INNOVATIVE MECHANICAL ENGINEERING **ISSN 2812-9229 (Online)**
University of Niš, Faculty of Mechanical Engineering
VOL. 3, NO 3, 2024, PP. 9-19

Original scientific paper *

CFD INVESTIGATION OF LAMINAR AND TURBULENT FLOW AROUND A SMOOTH SPHERE AND AN NFL BALL

Petar Miljković[1], Veljko Begović[1], Miloš Jovanović[1] and Branka Radovanović[1]

[1] University of Niš, Faculty of Mechanical Engineering, Serbia

Abstract. *This paper presents a detailed analysis of the flow around a scaled-down National Football League (NFL) ball submerged in water, with special attention given to the velocity field in the wake region behind the ball. The simulation was performed in a channel with a rectangular cross-section. Laminar and turbulent flows were developed, where the Reynolds number (Re) ranged from 1,300 to 50,000. An unstructured tetrahedral mesh with refined regions was used to accurately represent the flow characteristics around the ball, particularly in high-velocity zones. The CFX module of the ANSYS software was used to obtain numerical results and the velocity field. The reduced model of the NFL ball was made in SolidWorks. The simulation results were validated by comparing the flow around a sphere (ping-pong ball) with widely recognized experimental and numerical data from the literature. The results reveal the formation of distinct flow patterns and vortex structures in both laminar and turbulent regimes, with notable differences in the wake region as the Reynolds number increases.*

Key words: *CFD, Numerical simulations, ANSYS CFX, Flow, NFL ball*

1 Introduction

The flow around bodies in fluids is a crucial aspect in understanding fluid dynamics and engineering applications involving the flow of air or liquids around solid objects. Whether it is the aerodynamic design of vehicles, the assessment of ship resistance, or the analysis of blood flow through vessels, studying the interaction between fluids and solid bodies plays a vital role in various disciplines.

An NFL ball, with its specific elongated shape, represents a challenging object for flow analysis. Its unique geometry and possibility of variable angles of incidence make the flow patterns around it different from those around more conventional shapes such as spheres or cylinders. Understanding the aerodynamic characteristics of an NFL ball is not only important for improving sports performance but can also contribute to the broader field of aerodynamics.

***Received: October 22, 20224 / Accepted December 27, 2024.**
Corresponding author: Petar Miljković
University of Niš, Faculty of Mechanical Engineering, Serbia
E-mail: petar.miljkovic@masfak.ni.ac.rs

© 2024 by Faculty of Mechanical Engineering University of Niš, Serbia

Historically, researchers have developed numerous methods and models for predicting and analyzing flow. From basic experiments in wind tunnels [1,2] to advanced numerical simulations using methods such as finite volumes and finite elements [3,4], scientists strive to better understand how different factors, such as body shape, fluid velocity, and fluid properties, influence flow and the forces acting on the body.

Flow around a cylinder is a phenomenon that is often investigated in the field of aerodynamics and fluid dynamics, where it refers to the situation when the fluid flow around the cylinder becomes unstable, which can cause the appearance of various effects such as eddy currents, turbulence and the like.

In 1851, George Gabriel Stokes studied the effects of fluid viscosity on the damping of a pendulum's oscillations. Stokes analyzed how the internal friction of the fluid (viscosity) slows down the movement of the pendulum, leading to key conclusions about the drag force exerted by the fluid on objects moving through it [5]. This research led Stokes to formulate principles for deriving equations of resistance for a sphere moving through a viscous fluid, which later became known as Stokes' law.

In 1883, Osborne Reynolds published a paper [6] which made a fundamental contribution to the field of fluid mechanics by introducing the concept we now know as the Reynolds number. This became a key parameter in fluid mechanics, enabling the prediction of flow regimes in various systems.

Today, much attention is paid to the study of flow around a body, especially in the context of aerodynamics and hydrodynamics. The development of advanced simulation techniques, such as computational fluid dynamics (CFD), enables a detailed understanding of how fluids, such as air or water, flow around objects of various shapes and sizes. These studies are essential for a variety of applications, including vehicle design, the development of wind-resistant buildings, and in biomedical research where they analyze how fluids flow around various anatomical structures. In addition, experimental investigations in aerodynamic and hydrodynamics testing tunnels enable verification of theoretical models and improvement of facility design to reduce drag and increase efficiency.

Bakić [7] presented experimental results for the flow around a sphere, including a smooth sphere in low inlet turbulence, a sphere with a trip wire, and a sphere in high free-stream turbulence, at subcritical Reynolds numbers. Mean velocity fields and turbulence quantities were obtained using laser-Doppler anemometry, and a comparison of velocity fields and turbulence characteristics for different flow configurations was provided.

In paper [8], the effect of free stream turbulence, wind speed and ship motion on the airwake frequency spectra of two 1:50 scaled ships with stern flight-decks was investigated using wind tunnel testing.

Watson et al. [9] described an investigation in which piloted flight simulation was used to study the effect of turbulent air flow on helicopter recovery to an offshore platform. A helicopter flight simulation environment was developed in which the unsteady air flow over a full-scale offshore platform was modeled using time-accurate Computational Fluid Dynamics.

Le and Hong [10] utilized numerical computation to investigate the hydrodynamic characteristics of a torpedo-shaped underwater glider. The physical model of this glider was developed using Myring profile equations and analyzed through a computational fluid dynamics approach.

Constantinescu and Squires [11] compared LES and DES predictions, focusing on important flow characteristics such as the drag coefficient, wake frequencies, the position of laminar separation, and the distribution of pressure and skin-friction along the sphere.

In this study, the focus is on the analysis of the aerodynamic flow around an NFL ball. The goal is to gain a deeper understanding of the flow mechanisms around an NFL ball and to identify the key parameters that influence its aerodynamic performance. The authors' main motivation for writing this paper was to analyze the flow around an NFL ball in water. Previously, the research on the topic of the flow around various bodies was conducted in air. In paper [12], an experimental and numerical investigation of the flow around an NFL ball in a wind tunnel was carried out.

Through research like this, scientists and engineers are constantly advancing our understanding of fluid dynamics, leading to innovations in various industries.

2. Methodology

The numerical simulation was conducted using the ANSYS software package, which provides advanced tools for modeling and analyzing fluid flow phenomena [13]. Specifically, the CFX module, an integral part of the software suite, was employed for Computational Fluid Dynamics (CFD) simulations due to its robust capabilities in handling complex fluid flow problems. The CFX module is particularly well-suited for simulating turbulent flows and offers advanced turbulence modeling options, making it ideal for the present study.

The development of advanced software such as ANSYS, particularly its CFX module, has significantly advanced the ability to solve the Reynolds-Averaged Navier-Stokes (RANS) equations that describe fluid flow. The RANS equations are a form of the Navier-Stokes equations where the instantaneous quantities are decomposed into mean and fluctuating components. Solving these equations requires discretizing the partial differential equations that govern fluid flow, which is accomplished using a finite volume discretization method in ANSYS CFX.

2.1 Model

Two cases were considered: the flow around a ping pong ball and a scaled-down model of an NFL ball. The objects being studied were placed inside a rectangular channel of sufficient length to allow the development of the selected flow profiles.

For the first part of the analysis, the geometry of a ping pong ball was considered. The model was generated using the actual dimensions of the ball, with a diameter of d = 40 mm and a perfectly smooth surface. This was done to validate the numerical results, which will be explained in detail below.

For the second part of the analysis, the geometry of an NFL ball was considered. An NFL ball has precisely defined dimensions according to the standards of the National Football League (NFL) [14]. For the purposes of the simulation, a model was created at 23.65% scale of a real NFL ball. The SolidWorks software was used to create both models. Table 1 presents the characteristic dimensions of the NFL ball and the model used in the simulations.

The rectangular channel used in the simulation was specifically designed to provide sufficient space for both objects and to facilitate the development of accurate flow profiles.

Its cross-sectional dimensions were scaled relative to the characteristic diameter *d* of the objects, with a height of *3d*, a width of *2d*, and a length of *50d*. These proportions ensured that the flow around the objects could be fully developed and that any boundary effects from the walls of the channel were minimized.

Table 1 Characteristic dimensions of the NFL ball

	Dimensions of the ball	Dimensions of the model
Length [mm]	280	66.23
Circumference (longitudinal) [mm]	700	164.4
Circumference (lateral) [mm]	530	125.66

To avoid difficulties with mesh generation and subsequent simulation results, it was necessary to adapt the model by removing unnecessary details on the ball itself. This approach created conditions that realistically represent the flow around an NFL ball, which is important for obtaining accurate results.

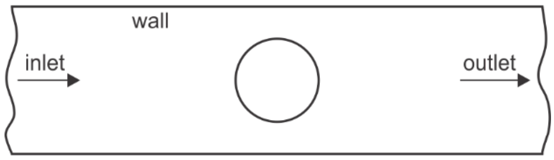


Fig. 1 Model and boundary conditions for flow around a sphere

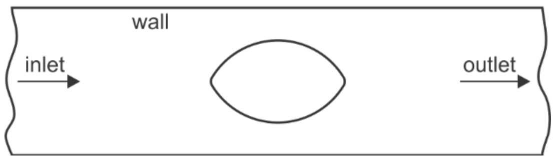


Fig. 2 Model and boundary conditions for flow around an NFL ball

Depending on the Reynolds numbers, which ranged from 1,300 to 50,000, different models were used. For Reynolds numbers less than 2,000, a laminar flow model was employed. For higher Reynolds numbers, the k-ω turbulence model was selected due to its robustness and wide acceptance in engineering applications. This model calculates the turbulent kinetic energy (k) and the specific rate of its dissipation (ω), which are essential for determining the turbulent viscosity and closing the RANS equations. The k-ω model is particularly effective for simulating high Reynolds number flows where turbulence plays a significant role.

The decision to employ the k-ω turbulence model stemmed from a comprehensive evaluation of alternative models to ensure accurate and efficient simulation of the flow conditions under consideration. While the k-ε model is commonly applied in industrial and free-shear flow scenarios, it was deemed unsuitable for this study due to its limitations in resolving near-wall phenomena, which are critical for capturing the intricate flow patterns around the NFL ball and within the channel. Similarly, the k-ω *SST* model, known for its enhanced accuracy in boundary layer predictions, introduced additional complexity without offering significantly better results compared to the standard k-ω model for the studied flow regime. The selected k-ω model provided the best balance between accuracy and computational efficiency, demonstrating excellent agreement with experimental data and reliable predictions of turbulence characteristics across the range of analyzed Reynolds numbers.

Boundary conditions were carefully defined to accurately represent the physical scenario. The inlet boundary condition was set as a uniform velocity profile corresponding to the desired Reynolds numbers to simulate different flow regimes. The outlet boundary condition was specified as an opening boundary condition with zero relative static pressure to allow for the smooth exit of fluid from the computational domain. No-slip wall conditions were applied to the surface of the NFL ball and the walls of the channel to accurately simulate the viscous effects. The convergence criteria for continuity, momentum, and transport (model) equations were always less than 10^{-7}. The boundary conditions used in the numerical simulations are represented in Table 2.

Table 2 Boundary conditions

Boundary	Condition
Inlet	Velocity (Depending on the Re number)
Outlet	Opening boundary condition
Walls	No slip

2.2 Meshing

The meshing process is crucial for the accuracy and precision of the simulation. A fine mesh was created around critical areas of the NFL ball, especially at the front and rear, where complex flow patterns and vortex structures were expected to form. Furthermore, an adaptive mesh was used to ensure high resolution in areas with large pressure and velocity gradients.

For the simulation, a tetrahedral mesh with an inflation layer and zones with a higher number of elements was used. This choice of mesh was made due to the complex geometry and the need for more efficient problem-solving, especially in the boundary layer. Tetrahedral control volumes allow easier adaptation to complex shapes, particularly for an NFL ball. They are especially useful for domains with complex geometric details where maintaining flexibility in cell distribution is necessary. Initially, there were some issues with the mesh quality, which could have affected the accuracy of the simulation results. To achieve a certain level of accuracy and to ensure the stability of the simulation, cell sizes had to be appropriately adjusted. Setting the maximum cell size to 10 mm and the minimum cell size in zones of special importance to 0.5 mm was used to ensure the stability of the simulation. The mesh details for both models are provided in Fig. 3 for the sphere and Fig. 4 for the NFL ball.

As a result of these considerations, the mesh for the simulation of flow around the sphere consisted of 2,932,317 elements. For the NFL ball, due to its more complex geometry and intricate flow patterns, an even finer mesh with 3,522,120 elements was utilized. The increased number of elements in the mesh for the NFL ball was necessary to precisely capture detailed flow characteristics, especially in areas with high pressure and velocity gradients.

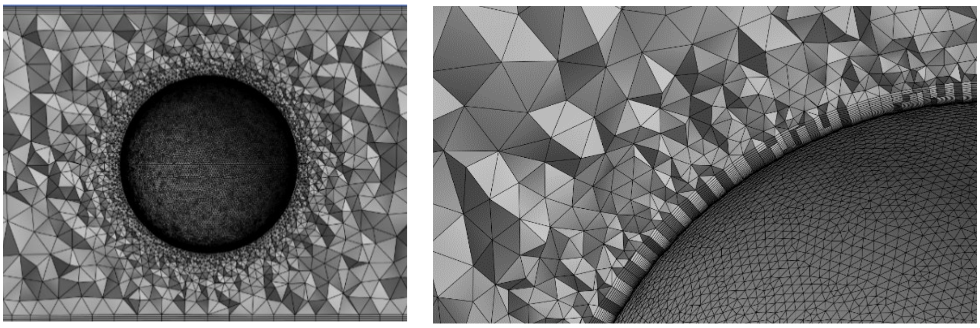


Fig. 3 Mesh for a sphere (ping-pong ball)

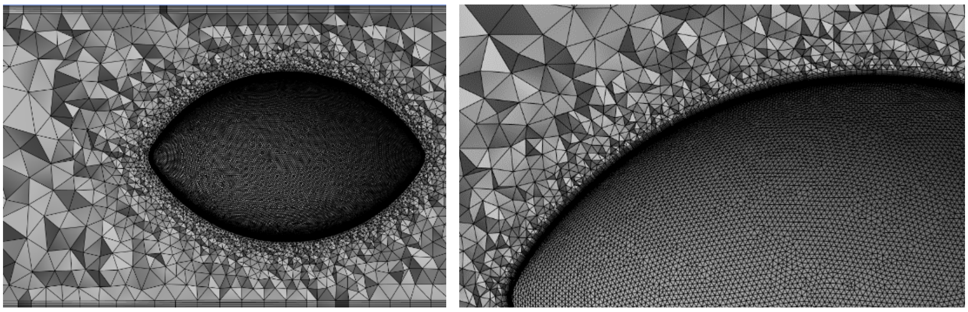


Fig. 4 Mesh for an NFL ball

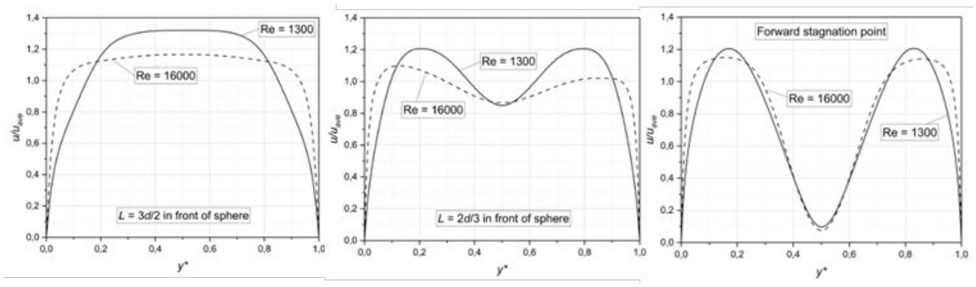
A detailed mesh analysis was conducted, including parameters such as orthogonality, skewness, and aspect ratio. These parameters indicated that the mesh quality was satisfactory, leading to minimized numerical error and ensuring simulation stability. This mesh generation allowed for:

1. Accurate analysis of turbulent flow in the boundary layer, which is crucial for understanding different types of flow around an NFL ball as well as other geometric bodies.
2. Efficient use of computational resources through mesh optimization, enabling the numerical simulation to be performed within an acceptable time frame.

2.3 Validation

The study presented in [15] was used to validate the results obtained from the numerical simulation. In this study, the authors presented both experimental and numerical results of the flow around a sphere (ping-pong ball), providing detailed data on velocity profiles. Their observation on the flow around a sphere serves as a benchmark for validating computational fluid dynamics (CFD) models of spherical objects.

By comparing the numerical results with the experimental and numerical data from paper [15], the accuracy and reliability of the simulation approach were assessed. The close agreement between the obtained results and those presented in [15] confirmed that the modeling techniques and computational settings were appropriately chosen. This validation step was crucial to ensure that the simulation of the flow around the NFL ball yielded trustworthy and meaningful results.



a) b) c)

Fig. 5 Results of the flow around a sphere

Fig. 5 shows the numerical results of the flow around a sphere. These diagrams are almost identical to the diagrams presented in [15]. The diagrams display the relationship u/u_{ave} (dimensionless velocity, where u is the instantaneous velocity and u_{ave} is the average velocity) versus y^* (dimensionless transverse coordinate). In diagram (a), the velocity distribution is shown for a cross-section located at a distance of $L=3d/2$ (where d is the diameter of the ball experiencing flow around it, $d = 40$ mm); in diagram (b), the distribution is shown at a distance of $L=2d/3$; and finally, diagram (c) represents a cross-section that coincides with the forward stagnation point. The cross-sections were selected to correspond to those in [15].

For the numerical simulations of the flow around the NFL ball, the same velocity distribution was ensured in the undisturbed part of the stream in front of the object being studied.

3. RESULTS

The results obtained from the numerical simulations of flow around the NFL ball are presented and analyzed in this section. These results include detailed visualizations of the velocity fields and vortex trails for the laminar flow case at a Reynolds number of Re=1,300, as well as for two turbulent flow cases at Reynolds numbers of Re=16,000 and Re=50,000. By examining these different flow regimes, we aim to understand how the flow characteristics around the NFL ball evolve with increasing Reynolds numbers.

Fig. 6 shows the velocity field of laminar flow at a Reynolds number of Re=1,300.

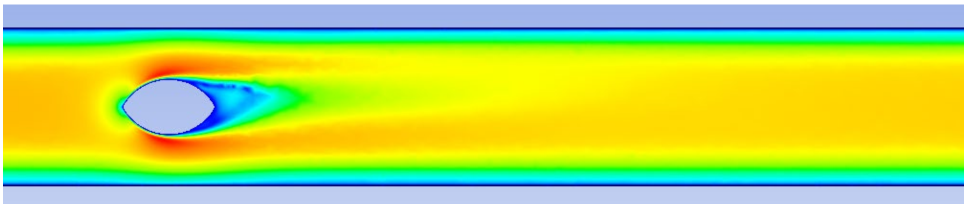


Fig 6. Velocity field for laminar flow (Re=1,300)

The stagnation point on the front side of the ball can be clearly observed from Fig. 6. Furthermore, the velocity field shows a regular pattern where, at a distance of *11d* from the body, the velocity profile re-establishes to one identical to that in front of the body. Fig. 7 presents the vortex wake on the rear side of the ball, where the recirculation zones are clearly visible.

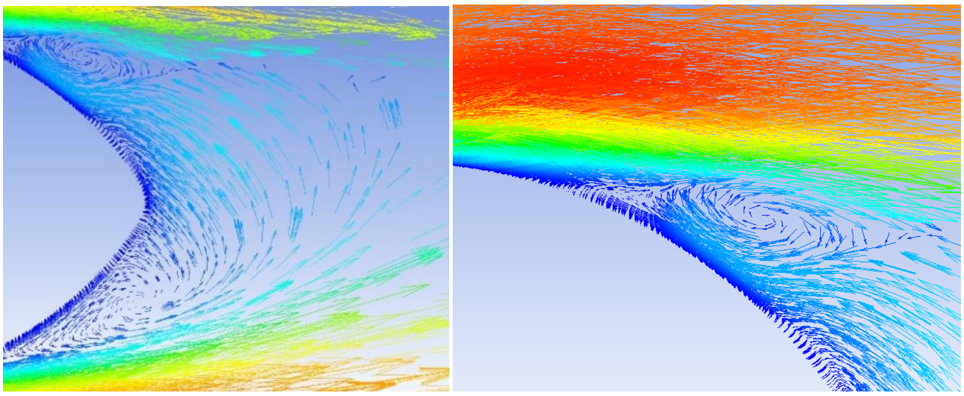


Fig 7. Vortex flow on the rear side of the ball (Re=1,300)

Fig. 8 shows the velocity field of turbulent flow at a Reynolds number of Re=16,000.

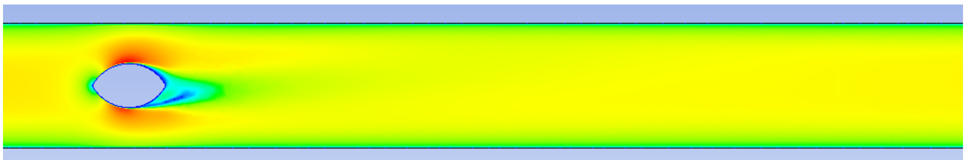


Fig 8. Velocity field for turbulent flow (Re=16,000)

The stagnation point on the front side of the ball can once again be clearly observed. Additionally, the flow disturbance in the region behind the ball is significantly smaller compared to the laminar flow case (Re=1,300). However, the velocity field in the undisturbed flow region re-establishes at a distance of approximately *16d*, which is greater compared to the laminar flow. Fig. 9 presents the vortex wake on the rear side of the ball

(Re=16,000). Unlike the previous laminar model, we can now observe that the recirculation zones are significantly less pronounced and are mainly localized in the lower part of the ball.

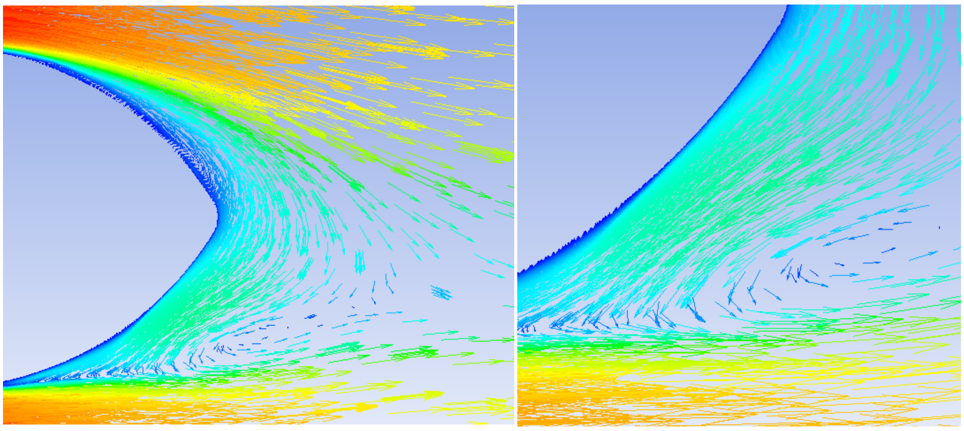


Fig 9. Vortex flow on the rear side of the ball (Re=16,000)

Fig. 10 shows the velocity field of turbulent flow at a Reynolds number of Re=50,000.

Fig 10. Velocity field for turbulent flow (Re=50,000)

The stagnation point on the front side of the ball can once again be clearly observed. Additionally, it can be seen that the flow disturbance in the region behind the ball is significantly smaller compared to both the laminar flow case (Re=1,300) and the turbulent flow case (Re=16,000). However, the re-establishment of the velocity field in the undisturbed flow region occurs at a much greater downstream distance compared to the previously considered turbulent flow case (Re=16,000). Fig. 11 shows the vortex wake on the rear side of the ball, where the recirculation zones can now be clearly observed. These zones have moved significantly closer to the center of the ball, resulting in a much narrower wake compared to the previous two cases.

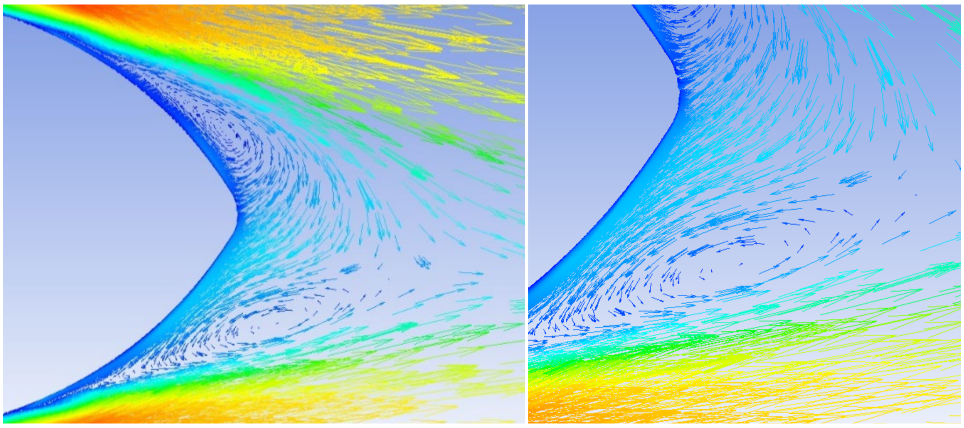


Fig 11. Vortex flow on the rear side of the ball (Re=50,000)

3. CONCLUSION

The use of the ANSYS CFX software, along with an unstructured tetrahedral mesh, enabled precise modeling of the flow characteristics and detailed representation of phenomena such as vortex shedding and velocity distributions.

The results showed that at lower Reynolds numbers (Re=1,300), the flow remained laminar with a well-defined wake, and the velocity profile reestablished at a distance of approximately 11 times the ball's diameter downstream. As the Reynolds number increased to turbulent regimes (Re=16,000 and Re=50,000), the wake region showed reduced disturbances, and the re-establishment of the undisturbed flow profile occurred at greater downstream distances. The vortex structures behind the ball became increasingly complex as the Reynolds numbers rose, indicating the onset of turbulent flow and the dominance of inertial forces over viscous forces.

This study improves the understanding of fluid flow around complex geometries like the NFL ball and emphasizes the significant impact of the Reynolds number on flow behavior. The insights gained from this study can guide the design and optimization of objects moving through fluids, such as sports equipment, underwater vehicles, and aerodynamic bodies, where controlling flow separation and reducing drag are crucial for improving performance.

Future work could expand this analysis by incorporating the effects of surface roughness, varying angles of incidence, and unsteady flow conditions to more accurately simulate real-world scenarios.

Acknowledgement: *This research was financially supported by the Ministry of Science, Technological Development and Innovation of the Republic of Serbia (Contract No. 451-03-66/2024-03).*

REFERENCES

1. Greenblatt, D., 2016, *Unsteady Low-Speed Wind Tunnels*, AIAA Journal, 54 (6), pp. 1817-1830.
2. Mühle, F. V., Heckmeier, F. M., Campagnolo, F., Breitsamter, C., 2024, *Wind tunnel investigations of an individual pitch control strategy for wind farm power optimization*, Wind Energy Science, 9 (5), pp. 1251-1271.
3. Malikov, Z.M., Madaliev, M.E., 2022, *Numerical simulation of separated flow past a square cylinder based on a two-fluid turbulence model*, Journal of Wind Engineering & Industrial Aerodynamics, 231, 105171, ISSN: 0167-6105.
4. Gurreri, L., Tamburini, A., Cipollina, A., Micale, G., 2012, *CFD analysis of the fluid flow behavior in a reverse electrodialysis stack*, Desalination and Water Treatment, 48 (1), pp. 390-403.
5. Stokes, G. G., 1851, *On the Effect of Internal Friction of Fluids on the Motion of Pendulums*, Transaction of the Cambridge Philosophical Society, 9 (2), pp. 8-106.
6. Reynolds O., 1883, *An Experimental Investigation of the Circumstances Which Determine Whether the Motion of Water Shall Be Direct or Sinuous*, Philosophical Transactions of the Royal Society of London, 174, pp. 935-982.
7. Bakić, V., 2004, *Experimental Investigation of a Flow Around a Sphere*, Thermal Science, 8 (1), pp. 63-81.
8. Wall A., Thornhill E., Barber H., McTavish S., Lee R., 2022, *Experimental investigations into the effect of at-sea conditions on ship airwake characteristics*, Journal of Wind Engineering and Industrial Aerodynamics, 223, 104933, ISSN: 0167-6105.
9. Watson N., Owen I., Prior M., White M., 2022, *Assessing Helicopter Recovery to an Offshore Platform using Piloted Flight Simulation and Time-accurate Airwakes*, 48th European Rotorcraft Forum, Winterthur, Switzerland.
10. Le T. L., Hong D. T., 2021, *Computational Fluid Dynamics Study of the Hydrodynamic Characteristics of a Torpedo-Shaped Underwater Glider*, Fluids, 6 (7), 252.
11. Constantinescu, G., Squires, D., 2003, *LES and DES Investigations of Turbulent Flow over a Sphere at Re=10,000*, Flow, Turbulence and Combustion, 70 (1), pp. 267-298.
12. Alam F., Subic A., Watkins S., Naser J., Rasul M.G., 2008, *An Experimental and Computational Study of Aerodynamic Properties of Rugby Balls*, WSEAS Transactions on Fluid Mechanics, 3 (3), ISSN: 1790-5087, pp. 279-286.
13. ANSYS, Inc., 2023, *Ansys CFX-Solver Theory Guide*, USA, Canonsburg.
14. Goodell, R., 2024, *Official Playing Rules of the National Football League*, National Football League.
15. Bogdanović-Jovanović J., Stamenković Ž., 2011, *Experimental and CFD analysis of MHD flow around smooth sphere and sphere with dimples in subcritical and critical regimes*, Thermal Science, 25 (3A), pp. 1781-1794.