SEMARAK ILMU PUBLISHING

CFD Letters

Journal homepage:
https://semarakilmu.com.my/journals/index.php/CFD_Letters/index
ISSN: 2180-1363

3-Dimensional CFD Simulation and Experimental Analysis on Performance of a New Diffuser and Wind Channel for a Micro Wind Turbine

Zeinab Darvishi[1], Gholamhassan Najafi[1,*], Barat Ghobadian[1], Nor Azwadi Che Sidik[2], Rizalman Mamat[3], Mohd Fairusham Ghazali[3], Seyed Salar Hoseini[1]

[1] Mechanical of Biosystems Engineering Department, Tarbiat Modares University, Tehran, Iran
[2] Malaysia-Japan International Institute of Technology, UTM Kuala Lumpur, Jalan Sultan Yahya Petra, Kuala Lumpur, Malaysia
[3] Center for Research in Advanced Fluid and Process, University Malaysia Pahang, Lebuhraya Tun Razak, Gambang, Kuantan 26300, Pahang, Malaysia

ARTICLE INFO

Article history:
Received 17 March 2022
Received in revised form 30 May 2022
Accepted 1 June 2022
Available online 7 June 2022

Keywords:
Micro turbine; Wind channel; Electrical production; RSM; CFD

ABSTRACT

Producing electricity from waste energies has been introduced as a useful approach in moving towards stability. Therefore, in the current study a new wind channel has been designed and fabricated to convert created airflow resulting from automobiles movement on the highways to electricity. This channel is fabricated to guide airflow from automobiles movement inside the same directed channel and towards a microturbine located at the bottom of the wind channel. To uniform the airflow inside the channel, a new diffuser has been designed and fabricated. The diffuser minimizes airflow turbulence in a part of the channel where microturbine is located. The maximum velocity is obtained in the horizontal distance is 1.9 m, the vertical distance is 0.95 m, and the angle is 45° according to automobiles' movement. This study also studied fluid behavior through the channel using 3-Dimensional CFD simulation technique and the best type of channel and diffuser fabricated based on numerical results. According to the numerical simulation results, the velocity of wind in channel venturi is 3.9 times more than the channel entrance velocity. As a result of numerical analysis and experimental investigation, by using the new diffuser and the new fabricated wind channel, the wind velocity between blades of the microturbine is 4.5 times more than the inlet velocity of the wind channel.

1. Introduction

The wind is among the renewable resources in nature, used as a suitable resource to produce energy. Besides applications like grinding and water pumping [1], wind power is used to produce electricity. Establishing a wind farm needs lots of studies and spending high financial costs, and also it produces high noise pollution [2]. The potentiality of wind extractions is reviewed to achieve the clear overview of this new progressive ideas and the important configurations is accentuated. Most findings indicated that this energy recovery device converts wasted energy to a more profitable form by converting it to electricity, resulting in a rapid return on investment. Moreover, the enclosing the

* *Corresponding author.*
E-mail address: g.najafi@modares.ac.ir (Gholamhassan Najafi)

https://doi.org/10.37934/cfdl.14.6.116

output area of wind turbines for recovering energy enhances overall efficiency [3]. It is possible to use other ways to produce electricity using wind potential. One of the wind flows resources that is used to produce electricity is airflow resulting from automobiles movement in high-ways. When an automobile stare moving in a path, a low-pressure area is created at the end of it that the surrounding air moves towards this area after the automobile traverses due to pressure difference and to balance the pressure of the low-pressure area. This phenomenon occurs continuously with the automobile's movement in the movement path [4,5]. Air displacement to this low pressure creates airflow in movement path after automobile transverse, whose severity depends on automobile velocity, dimensions, and shape. To improve microturbines' performance, different studies have been done so far that some of them are reviewed. Allaei *et al.,* [6,7] used a new design called INVELOX, which removed turbine insertion in the highest and represented a solution for all wind energy challenges by aligning wind flow. Kazuhiko *et al.,* [8] constructed a microturbine with a wind lens. Their study results showed that the turbine equipped with lens production power is significantly higher in all states. In another study, Han *et al.,* [9] created an aperture between input and output flow that the results showed that system efficiency increases 66% to 73% in low velocities. Habali and Saleh estimated manufacturing wind microturbines' cost, the 33% design fan of normal turbines. Their results revealed that fan design microturbine has higher efficiency than normal wind microturbine during research on fan design microturbines. investigation of output power by changing the blade torsion angle performed by Zakaria *et al.,* [10]. The results showed that the most amount of output power is achieved in ±30° blade. In another study marine floating wind turbines with a lab model investigated. Marine wind turbines experiments aimed at collecting data and investigating challenges faced with them in experiments. The results showed that the information, online databases, and wind turbines design codes are a basis for comparison and are a criterion for real experiment data [11]. Based on the numerical simulation, a new study conducted on jet fans. This model was able to calculate all fan features, but a set of parameters and data is needed for a more accurate model, which helps the perfect validity of numerical models, but no study has been conducted in this field so far [12]. In another study on wind turbines, a wind turbine was modeled numerically. Also, the wind tunnel is used to validate experiments. Three wind velocity, turbine rotation velocity, and distance from wind turbine rotor parameters have been considered regarding the diversity of variables. So, a wind tunnel test experiment on a small scale was implanted, and the velocity was measured. Simulation investigated using three governing equations of Jensen, Larson, and Frandon. Comparing the theoretical and experimental results of Larson's mathematical model with experiments, data conformed to each other well. In contrast, Jensen and Frandon's mathematical models can identify the average and peak of velocity, respectively [13]. A new axial turbine named NACA0021 vertical axis turbine with by the straight blade studied numerically by using SST k-ω turbulent model. In this study, location, slope angle, and flexibility length parameters investigated. Simulation results showed that the power coefficient improve considering reinforcement flow (deflector). Means coefficient of the torque is 47.1% more in the presence of deflector rather than in the absence of deflector [14]. In recently study, compares the performance of NACA0018 and S1046 aerofoil profiles for a range of Speed Ratios (TSRs) and blade pitch angles. It has been found that the S1046 is less sensitive to changes in wind speed, and is thus, a superior choice for urban applications where the wind speed is comparatively low and varies a lot. Three bladed VAWTs of solidity 0.1 was modelled using Solidworks for this study. The CFD simulations were then performed in ANSYS Fluent, utilising the k-ω SST turbulence model. The model was validated at first before analysing the VAWT performance with the intended aerofoils. Key results indicate that increasing the TSR leads to increases in aerodynamic performances for nearly all cases, and especially so, for lower blade pitch angles. However, this study concludes that VAWT consisting of S1046 aerofoils at -2 degrees of blade pitch and operating at TSR 4 will provide the optimum performance. In another study, Tahzib *et al.,* [15]. In recently study, the current work is carried out to investigate the effect of combining PBCF and ducted on B-series and Kaplan-series propellers to discover further advantages of using the two types of energy saving device (ESD) tools. The study is carried out numerically using computational fluid dynamics (CFD) approach which is based on Reynolds-averaged Navier-Stokes Equations (RANSE) together with the use of k-epsilon turbulence modelling. Overall results show that the use of Kaplan-series with combination ducted and PBCF can even further improve the thrust, torque, and efficiency of propeller [16].

In recently study, wind turbines and consecutive turbines in highways investigated numerically to recover wind energy resulted from automobiles' movement in highways. Vertical axis turbines can recover wind energy, so the performance of three different rotors compared and investigated numerically using CFD technique. The results showed that the Banki rotor has better performance on the highway [17].

This paper presents a numerical validation study using the experimental data collected by the National Renewable Energy Laboratory (NREL). All the simulations are performed on the sequence S of the extensive experimental sequences conducted at the NASA/Ames wind tunnel with constant RPM and variable wind speeds. The results show close agreement with the NREL UAE experimental data. The CFD model captures closely the totality of the defining quantities. The shaft torque is well-predicted pre-stall but under-predicted in the stall region. The three dimensional flow and stall are well captured and demonstrated in this paper. Results show attached flow in the pre-stall region. The separation appears at a wind speed of 10 m/s near the blade root. For V>10m/s, the blade appears to experience a deep stall from root to tip [18].

In the present study three wind channels have been evaluated numerically and experimentally to operate electrical power from wind flow resulting from automobiles movement in highways. Wind channels have been designed to align wind flow to increase wind velocity. Wind channel number 1 include two air entries, which only absorbed wind from the automobile's movement on both sides of the traffic barrier. INVELOX design includes four air entries; the channel mechanism is in such a way that guides wind flow towards microturbine through a knee joint. Still, the dimensions of wind channel number 2 include eight 45° input air considering the designed traffic barrier and wind input and output flow is connected through a knee joint, which makes airflow inside the channel turbulent. Therefore, wind channel number 3 is designed like wind channel number 2 by removing input and output flow knee joint and based on the middle traffic barrier's present condition and the experimental results of measured wind velocity (resulted from automobile movement) in highways. Wind channel number 3 consists of eight 45° entries (measured maximum wind), which aligns wind flow resulted from automobiles movement from both sides of the middle traffic barrier and six other entries. This study's innovation is to minimize flow turbulence in wind channel venturi by the diffuser, which no study has been conducted in this field so far, and this matter has been studied in this research for the first time. The diffuser creates a low-pressure area inside the wind channel by reducing volume inside the wind channel. When diffuser is placed inside the channel at a suitable distance, it increases velocity in the channel neck with a steadier slope rather than channel without diffuser and minimizes airflow turbulence where wind microturbine is located, consequently, a steady velocity is created in the neck, and wind microturbine produces more power.

2. Experimental Setup

The fabricated device includes a wind channel, diffuser, and wind microturbine. Besides the mentioned items, the chassis of the wind channel was fabricated. This study used an environmental anemometer, propeller anemometer, and chassis to do primary experiments. An aluminum sheet with 0.0005 m thickness used to fabricate a wind channel, also wood used to fabricate diffuser to decrease weight and cost. Microturbine is placed at the proper distance with angle to the automobile. The goal was to determine the optimal value of horizontal distance and vertical distance in order to use maximum wind resulting from automobiles movement. Based on the optimal angle of microturbine placement to the road and the direction of automobiles movement, a chassis was designed and fabricated as shown in Figure 1.

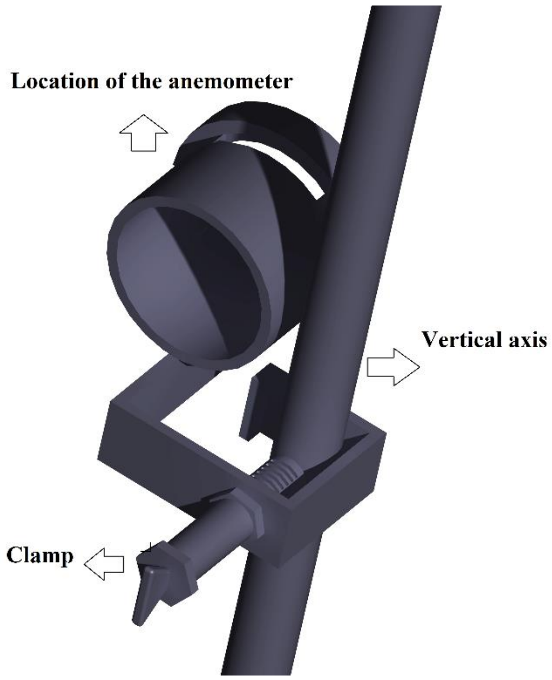


Fig. 1. Designed anemometer chassis

Different experiments were conducted to determine horizontal distance, vertical distance, and suitable placement angle on a smooth road. To segregate wind velocity resulted from automobiles transverse from environment wind velocity measured separately in each experiment. Wind flow velocity produced by automobile measured using an impeller anemometer model S-AR856. Moreover, environment wind velocity measured using a cup anemometer model LUTRON AM-4220 manufactured by Lutron Company in Taiwan, the accuracy of anemometer measurement was 0.01. The average automobile velocity was 55 km/h, height was 2.55 m, automobile width was 1.94 m, and the environmental wind velocity was 0.1-0.2 m/s in all experiments. Table 1 shows independent and dependent variables in which selecting independent variables was based on measured wind velocity resulted from automobiles movement.

Table 1
Independent and dependent variables related to design experiment

Variables		Unit	Symbols	Levels of each factor (encoded values)		
				-1	0	1
Independent variables	Vertical distance	m	A	0.5	1	1.5
	The horizontal distance	m	B	1	2	3
	Placement angle	degree	C	0	45	90
Dependent variable	Production power	W	Y		_	

Measuring wind velocity resulted from the automobile movement consists of 81 experiments considering three variables of vertical distance, horizontal distance, and anemometer placement angle in three levels. Performing these experiment levels were difficult and have less accuracy considering experiment conditions, so using a reliable way to reduce experiment level is necessary. In order to reduce the experiments, Response Surface Methodology (RSM) method was used as an analytical and optimization method in this experiment. RSM contains a group of mathematical and statistical technologies that can achieve an optimal condition in complicated systems. This method searches a relationship between independent variables and dependent variables. Design Expert 2016 was used to prepare the experiment matrix based on the RSM. Box Behnken experiment design method as powerful design experiment of RSM technique used to design considered experiments.

In the next step, wind channel fabricated to align airflow, reduce flow turbulence, and increase wind velocity resulted from automobiles movement. in order to improve the power production from the microturbine, different parameters related to the installed location of the wind channel in the highways should be considered including the vertical distance, horizontal distance, and suitable angle. The traffic barrier's width is 1.2 m, and its height is 0.84 m from the surface of the ground. An aluminum sheet with 0.5 mm thickness was used as the material of the wind channel. The wind channel has been made up of two corn with 1.2 m and 0.3 m diameter. The second corn has been located 0.660 m lower than the first one. The third corn with 0.3 m and 0.15 m diameter was connected to the second corn; in each corn, volume loss increases air velocity. A framework fabricated to maintain wind channel. Besides maintaining the wind channel's corns and stability, it is stable enough against rain and storm in the middle traffic barrier. Furthermore, in order to minimize airflow turbulence inside the channel, a new diffuser designed and fabricated as shown in Figure 2.

(a) (b)

Fig. 2. (a) Fabricated wind channel, (b) Fabricated diffuser

After optimizing wind channel, a proper wind microturbine used to convert wind flow to electricity. Regarding the conducted studies, fan design microturbine is more economical and simpler considering manufacturing, also this microturbine produces more electrical power rather than airfoil model microturbine. the maximum rotation of microturbine was 1300 rpm. Diameter of the turbine was 0.12 mand the weight of the microturbine was 0.181 kg.

3. Computational Fluid Dynamics

This study used CFD method to stimulate fluid movement in wind channel and study the best placement area for diffuser and wind microturbine. ANSYS CFX-2018 edition 64x used to numerical study. ANSYS CFX software is capable of modeling 2D and 3D flows. This software has been based on the finite volume method, which is a very strong and suitable method in computational fluid dynamics methods.

3.1 Geometries

3D geometry of three wind channels was created, in SOLIDWORKS 2016x64 edition. The width and entrance channel from the ground's surface were measured based on the dimensions of standard traffic barrier and channel height. Airflow entrance angle designed based on measured wind velocity results. Figure 3 shows wind channel geometries, with a total height of 1.5 m, width of 1.2 m, and height from the ground surface of 0.84 m. Another part of the wind channel that should be created is diffuser. Moreover, in order to improve the efficiency of wind microturbine, diffuser 's distance inside the wind channel should be considered (Figure 3(e) and Figure 3(f)).

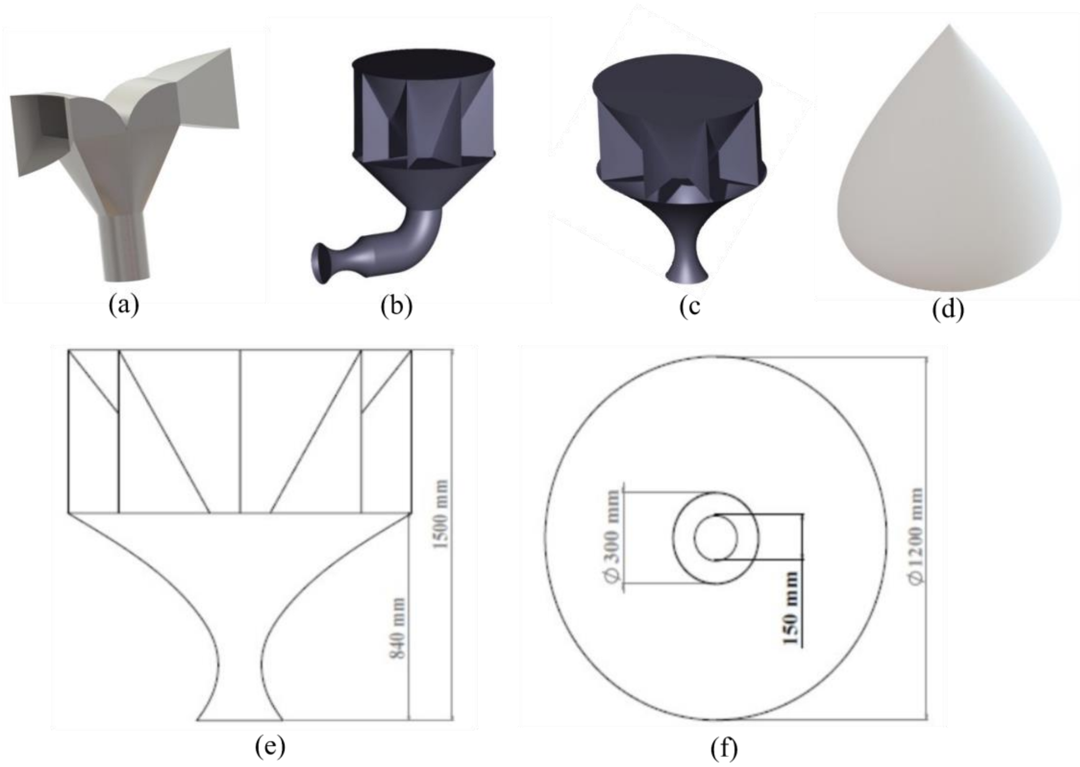


Fig. 3. Geometry of the created channels: (a) wind channel 1; (b) wind channel 2; (c) wind channel 3; (diffuser); (e) Accurate dimensions and wind channel 3 geometry; (f) configuration by eight 45° entries

In the next section, geometry of the microturbine created. The number of blades and the angle of root placement and the angle of blade tip placement to the horizon were among the items that were investigated in designing microturbine. Figure 4 shows the fan design microturbine configuration.

Fig. 4. Fan design microturbine geometry

The microturbine installed in wind channel venturi. Fan design microturbine investigated based on three specified variables (the number of blade s, the angle of connection to the hub, and the angle of blade tip) (Stewart & Muskulus, 2016). Figure 5 shows the perfect mechanism of the new wind channel. The new wind channel named wind channel number-3 includes 5 main parts including (1) eight 45° entries; (2) rectifier corn of wind flow; (3) diffuser; (4) fan design microturbine; (5) wind flow diffuser.

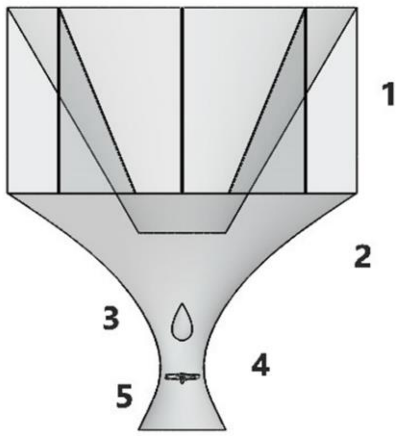


Fig. 5. Wind channel 3 transfer system schema with its main components, (1) entry), (2) flow rectifier, (3) diffuser, (4) wind microturbine, (5) diffuser

3.2 Mesh Generation

In the next section, the created geometry of the wind turbine should be discretized. Mesh structure shows simulation resolution and accuracy. Thus, the type and number of considered elements have to be used accurately. the independence of the mesh has to be investigated to ensure the accuracy of the results. To discretize wind channel without a diffuser, mesh size has to be changed, and analysis is done, and the output velocity has to be recorded. The effect of different mesh size on output wind velocity values is less than 1%. Therefore, in this size of mesh, the effect of mesh size on the result is negligible. Finally, the numbers of meshes in the wind channel is equals to 10607868 and the mesh number of the wind channel with diffuser is equals to10870324. For a turbine with 0.001 mesh size, the number of mesh is equals to 1094899. Figure 6 show meshing independency.

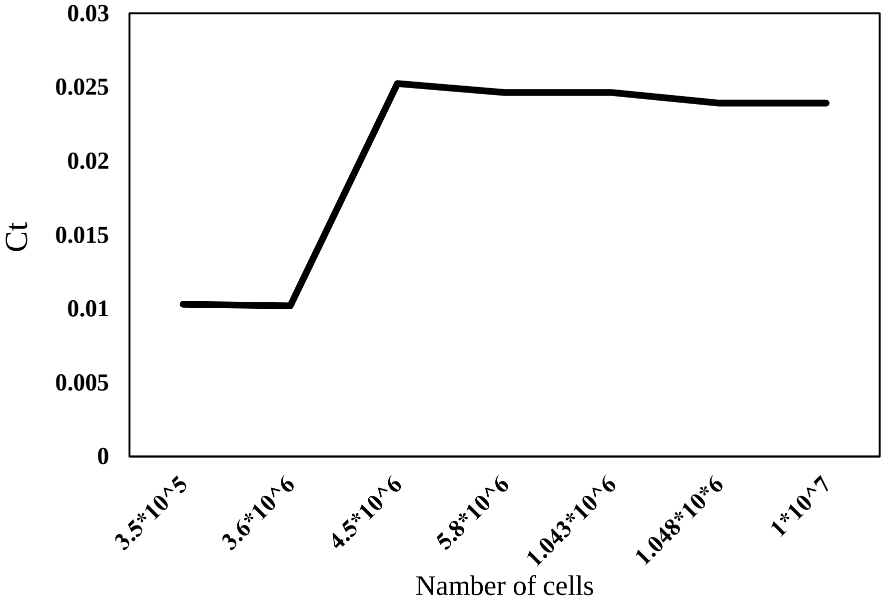


Fig. 6. Mesh independency

3.3 Computational Schemes

ANSYS CFX-2018 edition 64x software was used to numerical study. Reynolds Averaged-Navier Stokes (RANS) model has been used by different researchers to simulate turbulent flow. After decomposing all values, the continuity equation and equation of motion are considered a group equation. Eq. (1) and Eq. (2) are known as RANS equations, which are the equal form of momentum Navier-Stokes equations with meantime values for velocity and pressure [17].

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial t}(\rho ui) = 0 \tag{1}$$

where ui is the meantime of velocity vector (m/s), ρ is density (kg/m^3), µis fluid viscosity (m^2/s), ijδ is Kronecker delta (δ=0 when i≠j when i=j), t is time (s), and xi is spatial coordinate.

In the Eq. (2), (-ρủi¯ủj) shows reynolds stresses which have to be modeled to complete Eq. (1). ANSYS CFX software provides different turbulent RANS models which include k-ω, k-ε, Reynolds Stress Model (RSM), turbulent SST mode, etc.

$$\frac{\partial}{\partial xj}(\rho uiuj) + \frac{\partial}{\partial t}(\rho ui) = -\frac{\partial p}{\partial xi} + \frac{\partial}{\partial xj}\left[\mu(\frac{\partial ui}{\partial xj} + \frac{\partial uj}{\partial xi} - \frac{2}{3}\frac{\partial ui}{\partial xi}\right] + \frac{\partial}{\partial xj}(-\rho uiuj) \tag{2}$$

ANSYS CFX software provides different turbulence RANS models that computations cost and prediction needed by the considered flow have to be regarded to select a turbulence model to do numerical simulation. One of the flow characteristics is rotation, so friction component between experimental results and simulation results is an important part in turbulence.

Solution convergence test of three-dimensional Reynolds Navier-Stokes (RAN-S) equations solved by numerical mean (RAN-S) with a two-equation model with second-order accuracy (Allaei & Andreopoulos 2014).

3.4 Computation Domains and Boundary Conditions

Two computational areas including a cube for wind channel and a corn for rotor have been modeled to simulate wind resulting from automobile movement in highway and wind inside channel. System boundary and primary conditions are presented in Table 2.

Table 2
System boundary and primary conditions

Wind channel	Symbols	Unit	amount
Wind channel input	u	(m/s)	0
	v	(m/s)	5
	w	(m/s)	0
	T	◦C	25
Wind microturbine input	u	(m/s)	0
	v	(m/s)	20
	w	(m/s)	0
	T	◦C	25
Turbulence	Medium Intensity	%	5
Wind channel output	P	(Mpa)	0
Wind microturbine output		P(Mpa)	0
Channel wall	u	(m/s)	0
	v	(m/s)	0
	w	(m/s)	0
Wind microturbine wall	u	(m/s)	0
	v	(m/s)	0
	w	(m/s)	0

Cube input velocity considered 5 m/s and corn input velocity (velocity in channel venturi) considered 20 m/s. This channel has eight 45° entries as well as velocity internal surfaces in boundary layer which is zero and output pressure considered zero in cube and corn. Channel made up of galvanized sheet (high stability against wind and rain), diffuser made up of wood (rigid material and easy to shave) and wind microturbine made up of compact plastic. Additional boundary conditions have been presented in Table 3. In this study the dimensions of surrounding air area cube considered 1.5 *1.5 *2 m, also the finite volume of corn diameter microturbine (corn) considered 0.15 *0.3 *0.24 m.

3.5 Model Validation

Considering that wind channel 3 has been constructed for the first time and no similar study has been conducted so far, the system is validated by comparing wind velocity results in experimental samples with channel wind velocity in numerical simulation after constructing the winch channel. So, a fan placed in front of the wind channel and wind velocity is measured at the entrance off-channel and mean velocity calculated at the channel entrance, then wind velocity calculated in channel exist and the ratio of output to input compared with simulation data. The constructed diffuser was also inserted for alignment at a suitable distance inside the wind channel, diffuser test results compared with numerical simulation results.

4. Results and Discussion

4.1 Statistical Analysis

Table 3 shows the software's statistical table for different models to model wind velocity. According to Table 3 results that use of quadratic model, there is close conformity between results obtained experimentally and resulted predicted by the model with R^2 = 0.92, which proves the developed model's validity. Analysis of variance results represented to evaluate the effect of independent variable on dependent one (production power). It is obvious that the used model to investigate the effect of the independent variable on the dependent one is significant, and it is possible to predict the effect of the independent variable on the dependent one using a quadratic polynomial equation.

Table 3
Statistical table summary

Statistical Model	Std. Dev.	Coefficient of Variance (CV) %	R^2	Adjusted R^2	Predicted R^2	
Linear	0.65	0.098	-0.11	-0.29	8	
2FI	0.74	0.099	-0.44	-1.17	13.4	
Quadratic	0.17	0.96	0.92	0.83	1	Suggested
Cubic	0.19	0.97	0.9	0	+	

Figure 7 shows the diagram of the predicted model by software and the actual model. Figure 7 shows the linear relationship among these values, demonstrating the correlation between experimentally obtained results and values out of the regression model. Figure 7 real data conform to predicted data, which proves the developed model's validity, consequently modeling by the software has been valid. The model has predicted real data with high accuracy. R^2=0.92 value reveals that 92% of the dependent variable changes depending on the independent variable, and only 8% of the dependent variable is not explainable using the independent variable. Table 4 shows the optimization experiments level by RSM method.

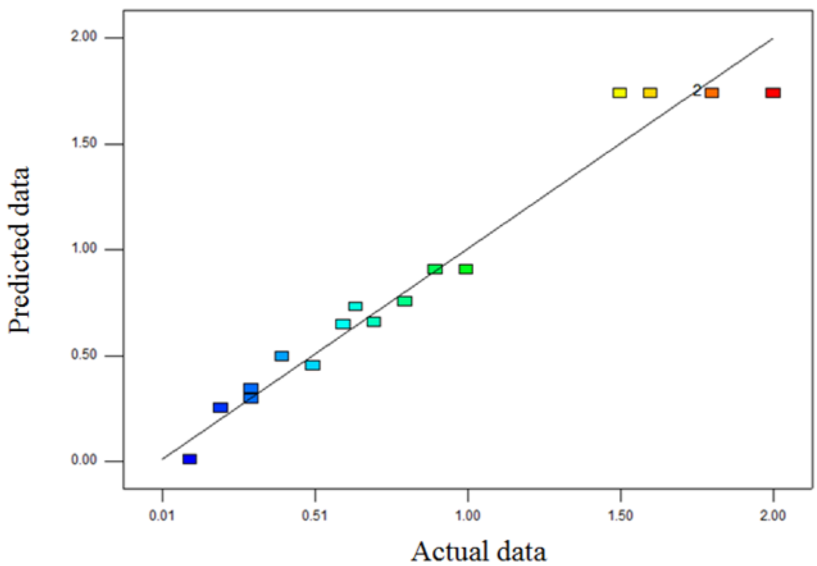


Fig. 7. Conforming real data to predicted data in RSM

Table 4
Experiments specified by RSM according to a maximum of velocity

NO	Vertical distance (m)	The horizontal distance (m)	Placement angle (degree)	Velocity (m/s)	Correctness probability
1	1.91	0.95	45	1.75	0.93
2	1.91	0.95	45	1.75	0.93
3	1.92	0.95	45	1.75	0.93
4	1.91	0.94	45	1.75	0.93
5	1.91	0.96	45	1.74	0.93

In order to optimize the process of wind velocity production, it is necessary to obtain a combination of independent variables that wind velocity is at its maximum level. Therefore, RSM introduced five experimental levels considering the maximum of wind velocity that the design is 93% correct at 1.91 m horizontal distance, 0.95 m vertical distance and 45° angle and are the maximum of wind velocity and absorption angle are obtained and also the results showed that independent variables do not influence each other reciprocally. In order to investigate the correctness of the conducted experiments, the level off experiments related to maximum and minimum wind velocity repeated in highway. Results related to maximum and minimum wind velocity 3 experiments level were repeated. Thus, according to Table 4 results and traffic barrier conditions, wind channel built in 1.5 m height, 1.2 m width (traffic barrier width) and eight 45° entries and 0.84 m height from ground dimensions, two 45° entries considered for the wind resulted from automobiles movement and six 45° entries to increase environment wind velocity for wind channel.

4.2 Simulation Results

Stimulating airflow inside the wind channel aims to select a suitable wind channel to increase wind velocity, reduce turbulence flow inside the wind channel, place the diffuser at a suitable distance inside the channel, and use suitable wind micro turbine use maximum wind micro turbine power to produce electrical power production as shown in Figure 8.

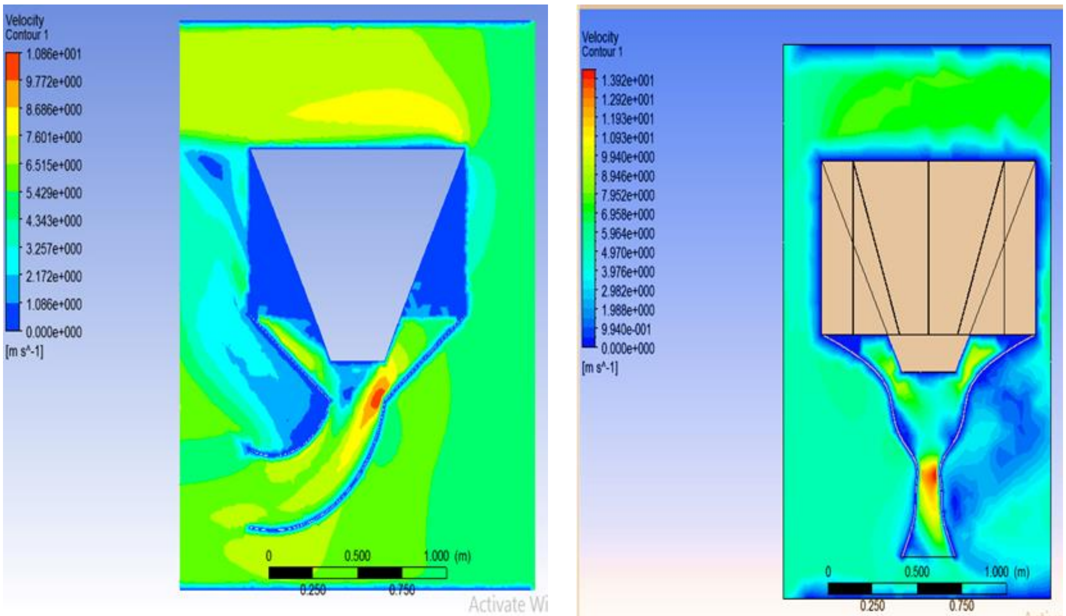


Fig. 8 The results of wind channels 2 and 3 in equal conditions

In Table 5 analysis result of three wind channels considering the velocity of output wind and the turbulence inside the wind channel presented. The maximum wind velocity in wind channel 3 reached more than 4 times, and in wind channel 2 reached more than 2 times rather than the input velocity. Wind channel 2 in specified dimensions (traffic barrier) is like INVELOX design but enjoys less turbulence inside the channel due to small dimensions, small dimensions rather than INVELOX design makes wind channel 2 turbulent, and input and output flow are related to each other through knee joint path, and also wind velocity reaches two times of its velocity. In contrast, wind channel 3 is similar to wind channel 2 in dimensions and includes eight 45° entries, but input and output flow inside the channel is straight and without curvature. Wind channels 1, 2, and 3 are equally based on the Table 5 and traffic barriers. Selecting a wind channel based on simulation results that wind velocity in the neck (microturbine location) is more than 4 times rather than input airflow and less turbulence rather than wind channels 1 and 2.

Table 5
Investigating wind channels 1, 2, and 3 analysis

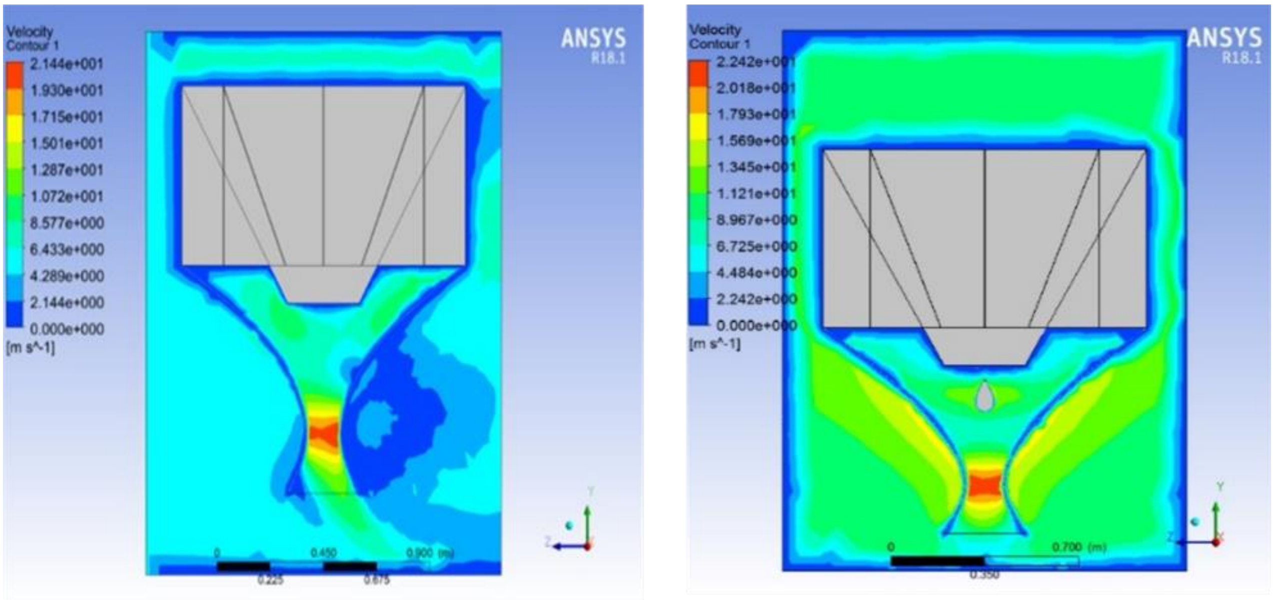
Wind channel	Results of channels fluid simulation analysis
Wind channel 1	Including two 45° entries, only absorbs wind resulted from automobiles movement and environment wind is not absorbed in all directions, and turbulence is high inside the channel.
Wind channel 2	Absorbs wind resulted from automobiles movement as well as environment airflow. It includes eight 45° entries, but due to curvature in the vertical and horizontal axis and small size rather than INVELOX, wind flow inside the channel is turbulent. Wind velocity is doubled like INVELOX design considering the dimensions needed in this study (traffic barrier dimensions and velocity measured by anemometer)
Wind channel 3	Including eight entries that absorb wind resulted from automobiles movement and environment wind, the system is designed continuously. Without curvature, less turbulence than wind channels 1 and 2, velocity has reached more than 4 times than the velocity of input velocity.

After selecting wind channel 3 as a suitable wind channel in the highway, determining the suitable distance from diffuser inside the channel is necessary to minimize flow turbulence inside the channel. Table 6 shows the effect of diffuser distance from the surface ground on wind velocity and consistency.

Table 6
Changing diffuser distance to make wind velocity consistent inside channel

Diffuser distance from ground surface (m)	Wind velocity in the place of inserting wind micro turbine (m/s)	Investigating consistency in the place of inserting micro turbine
0.34	24.6	Inconsistent
0.35	23.8	Inconsistent
0.36	22.1	Inconsistent
0.37	17.76	Inconsistent
0.4	14.3	Inconsistent
0.43	20.7	Inconsistent
0.45	21.3	Consistent

Figure 9a shows wind velocity changes inside the channel without diffuser, and Figure 9b shows wind velocity inside the channel with the diffuser. Using the diffuser to minimize turbulence inside channel was first investigated in this study; if diffuser exists, wind velocity will increase with a more consistent slope. As a result, the output velocity is more consistent. In fact, diffuser diffuses wind flow regularly and reduces win flow turbulence by reducing channel neck volume and placing at a suitable distance.



(a) (b)

Fig. 9 (a) Wind velocity changes inside channel without diffuser, (b) wind velocity inside the channel with diffuser

To validate the velocity simulated wind channel, wind velocity measured in highway, wind velocity recorded in 4 seconds, and entered simulation software as channel input and Figure 10 shows velocity simulation results in channel neck. The wind velocity mean is 5.62 m/s in highway and the mean of velocity in neck is 21.8 m/s. Also, the least ratio of output velocity to input velocity in channel 3.7 is 6.7 m/s in input velocity. According to the results, in velocities, less than 6 m/s win channel output velocity reaches more than 4 times rather than the input velocity to the wind channel in simulation that is flow turbulence happens inside wind channel in high input velocity which not only increases wind velocity but also produces less power due to the turbulence. Simulation results show that velocity in channel neck is 3.9 times more than input velocity. On average, it also channels output velocity and measured input and simulation results compared with each other to validate the simulated model. This experiment done by placing a big fan in front of the wind channel and wind velocity recorded in channel entrance and exist, experimental results of output to input velocity was 3.5 that with 10% difference from simulation results enjoys acceptable validity also in order to validate the diffuser, the diffuser placed in different distances inside channel and the results showed that at 0.45 m distance from ground surface, wind velocity in channel neck is consistent.

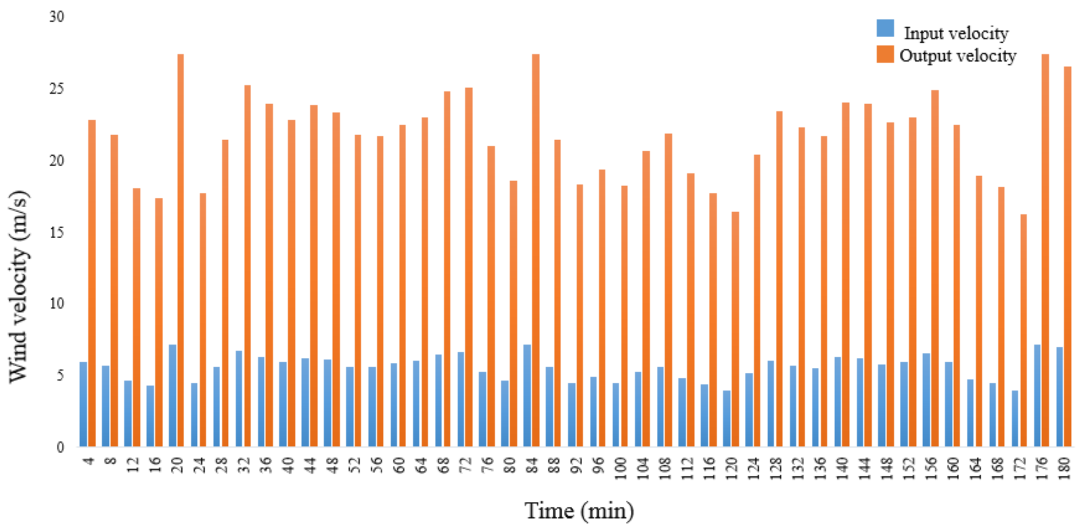


Fig. 10 Output and input velocity in the channel in simulation

4.3 The Results of Fan Design Wind Micro Turbine Optimization

The effect of the number of micro turbine blades on rotation velocity and, as a result, more power production investigated by changing the number of blades and fixing root and blade tip angle in 20 m/s input velocity. Velocity among blade s recorded according to the Table 7.

Table 7
The effect of the number of blades on wind velocity in micro turbine

The number of blades	The angle of blade tip to the horizon (degree)	The angle of root to the horizon (degree)	Velocity min and max) m/s) in analyzing CFX
3	15	40	1.14-46.28
5	15	40	39.43-59.1
7	15	40	69.2-76.1

Table 8
The effect of changing root and blade tip angle on wind max and min velocity in micro turbine

No	The number of blade s	The angle of blade root to the ground (degree)	The angle of blade tip to the ground (degree)	Maximum velocity (m/s)
1	7	74	14	46.28
2	7	65	5	61.65
3	7	45	15	72.65
4	7	45	25	76.5
5	7	35	23	77.64
6	7	30	20	80.5
7	7	30	15	84.36
8	7	30	10	84.67
9	7	30	7	85.1
10	7	30	5	85.67
11	7	25	15	86.82
12	7	25	5	89.85

According to the Table 8, the result shows that a 7-blade micro turbine with 20 m/s input velocity reaches velocity among blade s to 76.1 m/s. After selecting a 7-blade micro turbine, the angle of root and blade tip is changed, and wind velocity results are compared with each other. Angle change is considered based on the velocity results in each stage (max velocity). Table 8 showed the results of the effect of root and blade tip angle. Considering the results of Table 8 the maximum wind velocity in 7-blade micro turbine with 25° root angle and 5° tip angle to horizon was obtained. The most important reason for designing wind microturbine is the ratio between output and input velocity; the numerical simulation results showed that wind input velocity is 20 m/s and wind velocity among blade s is 89.85 m/s that is 4.5 times more than the input velocity.

5. Conclusion

The purpose of this study is to use is to wind energy due to the movement of vehicles, which increases the wind speed and reduces the turbulence of wind flow in the wind channel to produce maximum power. The most important results of this study are summarized as follows:

i. The feasibility of producing electrical energy from wind resulted from automobiles movement in highways investigated by chassis, capable of regulating height and angle. RSM obtained the reduction of experiment levels related to three horizontals, vertical, and placement angle variables. RSM could estimate the maximum velocity in vertical distance, horizontal distance, and micro turbine proper placement angle from the road well and with high accuracy and utilizing second order polynomial equation as 0.95 m, 1.95 m, and 45° respectively. Also, the mean velocity resulted from automobiles' movement in highways measured as 5.62 m/s.
ii. In the first stage, three wind channels designed considering the dimensions of traffic barrier and maximum wind velocity in RSM that wind channel 1 had two 45° entries and wind channel 2 had eight 45° entries in INVELOX design but in needed dimensions and channel input and output are related to each other through a knee joint path and velocity in the neck is more than two times rather than the velocity in the entrance and is turbulent. Wind channel 3 has eight 45° entries. The input and output flow are related to each other by removing the knee joint and are less turbulent than wind channels 1 and 2, and velocity in the neck reached more than 4 times rather than velocity in channel entrance. Velocity in wind channel 3 (neck) is 3.5 times more than input velocity on average while wind velocity is two times more than input velocity in INVELOX design which due to more input rather than INVELOX channel, input and output are connected without knee joint and wind resulted from automobile movement is created at a close distance from the channel.
iii. In the second stage, after selecting wind channel 3 as a suitable wind channel, placing diffuser at a suitable distance inside the channel to minimize wind flow turbulence is of high importance because reducing turbulence and increasing wind velocity increase electrical power, so distance 0.45 m from the ground surface along z axis is achieved as optimal diffuser distance by changing diffuser distance in CFX simulation software.
iv. In the third stage, fan design micro turbine is selected as wind micro turbine, and the number of blade s, root angle and tip angle are investigated to produce more power production. According to CFX results, fan design micro turbine with 7 blade s and 25° and 5° root angle and tip angle reaches more than 4.5 times more than input velocity to micro turbine rather than velocity horizon

References

[1] Sørensen, Jens Nørkær. "Aerodynamic aspects of wind energy conversion." *Annual Review of Fluid Mechanics* 43, no. 1 (2011): 427-448. https://doi.org/10.1146/annurev-fluid-122109-160801
[2] Grant, A. D., and N. J. Kelly. "The development of a ducted wind turbine simulation model." (2003): 407-414.
[3] Ismail, Ainaa Maya Munira, Zurriati Mohd Ali, Kamariah Md Isa, Mohammad Abdullah, and Fazila Mohd Zawawi. "Study On the Potentiality of Power Generation from Exhaust Air Energy Recovery Wind Turbine: A Review." *Journal*

of Advanced Research in Fluid Mechanics and Thermal Sciences 87, no. 3 (2021): 148-171. https://doi.org/10.37934/arfmts.87.3.148171

[4] Tian, Wenlong, Zhaoyong Mao, Baoshou Zhang, and Yanjun Li. "Shape optimization of a Savonius wind rotor with different convex and concave sides." *Renewable energy* 117 (2018): 287-299. https://doi.org/10.1016/j.renene.2017.10.067

[5] Al-Bahadly, Ibrahim H., ed. *Wind turbines*. BoD–Books on Demand, 2011. https://doi.org/10.5772/643

[6] Allaei, D., J. E. Gonzalez, A. S. Sadegh, Y. Andreopoulos, and D. Tarnowski. "INVELOX bringing noise & vibration issues to the ground level." In *Second international conference on wind turbine noise & vibration, Hamburg, Germany*. 2013.

[7] Allaei, Daryoush, and Yiannis Andreopoulos. "INVELOX: Description of a new concept in wind power and its performance evaluation." *Energy* 69 (2014): 336-344. https://doi.org/10.1016/j.energy.2014.03.021

[8] Toshimitsu, Kazuhiko, Hironori Kikugawa, Kohei Sato, and Takuya Sato. "Experimental investigation of performance of the wind turbine with the flanged-diffuser shroud in sinusoidally oscillating and fluctuating velocity flows." *Open Journal of Fluid Dynamics* 2, no. 04 (2012): 215. https://doi.org/10.4236/ojfd.2012.24A024

[9] Han, Wanlong, Peigang Yan, Wanjin Han, and Yurong He. "Design of wind turbines with shroud and lobed ejectors for efficient utilization of low-grade wind energy." *Energy* 89 (2015): 687-701. https://doi.org/10.1016/j.energy.2015.06.024

[10] Zakaria, Mohamed Y., Daniel A. Pereira, and Muhammad R. Hajj. "Experimental investigation and performance modeling of centimeter-scale micro-wind turbine energy harvesters." *Journal of wind engineering and industrial aerodynamics* 147 (2015): 58-65. https://doi.org/10.1016/j.jweia.2015.09.009

[11] Bingol, Ferhat. "Feasibility of large scale wind turbines for offshore gas platform installation." *AIMS Energy* 6, no. 6 (2018): 967-978. https://doi.org/10.3934/energy.2018.6.967

[12] Król, Aleksander, and Małgorzata Król. "Study on numerical modeling of jet fans." *Tunnelling and Underground Space Technology* 73 (2018): 222-235. https://doi.org/10.1016/j.tust.2017.12.024

[13] Brusca, Sebastian, Rosario Lanzafame, Fabio Famoso, Antonio Galvagno, Michele Messina, Stefano Mauro, and Mauro Prestipino. "On the wind turbine wake mathematical modelling." *Energy Procedia* 148 (2018): 202-209. 2018;148:202-9. https://doi.org/10.1016/j.egypro.2018.08.069

[14] Wong, Kok Hoe, Wen Tong Chong, Sin Chew Poh, Yui-Chuin Shiah, Nazatul Liana Sukiman, and Chin-Tsan Wang. "3D CFD simulation and parametric study of a flat plate deflector for vertical axis wind turbine." *Renewable energy* 129 (2018): 32-55. https://doi.org/10.1016/j.renene.2018.05.085

[15] Tahzib, Teeab, Mohammed Abdul Hannan, Yaseen Adnan Ahmed, and Iwan Zamil Mustaffa Kamal. "Performance Analysis of H-Darrieus Wind Turbine with NACA0018 and S1046 Aerofoils: Impact of Blade Angle and TSR." *CFD Letters* 14, no. 2 (2022): 10-23. https://doi.org/10.37934/cfdl.14.2.1023

[16] Adietya, Berlian Arswendo, I. Ketut Aria Pria Utama, and Wasis Dwi Aryawan. "CFD Analysis into the Effect of using Propeller Boss Cap Fins (PBCF) on Open and Ducted Propellers, Case Study with Propeller B-Series and Kaplan-Series." *CFD Letters* 14, no. 4 (2022): 32-42. https://doi.org/10.37934/cfdl.14.4.3242

[17] Tian, Wenlong, Baowei Song, and Zhaoyong Mao. "Numerical investigation of wind turbines and turbine arrays on highways." *Renewable Energy* 147 (2020): 384-398. https://doi.org/10.1016/j.renene.2019.08.123

[18] Tahzib, Teeab, Mohammed Abdul Hannan, Yaseen Adnan Ahmed, and Iwan Zamil Mustaffa Kamal. "Performance Analysis of H-Darrieus Wind Turbine with NACA0018 and S1046 Aerofoils: Impact of Blade Angle and TSR." *CFD Letters* 14, no. 2 (2022): 10-23. https://doi.org/10.37934/cfdl.14.2.1023