Basrah Journal for Engineering Sciences, Vol. 24, No. 1, (2024), 90-96

Original Article

Journal homepage: www.bjes.edu.iq
ISSN (Online): 23118385, ISSN (Print): 18146120

A Numerical Evaluation for a Newly Designed Closed Loop Subsonic Wind Tunnel

Ridha Mohammed Ali [1, *], Ahmad A. Alsahlani [2]

[1,2] *Department of Mechanical Engineering, College of Engineering, University of Basrah, Basrah, Iraq*
E-mail addresses: reza.mohammad.ali.abbas@gmaill.com , ahmad.mahdi@uobasrah.edu.iq

Received: 2 October 2023; Accepted: 24 January 2024; Published: 15 February 2024

Abstract

A wind tunnel is a piece of equipment specifically designed for studying the influence of air passing over solid matters in aerodynamic research. Computational Fluid Dynamics (CFD) was used to conduct methodical research into the design and modeling of flow characteristic in a closed-loop wind tunnel. The necessary intake fan velocity was established using an analytical velocity model, and the test section's inlet conditions were produced by applying the Reynolds number equation, assuming that the Reynolds number was 500,000. Instead than using the traditional method, a full-scale CFD model of the complete wind tunnel was taken into consideration. This made it possible to improve the flow quality over the entire circuit as well as only in the test area. The test section flow quality was more impacted by upstream flow circumstances than downstream conditions, according to analysis of the guide vane designs. Therefore, careful consideration has to be done while constructing the vanes at upstream curves, especially corners that are parallel to the test section. The simulation results showed that, in the case of a fully configured wind tunnel, flow uniformity in the test section is successfully attained.

Keywords: CFD, Flow uniformity, Subsonic wind tunnel design, Turbulence intensity.

https://doi.org/10.33971/bjes.24.1.10

1. Introduction

Small-scale wind tunnels are quickly developing into an important research tool used in aerodynamic studies to examine the effects of air passing solid objects [1, 2]. The contraction, test section, and diffuser section are the three main parts of a wind tunnel [3]. To make sure the flow into the test area is uniform, the contraction section is employed. Contraction ratios in small wind tunnels typically range from 6 to 9 [4]. Since power consumption does not significantly affect overall building costs, the majority of small research tunnels fall into the open-circuit category. The second kind is a closed-loop wind tunnel, where air circulates repeatedly while being susceptible to directional changes. This type of wind tunnel's advantage includes greater flow quality control via corner turning vanes and screens [5]. The aim in most wind tunnel is achieving a flow in the test section that is as similar to a parallel steady flow as is feasible, with a constant speed throughout the test section [6]. In contrast, each design is constrained by limitations such as the maximum cost, the available space, and the knowledge that is currently available [7]. At the Royal Institute of Technology's newly built laboratory for aeronautical sciences [8], the first wind tunnel was finished in the summer of 1932. It had a closed circuit and an open jet test section, meaning that there were no walls in the test area. The test part was spherical in shape, with a length of roughly the same and a diameter of about 1.6 meters. It was mainly utilized for gauging forces on scale models of airplanes and airfoils. It had an axial fan and basic guide-vanes on the corners composed of bent plates shaped like 14 circles. According to Malmer (1933), the contraction ratio was around 5 and the test section's top speed was at 50 m/s. It was later changed, adding a closed test section among other things, and was still in use just a few years age [9]. Depending on the use, there are numerous distinct wind tunnel design types. According to the fluid's speed in the test chamber, wind tunnels are categorized as subsonic, transonic, supersonic, and hypersonic [10]. It makes sense to try and lower the turbulence in wind tunnels. Several techniques, including the use of honeycombs or separating plates after test sections [1, 3, 11] The specifications and standards must be in line with the applications for wind tunnels because they are closely related to the design criteria [12]. The specifications of a wind tunnel have a strong bearing on its construction and maintenance expenses, which are simply a result of the applications that are anticipated [13]. Moonen et al. created a numerical method for simulating the flow conditions in a closed-loop wind tunnel [14]. To reduce power losses and increase the likelihood of achieving a higher Mach number, closed circuit wind tunnels create a closed loop of the airstream in the chamber [15]. Wind tunnels are tools that make it possible to examine how objects interact with the airflow around them [16]. To investigate the processes that happen when a flow passes by a researched object, they create a flow at the desired speed. Low turbulence levels, flow uniformity in the test chamber, and manageable operational expenses are the major criteria for Low-Speed Wind Tunnels [17]. Figure 1 illustrates the key components of this sort of wind tunnel:

- Diffusers
- Corners
- Contraction
- Fan

- Test section
- Splitting plate

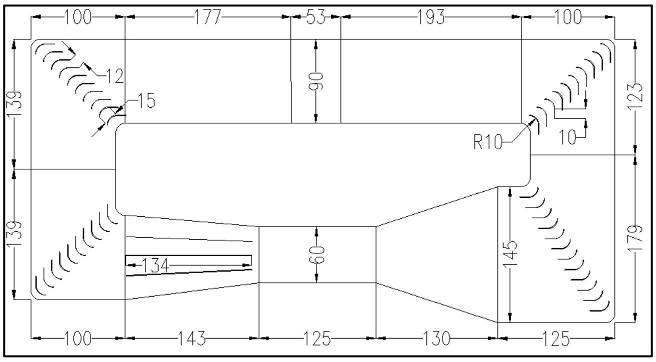

Fig. 1 Wind Tunnel Setup layout.

The purpose of this study is a part of the project intends to design and construct a new closed-loop wind tunnel to be installed in the mechanical department of the engineering collage in Basrah university. Therefore, this paper will focus only on the design configuration of the suggested wind tunnel and study its performance using CFD analysis.

The governor equations used in the analysis to simulate aerodynamic flow are based on:

1. Conservation of mass (continuity equation).
2. Conservation of momentum; second law of Newton.
3. Conservation of energy and First Law of Thermodynamics.

For more detail the reader can refer to the reference [7].

2. Wind tunnel model description

The wind tunnel system shown in Fig. 1 will be powered by a fan to provide constant air speed. A smooth transition from a circular to a 1.45 × 0.9 m^2 rectangular duct is present behind the fan. In the upper upstream elbow 10 guide vanes where guide vane diameter was 0.1 m. The trailing edge of the vanes are extended with 0.1 m and distributed with 0.12 m spacing to reduce the flow separation occurring it the turn, in lower upstream elbow 12 guide vanes with extended trailing edges are positioned in the lower upstream elbow where the lower upstream elbow driving the flow to be parallel to the test section center line and to enhance the uniformity of the flow before the contraction. The contraction section is designed with a 5.8:1 ratio with a length of 1.3 m. Also, a square cross-sectional area of 0.6 × 0.6 m^2 and 1.25 length duct is allocated for the test section. The diffuser section, which slows the air current to minimalize the loss of flow kinetic energy, locates downstream of the test section. In order to prevent these occurrences, special care is taken to prevent flow separation in the diffuser, which can drastically lower the wind turbine's overall performance, 3 horizontal splitting plates with 0.22 m spacing and 3 vertical splitting plates with 0.22 m spacing. In lower and upper downstream elbows has 10 guide vanes with 0.1 m extended trailing edges and 0.12 m distance between the vanes to mitigate the flow separation encountering in the turn. After the downstream, a smooth transition of a rectangular duct to a circular cross-section, that will match the fan dimension, takes the flow into the fan.

Table 1. Wind tunnel sections.

Name section	Description of component
Test section	Square test section, cross-sectional dimensions of 60 × 60 cm and 1.25 m length.
Fan	A 0.9 m diameter 4.6 kW axial variable-revolution fan
Diffuser 3	Diffuser area ratio of 2.25:1 and the included angle of 6°
Diffuser 2	Contraction ratio of 5.8:1, 1.45 m × 1.45 m (inlet) / 0.6 m × 0.6 m (exit) and 1.3 m (length)
Diffuser 1	circle 0.9 m diameter (inlet)/ Rectangular 1.45 m × 0.9 m (exit)
duct	Square 0.9 m × 0.9m (inlet)/circle 0.9 m diameter (exit)
Elbow 1, 2	0.9 m × 0.9 m (inlet) / 0.9 m × 0.9 m (exit)
Elbow 3	1.45 m × 0.9 m (inlet) / 1.45 m × 0.9 m (exit)
Elbow 4	1.45 m × 0.9 m (inlet) / 1.45 m × 1.45 m (exit)

3. Numerical methodology

The closed-loop subsonic wind tunnel's flow characteristics were predicted using Fluent numerical code. A uniform boundary condition of the calculated pressure and velocity was set to 20 m/s (inlet velocity)

Instead of using the traditional method, where just the flow in the test portion was modeled, a full-scale CFD model of the complete wind tunnel was taken into consideration. Four distinct wind tunnel configurations were looked into throughout the design phase. The reference configuration, which had no guide vanes, was the initial model. Three different configurations were compared to this one: one with only guide vanes at the upstream, one with only guide vanes at the downstream, and one with combined upstream and downstream guide vanes. The research assessed the effect of guide vanes on the test section's flow quality, including the uniformity of the velocity flow field, flow angularity, and turbulence intensity.

The assumption adopted in this analysis is the fan is providing a uniform flow with constant velocity and there is no passive effect from the return flow. Moreover, the flow rate will remain constant regardless the increasing in the pressure drop.

4. Results and discussion

Investigations were conducted on four alternative wind tunnel arrangements. The benchmark configuration, which had no guide vanes, was the initial model. Three different configurations were compared to this one: the first one is testing the effect of having only guide vanes at the upstream, while the second one will consider the guide vanes at the downstream, and the other case with having both the upstream and downstream guide vanes. The study assessed the effect of guide vanes' presence on the test section's flow quality by presenting the velocity contour at the various wind tunnel cross sections.

4.1. No guide vanes

The test section's velocity was 40 m/s, and the result in Fig. 2 shows the air velocity profile inside the wind tunnel with no guiding vanes at an entrance velocity of 20 m/s. Between the test portion and the lower upstream turn, the speed rose to 34 m/s. It is possible to see the inconsistent flow in the test section that was brought on by flow separations in the upstream and downstream sides of the wind tunnel corners.

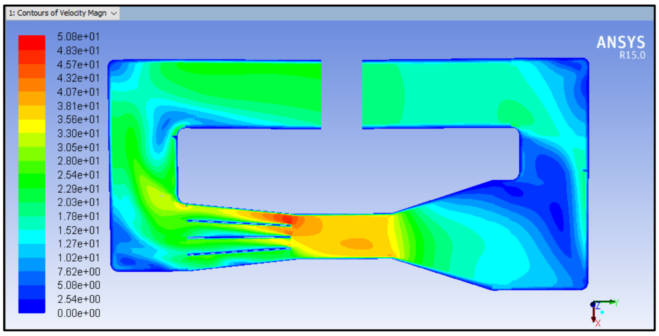

Fig. 2 Contours of velocity magnitude for configuration no guide vanes.

4.2. Upstream guide vanes

The flow profile inside the wind tunnel with upstream guide vanes is shown in the Fig. 3, the test section's velocity was lower than in the case with no guide vanes. As anticipated, after the integration of the guiding vanes, the average velocity in the test section was decreased to 35 m/s. The flow separations at the upper and lower corners were greatly reduced by guide vanes in the upstream section. There was better uniformity and symmetry in the air flow entering the test region. But it was discovered that the velocity profile was not constant; it fluctuated throughout the test segment. Due to the up-flow that was present in the top half of the test section exit, a highly disturbed flow was seen at the downstream. At the test section's departure, there was a significant flow separation that was brought on by the erratic flows in the downstream corners. Additionally, the results show that the is no improvement in the flow characteristic at the wind tunnel's downstream side.

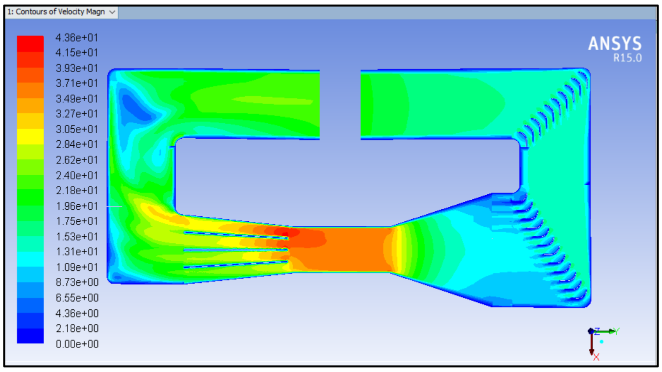

Fig. 3 Contours of velocity magnitude for upstream guide vanes.

4.3. Downstream guide vanes

Figure 4 presents the flow profile of the wind tunnel with the case of downstream guide vanes. The test section's velocity was higher than it was in the case 2 with upstream guide vanes. Significantly reducing the flow separations and circulations at the top and lower corners were guide vanes in the downstream section. However, compared to upstream guide vanes, the inclusion of downstream guide vanes did not significantly improve the test section's airflow homogeneity. This demonstrates that the quality of the flow at the tunnel's upstream section had a greater influence on the uniformity of the airflow at the test section's intake. Hence, the test section's first two upstream corners, particularly the one parallel to its center line, were crucial for achieving a consistent airflow.

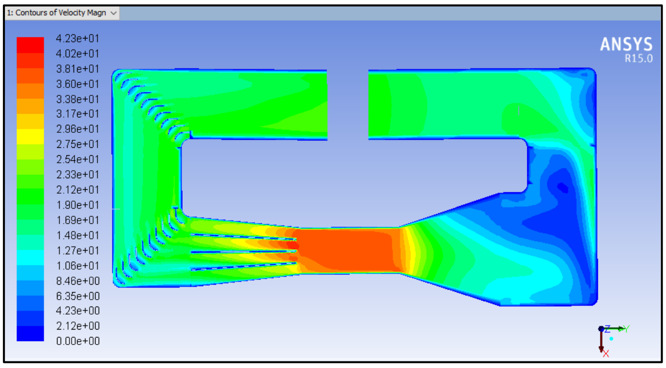

Fig. 4 Contours of velocity magnitude for downstream guide vanes.

4.4. Downstream and upstream guide vanes

The flow profile within the wind tunnel is shown in the Fig. 5 with combined upstream and downstream guide vanes, the test section's velocity was much lower than in other examples. Subsequent the integration of the guide vanes in the tunnel corners, the average speed in the test segment was decreased to 38 m/s. As anticipated, a more symmetrical, uniform flow was seen along the whole length of the test segment, with good airflow distribution throughout the full wind tunnel circuit. The velocity vectors inside the elbows crossing the vanes are shown in Fig. 6.

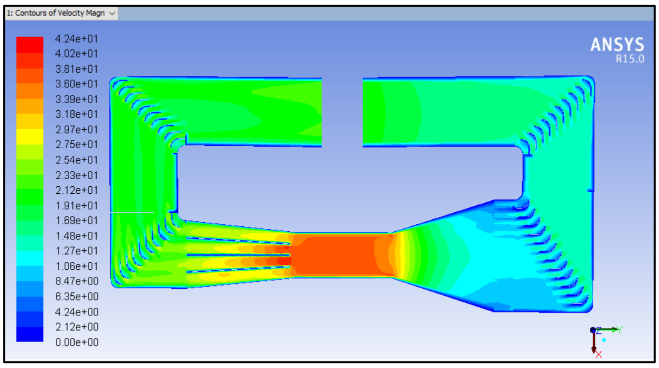

Fig. 5 Contours of velocity magnitude for guide vane.

4.5. Splitting plate

Initial CFD simulations were conducted to examine the impact of the splitting plate. Without the splitting plate as shown in Fig. 7, the flow separated at the downstream portion of the diffuser; however, the separation was significantly reduced after the integration of the splitting plate Fig. 8, as demonstrated by the uniformity of the flow field at the diffuser exit (velocity variation was reduced from 30% to 5% following the addition of horizontal and vertical splitting plates).

Fig. 9 and Fig. 10 show the cross-section velocity throughout the test section.

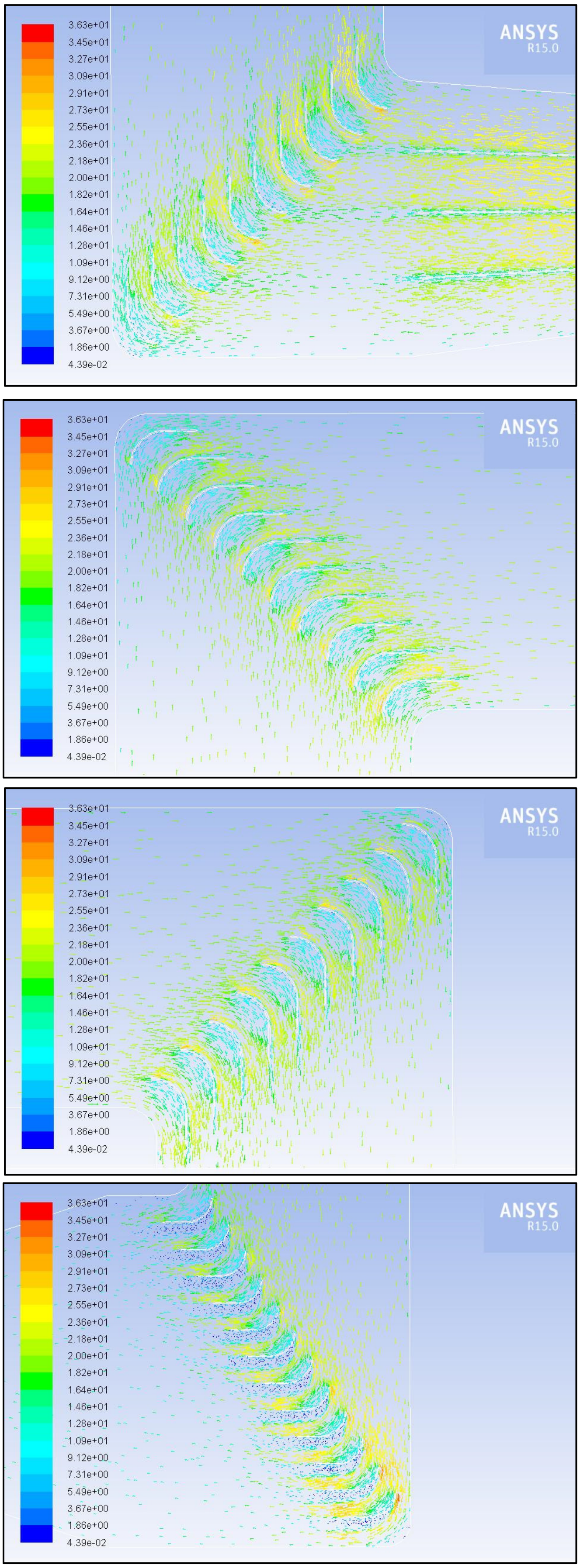

Fig. 6 velocity vectors inside the elbows crossing the vanes.

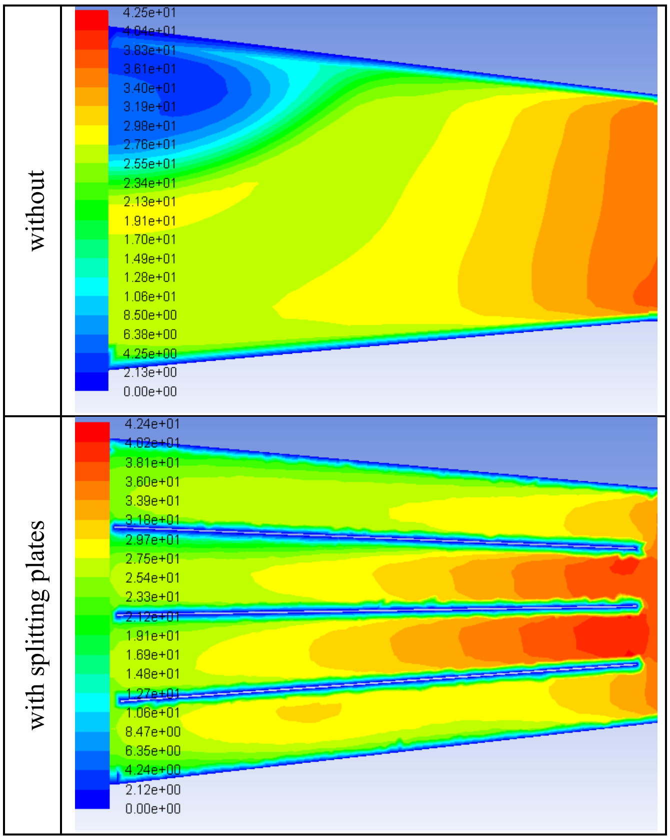

Fig. 7 Comparison of flow in the wide-angle diffuser: without and with splitting plates.

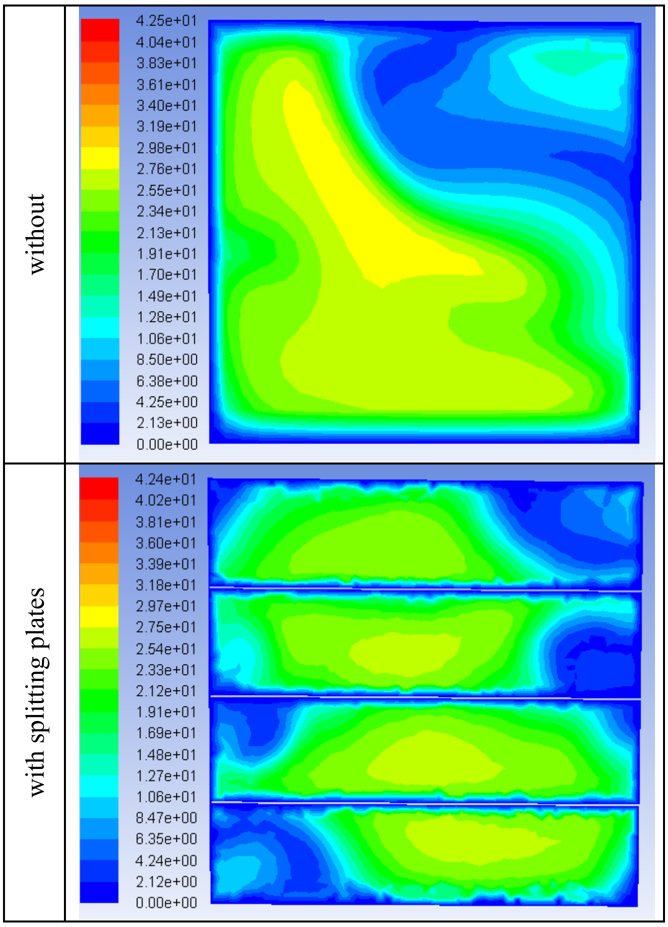

Fig. 8 Comparison of flow of cross section at outlet diffuser: without and with splitting plates.

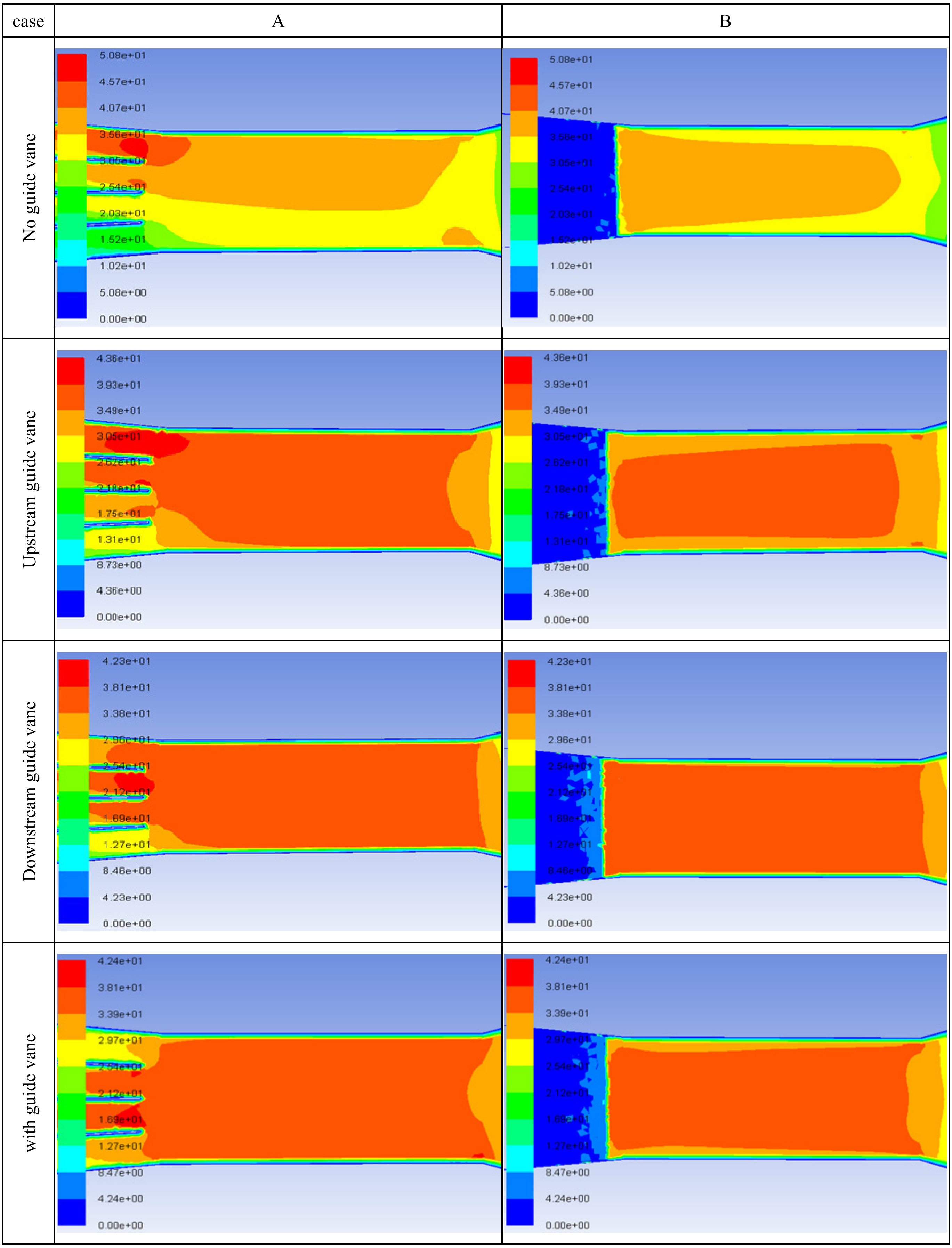

Fig. 9 Summary of velocity flow distribution in test section for the different guide vane configurations: (A) vertical (B) horizontal

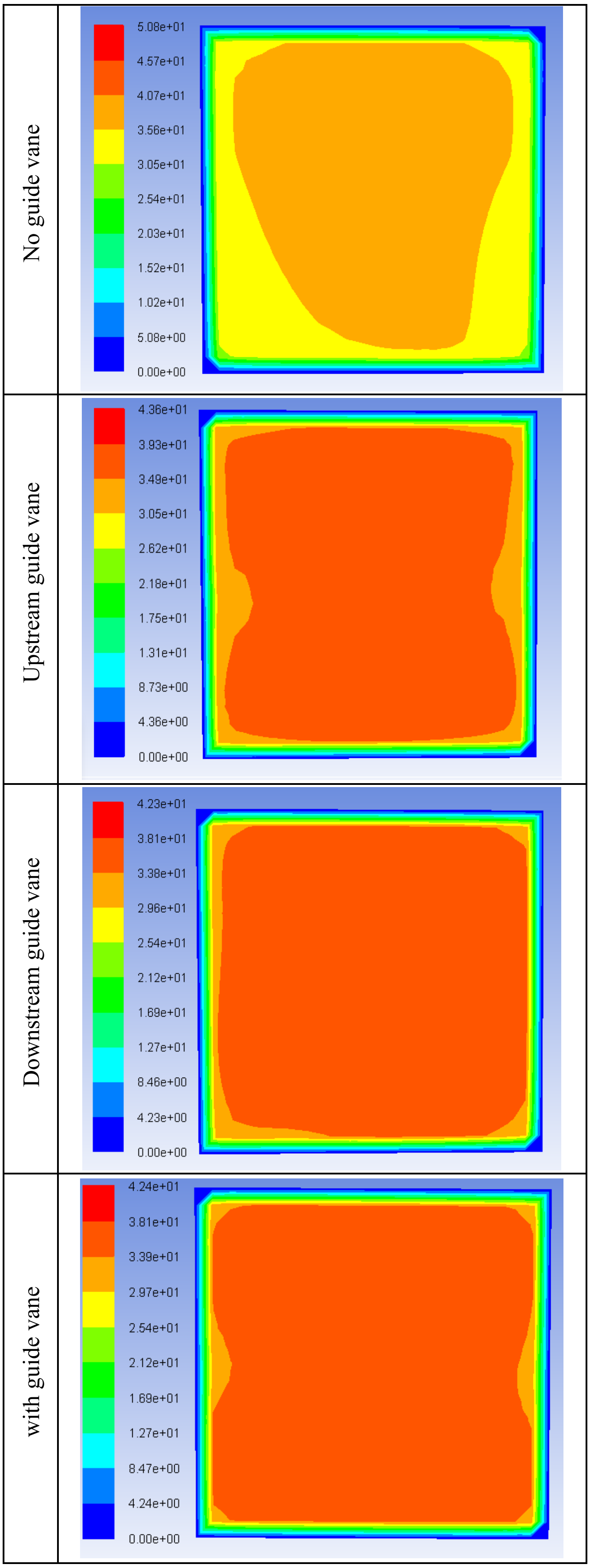

Fig. 10 Cross section in center of test section for the different guide.

5. Conclusion

Computational Fluid Dynamics (CFD) was used to conduct a numerical examination of the design and flow modeling in a closed-loop subsonic wind tunnel Fig 10. The necessary intake fan velocity was established using an analytical velocity model, where the test section's inlet conditions were derived by applying the Reynolds number equation and assuming that the Reynolds number was 500,000. Instead than using the traditional method, a full-scale CFD model of the complete wind tunnel was taken into consideration. This made it possible to improve the flow quality throughout the entire wind tunnel, not just in the test part. Through the use of guide vanes with extensions, the study created a more straightforward method for flow improvements that will undoubtedly increase the up-flow, cross-flow, and turbulence in the test section. Also, the airflow homogeneity was increased by 36% by adding guide vanes to the wind tunnel's upstream corners, and by combination of upstream and downstream guide vanes, it was increased by 65%. Only 10% of the uniformity was improved when only the downstream guide vane was present, making the situation worse than when only the upstream guide vane was present. This clearly shows that the flow condition in the upstream part had a greater impact on the test segment's flow quality than it did on the downstream section. When constructing the guiding vane at the upstream corners, special attention must be paid to the portion that is parallel to the test section. The velocity variations at the diffuser outlet were significantly decreased from 30 to 5% by the insertion of splitting plates at the diffuser section.

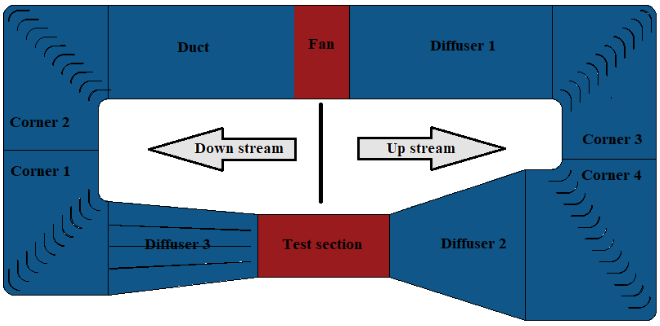

Fig. 11 Typical Wind Tunnel Setup

Reference

[1] O. D. Almeida, F. C. De Miranda, O. F. Neto, and F. G. Saad, “Low Subsonic Wind Tunnel – Design and Construction,” Journal of Aerospace Technology and Management, vol. 10, Feb. 2018. https://doi.org/10.5028/jatm.v10.716

[2] J. K. Calautit and B. R. Hughes, “CFD and experimental data of closed-loop wind tunnel flow,” Data in Brief, vol. 7, pp. 216–220, Jun. 2016. https://doi.org/10.1016/j.dib.2016.02.033

[3] K. Ghorbanian, M. R. Soltani, and M. D. Manshadi, “Experimental investigation on turbulence intensity reduction in subsonic wind tunnels,” Aerospace Science and Technology, vol. 15, no. 2, pp. 137–147, Mar. 2011. https://doi.org/10.1016/j.ast.2010.06.009

[4] R. D. Mehta, “Turbulent boundary layer perturbed by a screen,” AIAA Journal, vol. 23, no. 9, pp. 1335–1342, Sep. 1985. https://doi.org/10.2514/3.9089

[5] S. D. Kulkarni, M. A. Minor, M. Deaver, E. R. Pardyjak, and J. M. Hollerbach, “Design, Sensing, and Control of a Scaled Wind Tunnel for Atmospheric Display,” IEEE-ASME Transactions on Mechatronics, vol. 17, no. 4, pp. 635–645, Aug. 2012. https://doi.org/10.1109/tmech.2011.2113353

[6] L. López de Vega, F. J. Maldonado Fernández, and P. Muñoz Botas, Optimisation of a Low-Speed Wind Tunnel, Analysis and Redesign of Corner Vanes, 2014.

[7] J. B. Barlow, W. H. Rae, and A. Pope, Low-speed wind tunnel testing, 3rd ed. New York: Wiley, 1999.

[8] B. Lindgren, and A. V. Johansson, “Design and evaluation of a low-speed wind-tunnel with expanding corners”, Technical Reports from Royal Institute of Technology, Department of Mechanics, SE-100 44 Stockholm, Sweden, 2002.

[9] C. Wexler, “Experimental Measurement and Computational Simulation of the Flow in the Queen’s University Low Speed Wind Tunnel”, M.Sc. thesis, Mechanical and Materials Engineering, Queen's University, 2014.

[10] T. H. Yong and S. S. Dol, “Design and Development of Low-Cost Wind Tunnel for Educational Purpose,” IOP Conference Series: Materials Science and Engineering, vol. 78, p. 012039, Apr. 2015. https://doi.org/10.1088/1757-899x/78/1/012039

[11] R. D. Mehta and P. Bradshaw, “Design rules for small low speed wind tunnels,” The Aeronautical Journal, vol. 83, no. 827, pp. 443–453, Nov. 1979. https://doi.org/10.1017/S0001924000031985

[12] H. Weisweiler, J. Kojouharova, and R. Dückershoff, “Design of short low speed Göttingen type wind tunnel: CFD simulation,” MATEC Web of Conferences, vol. 145, p. 03015, 2018. https://doi.org/10.1051/matecconf/201814503015

[13] M. A. Gonzalez Hernandez, A. I. Moreno Lopez, A. A., J. M. Perales Perales, Y. Wu, and S. Xiaoxiao, “Design Methodology for a Quick and Low-Cost Wind Tunnel,” Wind Tunnel Designs and Their Diverse Engineering Applications, Mar. 2013. https://doi.org/10.5772/54169

[14] H. Chaudhry, J. Calautit, B. Hughes, and L. Sim, “CFD and Experimental Study on the Effect of Progressive Heating on Fluid Flow inside a Thermal Wind Tunnel,” Computation, vol. 3, no. 4, pp. 509–527, Oct. 2015. https://doi.org/10.3390/computation3040509

[15] A. Teseletso, M. Namoshe, N. Subaschandar, and S. Kutua, “Design of an Open-circuit Subsonic Wind Tunnel for Educational Purpose”, In Botswana Institution of Engineers (BIE), 14th International Biennial Conference, At Phakalane, Botswa, 2015.

[16] K. Breuer, M. Drela, X. Fan, and M. Di Luca, “Design and performance of an ultra-compact, low-speed, low turbulence level, wind tunnel for aerodynamic and animal flight experiments,” Experiments in Fluids, vol. 63, no. 11, Oct. 2022. https://doi.org/10.1007/s00348-022-03519-1

[17] P. Moonen, B. Blocken, S. Roels, and J. Carmeliet, “Numerical modeling of the flow conditions in a closed-circuit low-speed wind tunnel,” Journal of Wind Engineering and Industrial Aerodynamics, vol. 94, no. 10, pp. 699–723, Oct. 2006. https://doi.org/10.1016/j.jweia.2006.02.001