent Theory Guide, ANSYS Inc 2013.