Terbit online pada laman web jurnal: **https://journal.isas.or.id/index.php/JAMERE**

JOURNAL OF APPLIED MECHANICAL ENGINEERING AND RENEWABLE ENERGY (JAMERE)

Vol. 4 No. 2 Agustus 2024 38-45 ISSN: 2775-1031

Prediction of Airflow Velocity in Wind Tunnel Test Section Based on Blade Pitch Angle

Windy Maryani Pandiangan[1], Sugianto[2], Deden Masruri[3]
[1,2,3]Teknik Mesin, Politeknik Negeri Bandung
[1]windy.maryani.aer21@polban.ac.id, [2]sugianto@polban.ac.id*, [3]dedenmasruri@polban.ac.id

Abstract

The goal of this study was to determine the blade pitch angle configuration that provides the maximum velocity and to acquire the airflow velocity distribution in the test section using an open-circuit subsonic wind tunnel at Bandung State Polytechnic's Aeronautics Hangar. The wind tunnel has a total length of about 6 m with an octagonal test section measuring 40 cm x 40 cm x 155 cm and 7 cm chamfers at each corner. The research method involved numerical simulation using ANSYS and 3D modelling with Solidworks. Simulations were conducted at 500 RPM with blade pitch angles of 40°, 50°, 60°, and 70°. The results showed the highest speed at 50° blade pitch angle (16 - 18 m/s), with the lowest speed at 70° blade pitch angle (9 - 13 m/s). Tests at a blade pitch angle of 50° with variations in fan rotational speed (300 RPM, 500 RPM, and 700 RPM) also showed similar speeds (16.2 m/s at 500 RPM). It was found that the 50° blade pitch angle produced significant vortex strength which was influenced by the high airflow velocity. During the test, to improve the accuracy of airflow velocity measurement, it is necessary to ensure that all parts of the wind tunnel are closed.

Keywords: wind tunnel, fluid computation, fan simulation, moving reference frame, test section

1. Introduction

In the world of engineering and aerodynamics, airflow velocity testing in wind tunnel test sections plays an important role in the development of aerodynamic vehicles and structures. Subsonic open-circuit wind tunnels are used as the primary facility for testing and optimizing the aerodynamic performance of an object, such as a car, plane, or other object [1][2][3]. For example, an aeroplane is tested using a wind tunnel to observe the drag received by the aeroplane due to the wind flow. In a wind tunnel simulation, the model is assumed to be stationary and the air moves at a certain speed. On the contrary, in real conditions, the aeroplane or car is considered to be moving and the air is relatively stationary. That is why the airflow in the wind tunnel test section must meet certain requirements [1][2]. One of the critical aspects of testing in this subsonic open-circuit wind tunnel is the blade angle configuration that affects the resulting airflow velocity. Proper blade angle can improve test efficiency and generate accurate data [1][4][5].

In [1] it is mentioned that by changing the blade angle configuration it can be done to adjust the Mach Number value. The method of configuring the blade tilt angle has been carried out to predict the airflow velocity generated from the blade angle variation used [4][5]. In [5], the maximum shaft rotational speed of 178 rpm was produced at a horizontal blade pitch angle of 40° with a wind speed of 5.2 m/s, the variable pitch of the fan allows it to continue to produce flows up to 25 m/s in a stable and fairly calm condition because the shaft speed can be kept constant in low rotation conditions. Analysis from CFD (Computation Fluid Dynamics) simulations using ANSYS Academic Research 2023 R-1 can be performed to predict the airflow velocity in the wind tunnel test section. In [6], the design and simulation for a knockdown type open loop wind tunnel has a maximum speed in the test section reaching 20 m/s.

In [7] the results of numerical simulations showing the visualization of air flow velocity in various geometry variations show that the test section has the maximum air flow velocity, reaching 36 m/s. In [8] numerical simulations have been carried out for the LS-LST wind tunnel (LAPAN Small-Low Speed Tunnel) with the half body method to accelerate the results of the analysis aimed at evaluating the results of the wind tunnel design that has been carried out in 2013 with the target air flow velocity in the test section reaching 30 m/s. Based on the description and problems above, by understanding how blade angle configuration affects airflow velocity, researchers can optimize wind tunnel settings to achieve ideal test conditions [6][9]. Therefore, this study aims to investigate and predict the velocity in the test section based on the blade angle configuration in an open-circuit subsonic wind tunnel.

2. Research Methods

The method that will be used to solve this research problem is divided onto several stages which are explained onto a flow chart. The process carried out is divided into two, namely, the simulation process and the testing process. Then, the outcomes of the simulation process will next be compared to the test findings, and the validation results will be examined for consistency. After the comparison review process is carried out, conclusions can be drawn based on the results of the comparison review.

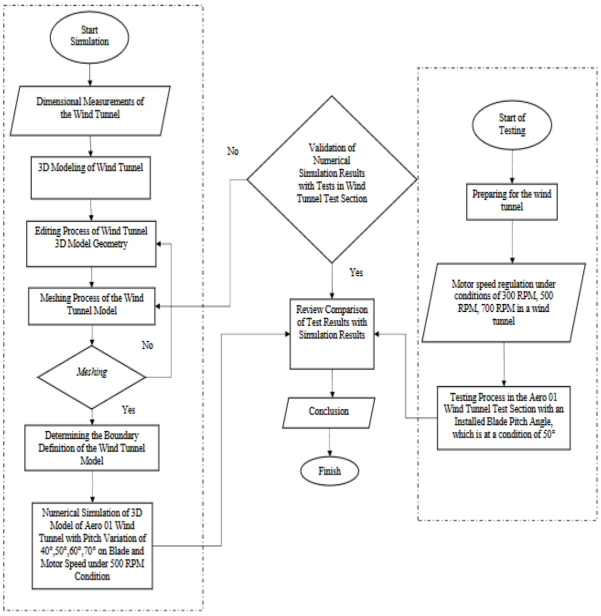


Figure. 1 Research Flow Chart

2.1. Research Variables

In the research in predicting the airflow velocity in the test section based on the configuration of the blade pitch angle, the pitch angle makes reference to the shaft's axis as shown in Figure.3, where the shaft becomes a reference when going to do numerical simulations. The blade pitch angle is in the form of a grade/line on the blade surface as shown in Fig.2.

The simulation is only run at 500 RPM and has pitch angles including 40°, 50°, 60°, 70° means that the blade will be simulated at four different angles in this study. Only the installed pitch angle that is, a pitch angle of 50 ° with 300 RPM, 500 RPM, 700 RPM was utilized throughout testing. Due to the fact that the pitch angle utilized is just the installed pitch angle at a pitch angle of 50 ° with the previously stated RPM, there are four variables during numerical simulation and three variables during testing.

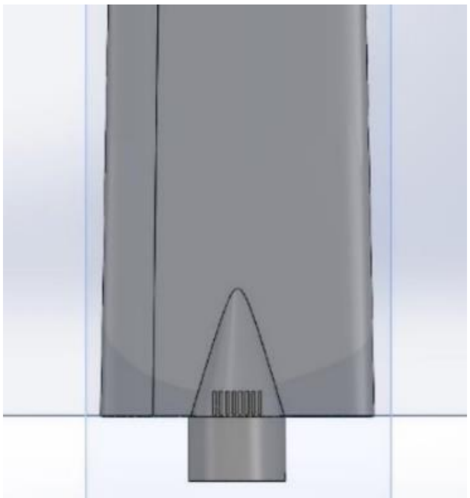


Figure. 2 Grade of Blade

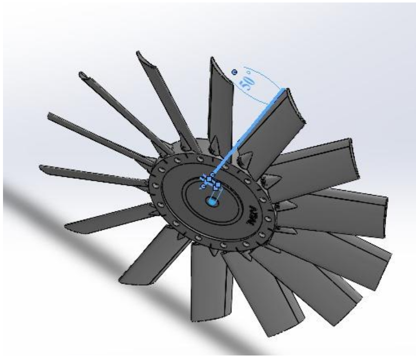


Figure. 3 Blade Pitch Angle Against Shaft Axis

2.2. Wind Tunnel 3D Model Building and Editing Process

In the process of making a 3D model of the wind tunnel using CAD (Computer Aided Design) software, namely Solidworks by drawing parts of the wind tunnel that refers to the results of dimensional measurements on the wind tunnel that have been carried out The process of making a 3D model of a wind tunnel begins with drawing using the Part feature in Solidwork for each component contained in the wind tunnel then each part is put together using the assembly feature in Solidwork, Figure.4 is an example of using the assembly feature to combine the blade on the hub.

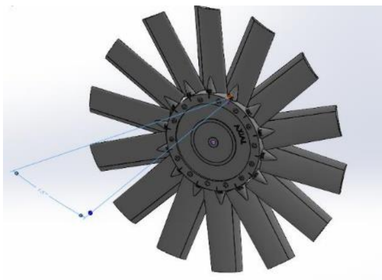


Figure. 4 Model 3D Fan Blade Assy

The results of making a 3D wind tunnel model using Solidworks software with SLDRPT format are imported into the IGS Format. So that the editing process can be carried out in ANSYS software. The editing process is carried out to improve or simplify the results of CAD 3D modeling of wind tunnels so that simulations can be carried out[10]. At this stage the scale for the geometry is reduced using a ratio of 1:1000 or 0.001 to uniform the units into meters. This can be done by using the "Scale" feature. Before doing the 'scale down' as mentioned above to uniform the unit in meters, use the 'face' geometry feature to create a 3D model of the wind tunnel according to the existing size. Then, use the 'sweep faces' feature in the volume geometry. Then by using the 'vector' feature and selecting 'magnitude', enter the number according to the measurement results that have been made into the direction that is adjusted to the needs of 3D modeling. In addition, the 'unite' feature can be used to combine all the individual parts, planes and volumes that have been created so that they can become one part. Then to eliminate one of the covered volumes by using the volume that covers it can use the 'substract' feature, the subtract process is a geometry editing process whose function is to reduce or eliminate one of the volumes using the volume that covers it. The purpose of removing one of the volumes in the wind tunnel geometry is to make the solid part into an empty part, for example in the volume geometry between the fan blade and the hub in the wind tunnel.

2.3. Meshing Process

The meshing process aims to divide the plane into small elements that can be studied in numerical simulations. In the meshing process, previously the file must be made into msh format with the aim that the file that has been done can be simulated in ANSYS Academic Research 2023 R-1. Fig.5 is an example of the meshing display on the 3D model of the wind tunnel test section, the method used in this process is the multi-block method where the geometry is meshed individually based on its volume. Therefore, each volume can have different types of connections. Each stage of the meshing process itself is carried out in stages starting from the most basic type of meshing, namely, edge meshing, face meshing, to volume meshing. This method is also used to provide different mesh variants at each geometry level as needed, and as a solution for complex geometry parts such as fan blades.

3. Results and Discussion

From the wind tunnel modeling that has been made in Solidworks software, then each part is combined using the assembly feature. Figure.5 is a comparison of 3D modeling results with the original shape of the wind tunnel and Figure.6 displays a comparison of 3D modeling results with the original shape of the fan blade assy.

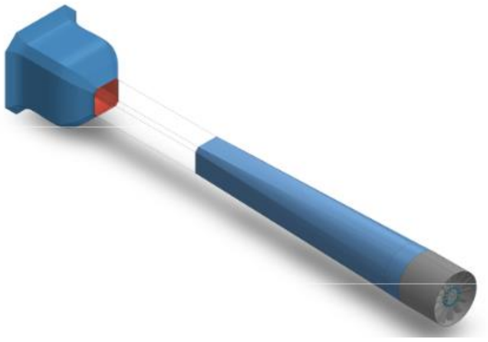


Figure. 5 Comparison of Open Circuit Subsonic Wind Tunnel 3D Model (top), real shape (bottom)

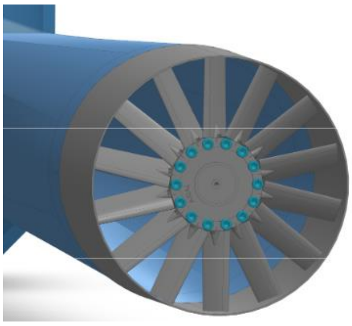


Figure. 6 Comparison of Fan Blade Assy 3D Model (top), real shape (bottom)

The meshing process that has been carried out produces various types of meshing in each part of the wind tunnel geometry, see Fig.7. The meshing type in the bellmouth, test section uses hexahedral meshing type as shown in Fig.8, then the tetahedral meshing type, where this element has four triangular surfaces and is used to represent the volume in the domain. Tetahedral meshing is applied to complex geometries such as diffuser, hub, and blade geometries, as well as fan cases., see Fig.9, Fig.10 and Fig.11. This element has four triangular faces and is used to represent volumes in the domain. After meshing is done, defining the boundary conditions is done as in Fig.7 where the inlet is located at the front of the wind tunnel (bellmouth) and the outlet is located at the back of the wind tunnel (after fan), the inlet and outlet used in the simulation is the Pressure Gauge at sea level, which is at 1 atm or 1.01325 bar in the fan off condition.

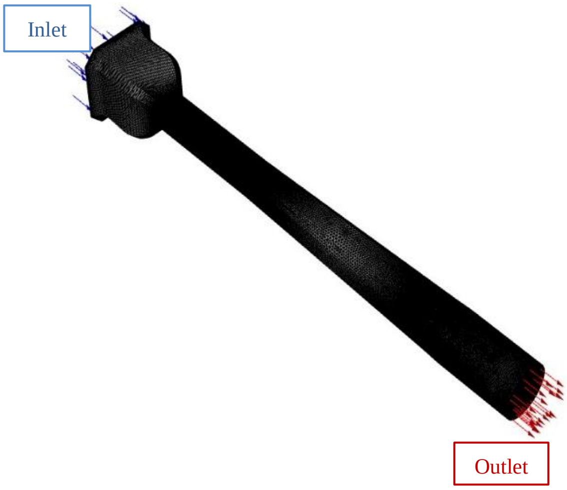


Figure. 7 Meshing on the Wind Tunnel

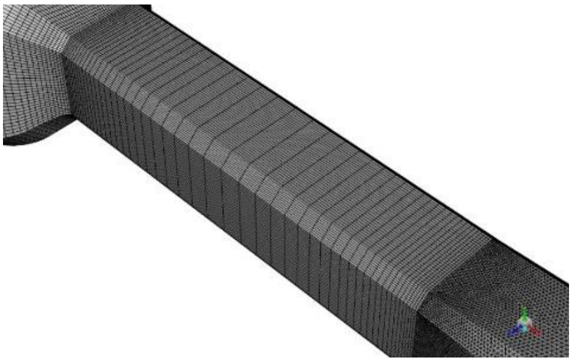


Figure. 8 Hexahedral Meshing of Bellmouth and Test Section Geometry

Figure. 9 Tetrahedral Meshing on Diffuser Geometry

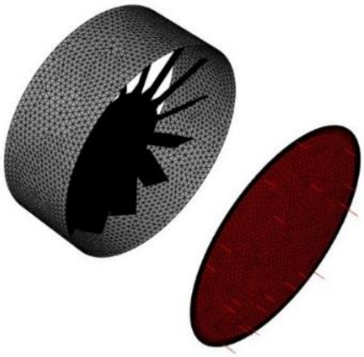


Figure. 10 Tetrahedral Meshing on Fan Hub Geometry

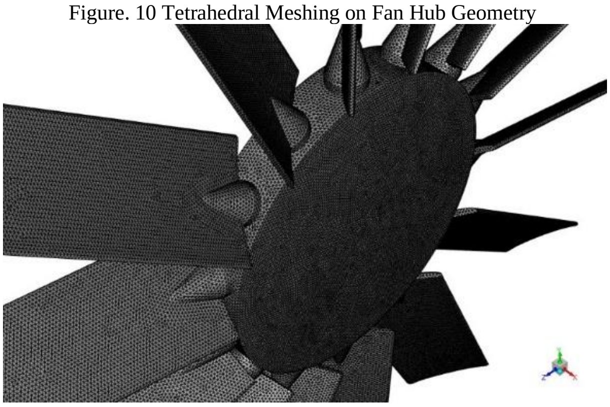


Figure. 11 Tetrahedral Meshing of Hub and Fan Blade Geometry

The simulation results are in the form of visualizations and graphs. The airflow pressure, turbulence severity, and speed inside the wind tunnel are all visualized. The airflow velocity and pressure profile seen on the graph is what happens in the wind tunnel when the fan rotates at 500 RPM and the blade pitch angles vary from 40° to 70°.

Fig.13 displays the airflow velocity cut against the isometric XYZ axis to clarify the difference in airflow velocity in each section of the wind tunnel. In the test section with a blade pitch angle of 40° reaches 11 - 13 m/s, at a blade pitch angle of 50° reaches 16 - 18 m/s, at a blade pitch angle of 60° reaches 13 - 16 m/s and at a blade pitch angle of 70° reaches 9 - 13 m/s. This can prove that the highest speed occurs in the test section where the smallest area of the entire wind tunnel section.

It also proves that the variation of blade pitch angle used affects the airflow velocity that occurs in each part of the wind tunnel. From Fig.14 we can also know that the airflow velocity increases from the blade pitch angle of 40° to 50°, then from the blade pitch angle of 50° to 60° and 70° there is a phenomenon of decreasing airflow velocity. This can occur with the assumption that the blade at a pitch angle of 50° is the most efficient angle and when the angle changes to 60° and 70°, the blade experiences a stall similar to the airfoil concept. In two dimensions, the blade's shape resembles an airfoil; that is, the greater the angle of attack, or aoa, as it is popularly known, the greater the airfoil's tendency to stall, the same phenomenon occurs on the blade where the higher the pitch angle given, the blade will experience separation, namely, a condition where the air flow is separated or does not stick (thoroughly) through the blade surface as shown in Fig.12 where the simulation results are made with a cutting plane at a radius of 25 cm to describe the airflow velocity distribution that occurs on the blade contained in the drive section.

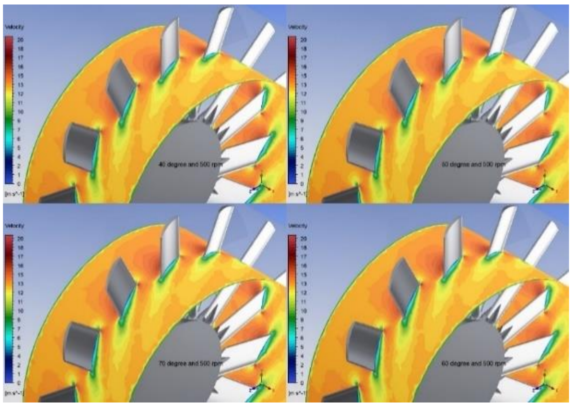


Figure. 12 Airflow Velocity in Drive Section with Blade Pitch Angle Configuration 40°,50°,60°,70° at 500 RPM

Fig.16 displays the absolute pressure cut against the isometric XYZ axis to clarify the difference in absolute pressure in each section of the wind tunnel. Test sections with 40°, 50°, and 60° blade pitch angles reach 1.0127 bar, 1.0119 bar, and 1.0125 bar, respectively, while test sections with 70° and 60° blade pitch angles reach 1.0114 bar. If it is related to the discussion in the airflow velocity visualization subchapter, the phenomenon of ups and downs in pressure can also occur because it is caused by a stall or separation condition, if there is separation then turbulence occurs, ai pressure will fluctuate if the airflow velocity fluctuates, which is caused by turbulence disrupting the airflow velocity. Figure.15 where the simulation results are made with a cut plane at a radius of 25 cm to describe the airflow pressure distribution that occurs on the blade in the drive section.

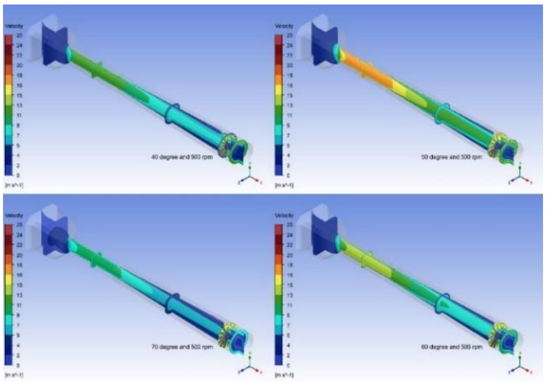
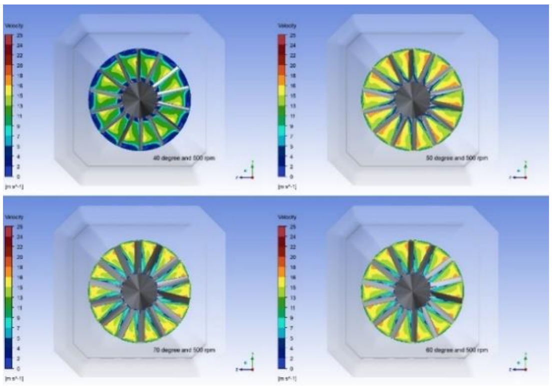


Figure. 13 Airflow Velocity with Blade Pitch Angle Configuration 40°,50°,60°,70° at 500 RPM against XYZ Isometric Cut Axis



Gambar. 14 Airflow Velocity Contours on Fan Blade with Blade Pitch Angle of 40°, 50°, 60°, 70° at RPM 500

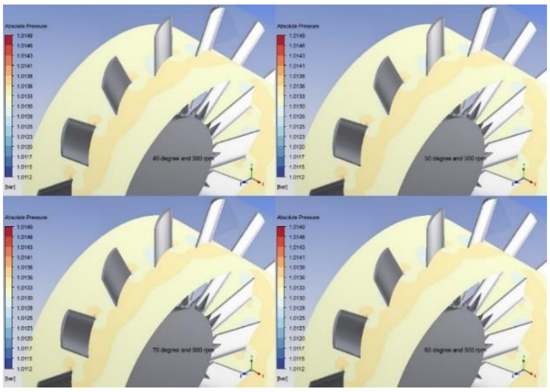


Figure. 15 Airflow Pressure in Drive Section with Blade Pitch Angle Configuration of 40°, 50°, 60°, 70° at 500 RPM

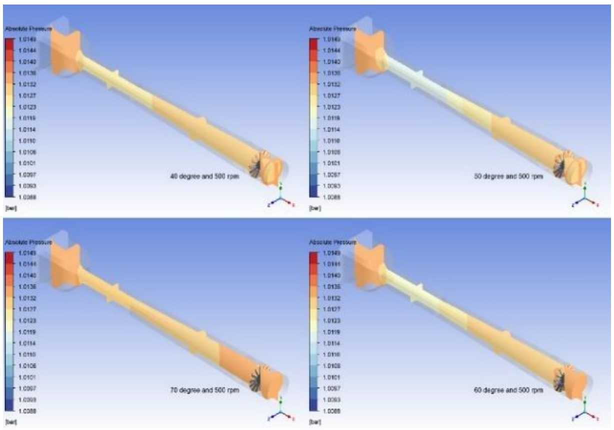


Figure. 16 Airflow Pressure with Blade Pitch Angle Configuration 40°,50°,60°,70° at RPM 500 Against XYZ Isometric Cut Axis

Fig.17 displays the intensity of airflow turbulence cut against the isometric XYZ axis to clarify the difference in airflow turbulence intensity in each section of the wind tunnel. In the test section with blade pitch angles of 40°, 60°, 70°, the numbers reach 0.00 - 0.23. However, at a blade pitch angle of 50°, it reaches 0.00 - 0.43. The turbulence intensity standard in the test section is 5% and based on the results of this simulation, it can be stated that the turbulence intensity in the Aero 01 wind tunnel test section is still within the tolerance standard.

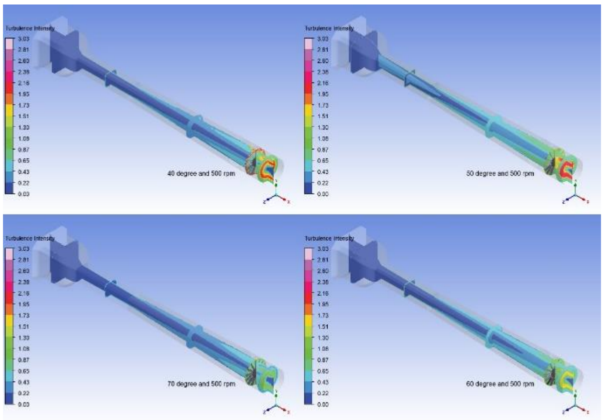


Figure. 17 Turbulence Intensity of Airflow with Blade Pitch Angle Configuration 40°,50°,60°,70° at 500 RPM against XYZ Isometric Cut Axis

Fig.18 is the airflow velocity distribution profile that occurs in the wind tunnel. The graph is obtained from the numerical simulation results. The graph is formed from the x-axis in the form of length (m) against the axis of the shaft, where the axis of the shaft is located at the hub of the fan blade, so the graph display starts from -6 because the positive x direction leads to the position after the fan blade and negative x leads to the position of the diffuser, test section, to the bellmouth. So the furthest position from the axis is -6 for the bellmouth position, -5.4 to -5 is the contraction cone position, -5 to -4 is the test section position, -4 to -2 is the diffuser position, and -2 to -1 is the drive section position. The Y-axis of the graph is the airflow velocity value along the wind tunnel. Referring to the law of continuity, the airflow velocity distribution graph above proves that the smaller the area, the higher the airflow velocity, and the larger the area, the lower the airflow velocity[11]. From the graph, we can also see that the highest airflow velocity is at a blade pitch angle of 50° and the lowest airflow velocity is at a blade pitch angle of 70°.

Fig.19 is the airflow pressure profile that occurs along the wind tunnel. From the graph, we can also know that the highest airflow pressure is at a blade pitch angle of 70° and the lowest airflow pressure is at a blade pitch angle of 50°. The airflow pressure distribution graph above proves that the larger the area, the higher the airflow pressure, and the smaller the area, the lower the airflow pressure.

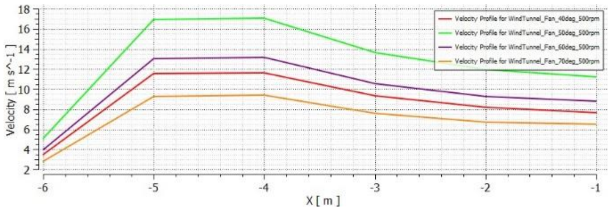


Figure. 18 Graph of Airflow Velocity in the Wind Tunnel

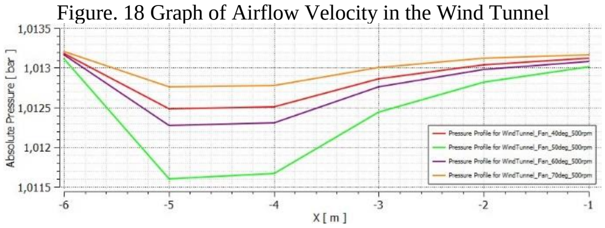


Figure. 19 Graph of Airflow Pressure in The Wind Tunnel

Fig.20 shows the test results that the airflow velocity increases as the fan rotation increases. At 300 RPM, the average air velocity is around 10.4 m/s. At 500 RPM, the air velocity increases to an average of around 16.2 m/s, at 700 RPM the average air flow velocity reaches around 22.6 m/s. The data collection process is carried out by measuring the airflow velocity using an anemometer at points in the wind tunnel section such as the bellmouth, the centre of the test section and the after fan where the measurement points are used as the Y axis and the results of measuring the airflow velocity at these points are used as the X axis on the velocity profile shown in Figure 20.

Comparison between test and simulation results seen in Fig.21 that the resulting graph has the same pattern, where the smaller the area traversed by the airflow, the airflow velocity will increase and the larger the area traversed by the airflow, the airflow velocity will decrease. From Fig.21, it can be seen that the dot is the value of the test results while the dotted line shows the simulated airflow velocity along the wind tunnel at a blade pitch angle of 50° at RPM.

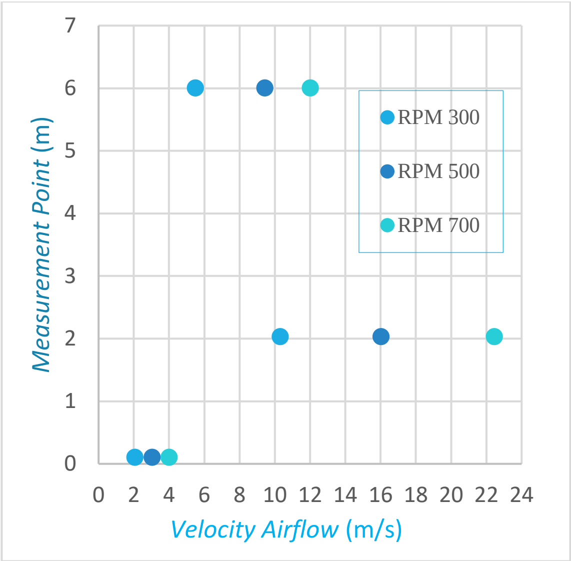


Figure. 20 Speed Profile in Wind Tunnel from Test Results

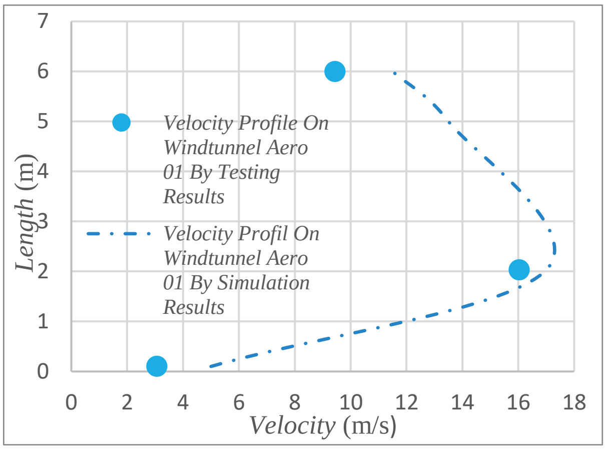


Figure. 21 Comparison of Airflow Velocity Values from Test Results and Simulation Results

It can be seen that there is a similar trendline where the largest airflow velocity occurs in the wind tunnel test section. A comparison between the test and simulation results shows that the airflow velocity values generated from the test and simulation are different. This may be due to several factors such as measurement accuracy and test conditions. Airflow velocity measurements during testing may not be 100% accurate, while simulations use mathematical models that can have assumptions and uncertainties. If calculated using the relative error percentage, the comparison between the test and simulation results reaches 6.25%. Meanwhile, the test conditions such as temperature and air pressure used are the pressure at Bandung altitude and the pressure used in the simulation is the pressure at sea level.

Although there is a difference in value, this comparison shows that the simulation model can be used to predict airflow velocity in the wind tunnel quite well because the relative error still shows a number below 10%, where 10% is the tolerance limit for the relative error value for comparison of test results with simulation results.

4. Conclusions

The conclusions obtained from this research are as follows:

Comparison of airflow velocity in the test section from numerical simulation results and testing with variations of fan rotational speed of 500 RPM and blade pitch angle of 50° get almost the same airflow velocity at ± 16 m/s. The blade angle configuration that produces the greatest velocity in the test section is in the condition of a 50° blade pitch angle. From the results of this study obtained suggestions, namely, for the next simulation can be done with variations in fan rotational speed (RPM) in accordance with the fan rotation speed used during testing. For the next test, it can be done by varying the blade pitch angle in accordance with the blade pitch angle used in the simulation. Then, when testing airflow velocity measurements in the wind tunnel, make sure the measuring instrument has been calibrated and is in good condition and also make sure the wind tunnel conditions are completely closed (avoid gaps) for accuracy in the airflow velocity measurement value.

Acknowledgements

DIPA Politeknik Negeri Bandung 2024 for final project cost assistance

Reference List

[1] B. Chanetz, J. Délery, P. Gilliéron, P. Gnemmi, E. R. Gowree, and P. Perrier, *Experimental aerodynamics: An Introductory Guide*. 2020.

[2] J. D, J. Anderson, C. P, and Cadou, *Fundamentals of Aerodynamics Seventh Edition*. 2023. [Online]. Available: https://www.crcpress.com/Fundamentals-of-Picoscience/Sattler/p/book/9781466505094#googlePreviewContainer

[3] Fariduzzaman, *Inovasi Teknologi Pengujian Aerodinamika dan Aeroelastika untuk Rancang Bangun Jembatan Bentang Panjang*. 2022. doi: 10.55981/brin.697.

[4] H. SAFI'I, "Studi Performa Turbin Angin Sumbu Vertikal NACA 0012 Dengan Turbin Angin Darrieus-H Pada Variasi Sudut Pitch 30°,40°,45°,50°,55°,60°," *Publ. Ilm. Univ. Muhammadiyah Surakarta Fak. Tek.*, p. 20, 2016.

[5] W. J. Rumbiak, "Pengujian dan Analisis Performansi Model Turbin Angin Tipe Poros Silang Kapasitas Daya 10 W," 2021.

[6] D. Rhakasywi and A. Suwandi, "Pengembangan Terowongan Angin Rangkaian Terbuka Dengan Sistem Piv (Particle Image Velocimetry)," 2017.

[7] H. N. Firmansyah, P. Wirardi, R. F. Naryanto, and K. Karnowo, "Simulasi 3D dan Studi Eksperimental Aliran Udara pada Variasi Geometri Menggunakan Wind Tunnel," *J. Rekayasa Mesin*, vol. 18, no. 3, p. 395, 2023, doi: 10.32497/jrm.v18i3.4973.

[8] T. M. I. Hakim, "Evaluasi Perancangan Terowongan Angin LS-LST Dengan Simulasi Numerik," *Litbangyasa Teknol. Pada Pesawat Terbang, Roket*, vol. 5, pp. 101–106, 2015, [Online]. Available: http://karya.brin.go.id/id/eprint/11046/%0Ahttp://karya.brin.go.id/id/eprint/11046/1/Bunga Rampai_Teuku_Pustekbang_2015.pdf

[9] A. T. Teseletso, M. Namoshe, N. Subaschandar, and S. Kutua, "Design of an Open-circuit Subsonic Wind Tunnel for Educational Purpose," *Botswana Inst. Eng. 14th Bienn. Conf.*, no. outubro, 2015.

[10] M. Jafar Sidiq, J. T. Mesin, and P. N. Bandung, "Simulasi numerik aliran udara di dalam terowongan angin kecepatan rendah tipe terbuka," 2014.
[11] I. G. Rasagama, K. Handinigrun, and R. F. Muldinani, "Pengembangan Model Praktikum 'Persamaan Bernoulli' Untuk Pembelajaran Konsep Fluida Dinamis Mahasiswa Politeknik Negeri Bandung," *Pros. Ind. Res. Work. Natl. Semin.*, vol. 9, pp. 790–801, 2018.