Iranica Journal of Energy and Environment
Journal Homepage: www.ijee.net
IJEE an official peer review journal of Babol Noshirvani University of Technology, ISSN:2079-2115

Investigation of Geometric Dimensions and Dust Effects on Invelo Wind Deflectors Performance

M. Bagheri*, I. Mirzaee, M. Khalilian, V. Mousapour

Department of Mechanics, Faculty of Technical Engineering, Urmia University, Urmia, Iran

PAPER INFO

Paper history:
Received 24 May 2023
Accepted in revised form 19 June 2023

Keywords:
Computational fluid dynamics
Invelox
Residential building
Wind deflector

ABSTRACT

The present study simulates Invelox in a three-dimensional and stable way. The flow regime is turbulent flow and an unorganized grid with 350000 cells was utilized. This work has studied the modeling of Invelox with conventional dimensions and four different sizes in the form of four modes for use in a residential building. The numerical data with an error of less than 6% are in good agreement with the available experimental and analytical data. The results show that considering the average velocity of mode 2 with a velocity of 6.54 m/s and a 5% difference from the other two modes, it can be operated in a residential building. It is worth noting that in this investigation, in addition, the effect of dust on the turbine performance was evaluated. The results represent that the oscillation frequency of the blades increases with the increase of the rotational speed. In the case of not considering dust particles on blades, this amount increases by 25%, while considering dust particles with an amount of 0.1%, it increases up to 300%, and this can cause irreparable damage to the turbine as well as the power generation system.

***doi**: 10.5829/ijee.2024.15.01.05*

NOMENCLATURE

A	Area (m^2)	u	Velocity x-component (m/s)
B	Bulck forces (N)	$\bar{u}$	The mean value of velocity
D	Diameter(m)	u'	Fluctuations
p	Pressure (Pa)	v	Velocity y-component (m/s)
Q	Flow rate (m^3/s)	w	Velocity w-component (m/s)
R	Radius (m)	**Greek symbols**	
t	Time (s)	ρ	Density (kg/m3)

INTRODUCTION

The use of energy has increased significantly in the world so fossil energy sources will no longer be the answer to the world's energy for evolution, development, and survival in the coming centuries. Favorable environmental conditions in buildings are due to excessive energy consumption in buildings compared to other sectors; it is of particular importance by reducing fuel consumption, increasing the quality of construction, and buildings, and most importantly, reducing fossil fuel consumption; our main goal is to optimize consumption. Energy in the building is in charge. Wind turbines in hot and dry and hot and humid areas can provide natural ventilation and passive cooling, which are used a lot in the countries of the Middle East region. In the distant past, windmills that worked with wind energy were used. The most important arrangements made in Iranian architecture for wind energy included windmills, reservoirs, windmills, and shabadan. Today, wind energy is used to rotate electricity turbines and produce electricity, the cost of electricity produced by wind turbines is zero, except for the cost of maintenance. The wind is an extensive source of energy and the cost of using wind power is very

---

*Corresponding Author Email: mohadesehbagheriaf@gmail.com
(M. Bagheri)

Please cite this article as: M. Bagheri, I. Mirzaee, M. Khalilian, V. Mousapour, 2024. Investigation of Geometric Dimensions and Dust Effects on Invelo Wind Deflectors Performance, Iranica Journal of Energy and Environment, 15(1), pp.48-55. Doi: 10.5829/ijee.2024.15.01.05

low, and since wind turbines are located near the place of consumption, they reduce operating costs and network losses. Wind turbines' main problems include production uncertainty [1, 2].

Many experimental and numerical works have been done in connection with this issue. For example, Göltenbott et al. [3] used multiple rotors instead of one rotor in a channel, and this significantly increased turbine performance. In such a way, the power increased, and compared to one rotor, it increased up to 5% for two rotors and up to 9% for three rotors. In 2018, Gavade et al. [4] investigated the flow in the Invelox turbine and concluded that the production power of the Invelox system at a free flow wind velocity of 6 m/s is about 5 to 6 times the production power of traditional turbines of the same size. In 2019, Anbarsooz et al. [5] investigated Invelox numerically. From four turbulence models k-ω SST and k-Ɛ RNG, k-Ɛ Realizable, and k-Ɛ Standard, they obtained the velocity in the Innolux venturi. In 2020, Venkatramakrishnan et al. [6] conducted a study on low-velocity wind turbines, which they envisioned as having great potential for energy harvesting at low wind velocities with a significant potential improvement in energy conversion efficiency. In his work, several potential areas of research in the field of low-velocity wind turbines were also mentioned [6].

In 2013, Allaei and Anderpoulos [7] modeled Invelox and investigated the flow field. They examine the flow field when the actual turbine is located. Inside the Invelox before the turbine and after the turbine is the outlet, which usually creates the exhaust to match the inlet flow and provide thrust for the inlet flow. In the calculations that were made, the change of the incoming flow and the incoming wind blowing in different directions were investigated. So, they concluded that it is possible to receive the wind, concentrate and accelerate it, and also increasing the wind velocity is effective in the output power [7]. In 2014, Allai and Anderpoulos [8] found a method for the problem of low efficiency in low-velocity flows by researching wind turbines and, inspired by the wind turbines of Yazd, they designed a structure that achieves good results with the airflow and its increase in the venturi. The way of designing and building wind deflectors in Yazd city is similar to Invelox, which has many advantages for the energy production market by removing super propellers on top of giant columns. In these turbines, which was accompanied by the removal of super propellers on top of the giant columns, the wind is absorbed, directed, and concentrated by the wind deflector, so its velocity increases, and turning the blades of the wind turbine with a much shorter length than the conventional models produce electricity [8]. In 2015, Kumar et al. [9] investigated the effect of using the Invelox system and by putting it in a supersonic wind tunnel and sending smoke flow to investigate the flow lines in the system and the effect of increasing the velocity of the inlet airflow on the output power of the turbine. So, they concluded to compare the Invelox turbine with other modern wind turbines in a similar condition and stated that the Invelox turbine has a higher efficiency in a similar condition [9]. In 2015, Allaei et al. [10] conducted a laboratory investigation of the Inulox system and compared the computational fluid dynamics data with the output result and obtained a similar result. They placed a turbine in the Inulox system and with laboratory investigations of the turbine effect and the optimal number of turbines in the system They obtained the result that 3 turbines in the Invelox system increase the efficiency compared to the case where 1 The turbine is in Invelox system, it doubles 2.2 times. That is, to increase the output power of Invelox, instead of using one turbine in the venturi, they have used several turbines [10]. In 2015, during 8 days of investigation, Khadka [11] managed to match the Involx turbine with a series of traditional turbines, in which case the average production power increased almost three times. In 2016, Abadi et al. [12] investigated a 5 MW turbine that was equipped with Invelox technology and obtained a series of results in which the average annual wind velocity is 4.11 m/s in Chabahar, Sistan and Baluchistan province, and the average annual energy production of 905-gigawatt hours and the cost of energy is 2.3 cents per kilowatt hour. In 2017, Georgescu et al. [13] concluded that the free flow velocity is equal to 6.7 m/s with the numerical analysis of the flow in the Invelox turbine throat, which increases the velocity in the throat up to two times. In 2017, Anbarsooz et al. [14] defined a series of basic concepts such as the geometry of the Invelox and the dimensions, and while a larger amount of air entering the Invelox exited from the other side and did not enter the venturi. Numerically, geometrical parameters such as the entrance area and the radius of the lower part of the cone were investigated at different wind velocities. They concluded that the change of the radius of the lower part of the cone (Dd) does not have a great effect on the operation of the invelox, while most of the area of the inlet part has a very good effect. It has the function of Invalu [14]. In 2017, Venters et al. [15] conducted a numerical study on which the maximum output power optimizer of turbines They examined the air duct, the position and angle of the channel concerning the rotor page, and also the coefficient They optimized the thrust for the rotor [15]. In 2017, Asl et al. [16] investigated the experimental design of the blade and the design of a ducted wind turbine and their effects They examined the design, number and angles of the blades on the rotational velocity in a channel and came to the conclusion that increases the wind velocity up to 2.46 times [16]. In 2017, Farahpour et al. [17] investigated an Invelox turbine in a laboratory, and they came to this conclusion They found that at any wind velocity, the free flow velocity in the venturi turbine increases and also the increase in the free flow velocity will increase the output power from 300 to 3600 watts. The most important result obtained is the constant output power from the input velocity onwards [17]. In 2018, Dighe et al. [18] numerically investigated the effect of the geometry of the cross-section of the duct along the length of the rotor. They examined three duct samples and concluded that generally the separation areas were obtained from suction Next to the front edge of the duct, which lowers the aerodynamic performance [18]. The upcoming study is the numerical investigation of the dimensions of Invelox to optimize it and operationalize it for implementation in a residential building.

Recently, Hodgkin et al. [19] studied wind turbine tip-vortex breakdown dynamics in a conventionally neutral atmospheric boundary layer. Taghinezhad et al. [20] evaluated and optimized the performance of dual-rotor wind turbines installed inside a developed duct. The effect of different operating conditions on the extracted power was compared between the dual-rotor wind turbines (DRWT) and single-rotor wind turbines. Zadehbagheri et al. [21] studied a multi-objective optimal energy planning strategy for a hub, incorporating renewable and non-renewable resources, like PV, tidal turbine, micro-turbine, and energy storage by utilizing the time of use program. Zhang [22] realized that he could install the multi-directional wind energy system on top of high-rise buildings in urban areas, demonstrating that wind energy is capable of supplying the electricity consumption of high-rise buildings with this system. In Zhang's investigations, the bottom diameter of the multidirectional wind energy system was one meter. Zhang [22] has developed these systems with numerical and experimental studies. Since some facilities such as the air conditioning system on the roof may limit the installed space, this multi-directional flow system is small. The geometry of the wind energy system is multi-directional. This wind energy system includes a cover and five inlet channels. Five channels can receive the wind input from all directions and transmit it to the vertical output, so the wind energy system is suitable for wind with successive changes in urban areas.

MATERIAL AND METHODS

Figures 1 and 2 show a view of the three-dimensional model and the parametric dimensions of the solution field, respectively. This geometry is made up of separate parts such as the input of the computational field, the output of the computational field, the upper part of the field which is the external environment conditions, the bottom of the field which is the ground level, and the Invelox turbine which is located in the center of the computational field. It is also clear that the entrance of the solution field from the input boundary condition of the exit velocity of the field has the pressure output condition for the floor of the field and also the walls of the Invelox turbine itself from the boundary condition of the wall. And likewise, investigating the behavior of the Invelox wind turbine in the validation part, changing the installation height and the size of the turbine has been used for simulation. In this work, the fluid flow is incompressible, stable, and turbulent. Similarly, the air is homogeneous and isotropic with constant viscosity. Flow equations (continuity and momentum), and turbulence transfer equations are numerically solved. The dimensions of the investigated Invelox are given in Table 1.

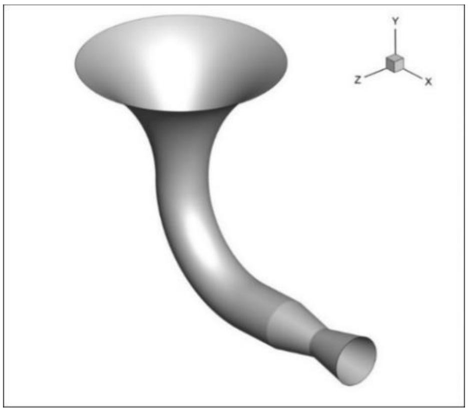


Figure 1. 3D model of Invelox

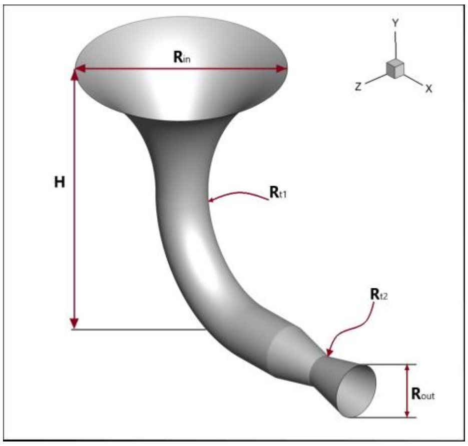


Figure 2. The dimensions required to solve the problem

Table 1. Geometry dimensions of the investigated invelox

	R_{in}(m)	R_{out}(m)	H(m)	R_{t1}(m)	R_{t2}(m)
Case (0)	6	1.5	15	1.5	0.9
Case (1)	1.5	0.375	3.75	0.375	0.225
Case (2)	0.5	0.375	3.75	0.375	0.225
Case (3)	0.5	0.375	2.75	0.375	0.225
Case (4)	1	0.375	2.75	0.375	0.225

Navier-Stokes equations include the main fluid field equations. Complementing the Navier-Stokes equations are the equation of state, mass conservation equation, and fluid properties. And by adding initial and boundary conditions to this set of equations, the fluid flow analysis can be done completely, and here we don't need to solve the energy equation. So, first, we write the equations for instantaneous quantities, that is, average quantities plus fluctuating quantities. Then we perform time averaging on both sides of each equation. Of course, in this regard, it should be noted that if equality is established for instantaneous equations, this equality will also be established for its time average (for a specific range of time). Finally, simplify the equations until the time average quantities appear.

It is regarded, the following equations are stated as a turbulent compressible flow [13]:

$$\frac{\partial}{\partial x_i}(\bar{\rho}\,\bar{u_\iota}) + \frac{\partial}{\partial x_i}(\overline{\rho'\,u_\iota'}) = 0 \qquad (1)$$

The momentum conservation equation will be as follows:

$$\rho\left[\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j\frac{\partial \bar{u}_i}{\partial x_j}\right] = \bar{B_\iota} - \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\frac{\partial \bar{u}_i}{\partial t} - \rho\overline{u_i'u_j'}\right] \qquad (2)$$

In the present problem, the standard method for pressure mediation is used. According to the nature of the flow in the present work, this mediation method will not have any problem and can be a suitable choice. For velocity and pressure dependence, the PISO algorithm is used. Fluid is considered a Newtonian fluid.

RESULTS AND DISCUSSION

The first step in the current modeling is to check the quality of the grid used. The type of meshing used in the modeling is unorganized meshing. To reach a certain number of meshes that do not experience a significant change after the results, diagram 3 is presented.

This graph, which was obtained for the maximum coefficient of the velocity value, shows that the results do not change significantly from the mesh number of 350,000 onwards, and therefore, this mesh number is considered to accelerate the convergence for the continuation of the modeling work.

At the beginning of the work, there is a need to compare the obtained numerical results with the relationships related to mass persistence. By integrating the continuity equation, the following relations will prevail in the two sections of the inlet and the bottleneck.

$$Q_{in} = Q_{t2} \rightarrow V_{in}A_{in} = V_{t2}A_{t2} \qquad (3)$$

By examining different input velocities, diagram 4 is extracted.

Diagram 4 shows that there is a good match between the data obtained in the present modeling of the continuity equation. The average error for the entire chart is approximately 5.13%, which is an acceptable difference.

Due to the very large dimensions of case 0 and the impossibility of implementing it on a residential building, the four cases mentioned in the previous chapter with much smaller dimensions have been investigated in this article, the results of which are presented below. It is necessary to explain that in all the studies conducted in this section, the inlet velocity of 1 m/s is considered. Figure 5 presents the results for different modes.

For a better and more accurate comparison of the results obtained the geometric dimensions in the four studied cases, the velocity diagrams on the axial line passing through the venturi throat for all four cases are shown in Figure 6. As can be seen, the data related to mode 1 has much more values than the other three modes, on the other hand, the dimensions of this mode are also larger and it may not fit well with the residential building, and therefore the desired mode should be selected from among the three modes 2, 3 and 4 take place.

Considering the average velocity in the venturi range where the power generation turbine is located, mode 2 can be operated in a residential building with a velocity of 6.54 m/s and a difference of 5% from the other two modes. Figure 8 shows the selected final model along with its dimensions.

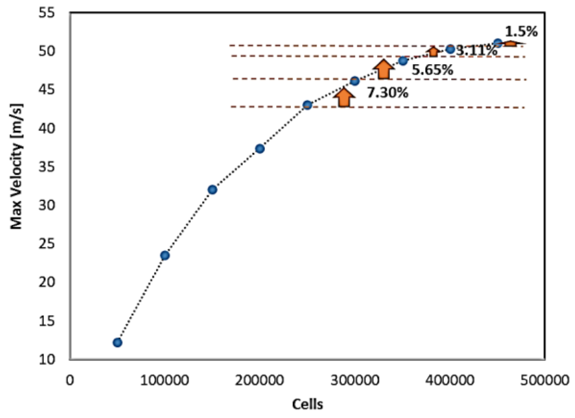


Figure 3. Grid independence diagram

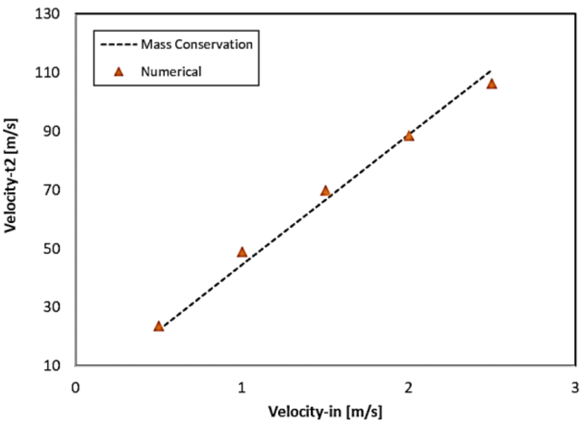
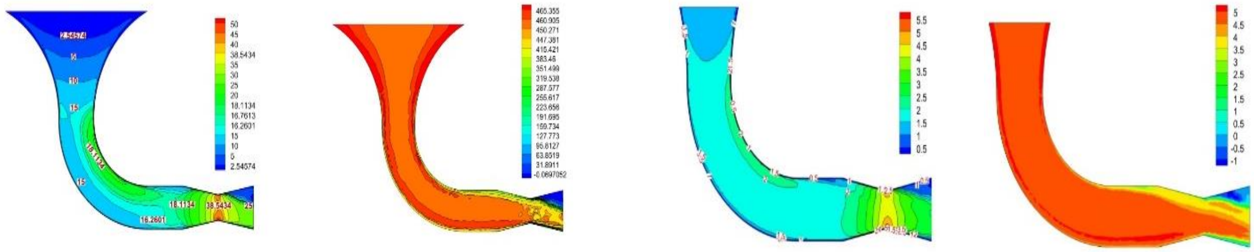


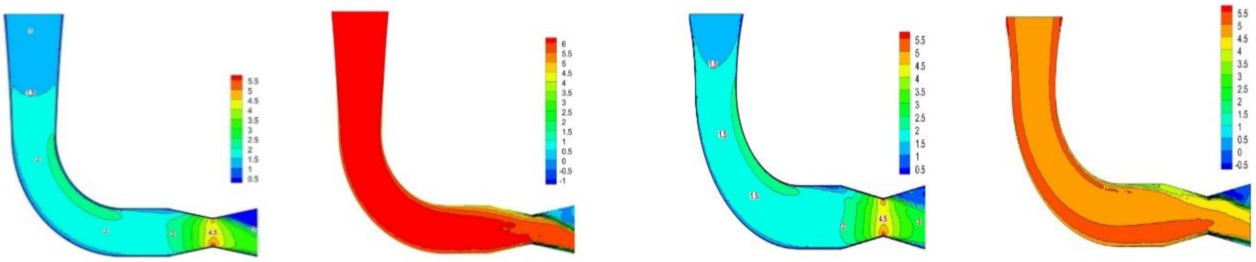
Figure 4. Validation chart

Mode 1: *Mode 3:*



Velocity Pressure Velocity Pressure

Mode 2: *Mode 4:*



Velocity Pressure Velocity Pressure

Figure 5. Pressure and velocity contours for the first case of different modes (Modes 1 to 4)

Investigating the effect of wind direction in Mode 2

In this section, the effect of the direction of the inlet flow to the Invelox on the average velocity of the venturi throat (where the turbine is located) in model 2 is investigated.

In the previous cases, the air entry angle was 90 degrees, and two angles of 30 and 45 degrees were also investigated which shows the velocity diagram in Figure 9. Based on the obtained results, it is clear that by keeping the inlet velocity constant and changing the inlet direction, the velocity in the venturi increases significantly from 30 to 90 angle. Table 2 shows the percentage of changes.

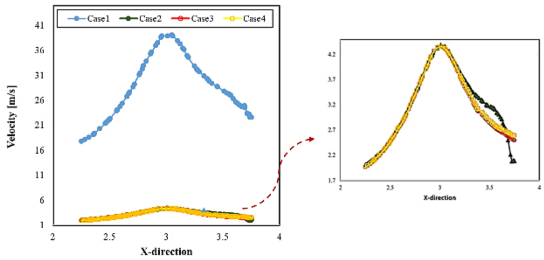


Figure 6. Velocity on the axial line passing through the venture

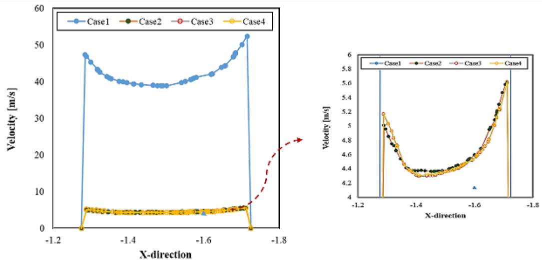
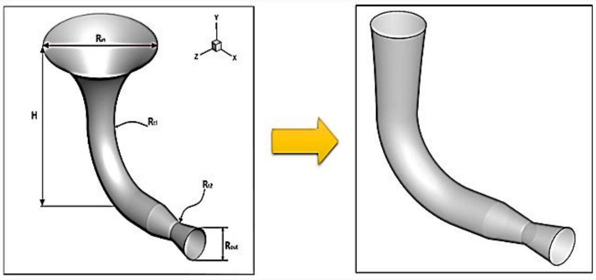


Figure 7. velocity chart comparison for different models on line 1



	Rin(m)	Rout(m)	H(m)	Rt1(m)	Rt2(m)
Case(2)	0.5	0.375	3.75	0.375	0.225

Figure 8. The selected final model

Table 2. Percentage of velocity changes at different angles

45 degrees to 30 degrees	90 degrees to 45 degrees	90 degrees to 30 degrees
42.34234	41.77215	101.8018
39.73214	43.76997	100.8929
40.41404	43.80342	101.9202
40.31532	44.78331	103.1532

Figure 9 shows the average velocity values in the venturi for the three angles investigated in mode 2. Based on these data, in the design, one should try to make the direction of the flow velocity at the inlet closer to the 90-degree angle, because the velocity values in the venturi throat at 90 degrees are more than 100% higher than the velocity values at the 30-degree angle with the same value. The velocity is at the entrance.

Investigating the effect of dust on turbine performance

In this part of the present study, the effect of dust on the performance of wind turbine blades in the venturi throat. It should be noted that the presence of dust causes a change in the density and viscosity of the fluid, which leads to a change in the applied force, and the accumulation of dust on the blade can cause an increase in the surface roughness of the blade and a decrease in turbine performance. In this step, in order to introduce the effect of dust in the simulation process, assuming that the dust fraction is known for the air of Urmia city, using the volume fraction technique, this phenomenon is applied in the simulation. To simulate the effect of residual dust on the blade surface, the surface roughness is replaced by other numbers. Figure 10 shows the modeled turbine blade.

According to Mahmoud [23], the thermos physical characteristics of clean air in Urmia are shown in Table 3. Simulating the airflow carrying dust and dust due to the low rate of volume fraction of solid particles and assuming the same velocity of particles and air as well as the assumption of homogeneity of the flow, the flow can be considered as a single phase and its properties can be changed. By determining the volume fraction of suspended particles, the following relationships are defined for air density and viscosity:

$$\rho_m = \rho_d \emptyset_d + (1-\emptyset_d)\rho_a \tag{4}$$

$$\frac{\mu_m}{\mu_a} = \frac{1}{(1-\emptyset_d)^{2.5}} \tag{5}$$

where ρ_d is the density of solid particles, ρ_a is the density of fresh air and $\emptyset_d$ is the volume fraction of solid particles. The results of the numerical study show that increasing the volume fraction of solid particles increases the pressure difference on the upper and lower surface of the blade. Figure 11 shows the frequency changes of the vane movement at different times with the volume percentage of different solid particles.

As shown in Figure 11, the oscillation frequency of the blades increases with the increase in rotational velocity. In the case of not considering dust particles, this value increases by 25%, while considering dust particles as 0.1% as an example, it increases up to 300%, and this can cause irreparable damage to the Turbine as well as the power generation function of the imported system.

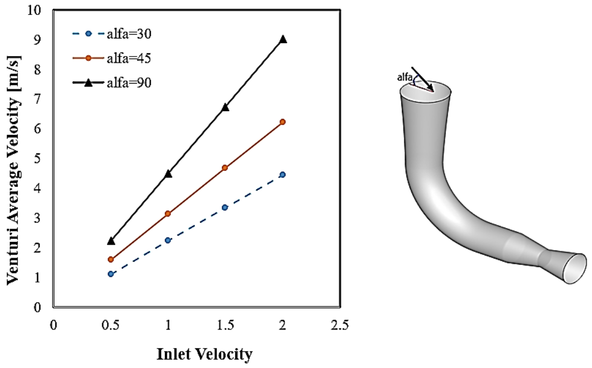


Figure 9. Average velocity comparison in venturi for model 2 for different angles

Table 3. Urmia air characteristics

Magnitude	Quantity
1.06	Density [kg/m^3]
1.6×10^{-5}	Dynamic viscosity[Pa.s]

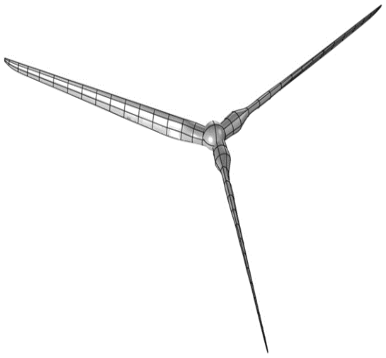


Figure 10. Modeled turbine blade

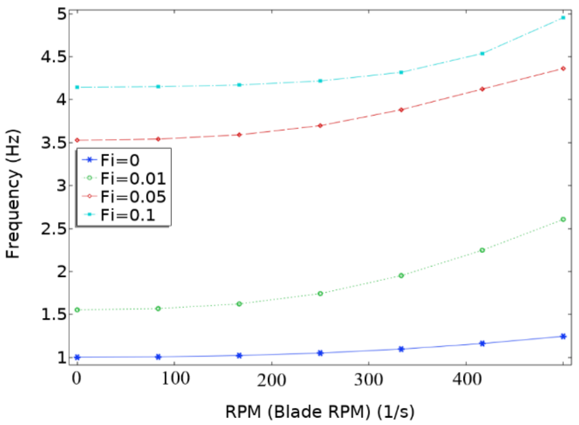


Figure 11. Frequency changes in different rotational velocities in different volume fractions of solid particles.

CONCLUSION

The present numerical work is a three-dimensional and reliable investigation of Invelox wind deflectors. In this study, the turbulent flow regime is considered, and unorganized meshing is used for analysis. The number of cells used is considered to be 350,000 based on the study of the independence of the network, and the piezo algorithm and skewness correction due to the curvature of the network have been used.

In this work, in the first step, in addition to the modeling of Invelox with conventional dimensions, four different sizes have been studied in the form of four modes for use in a residential building. In the second step, the effect of the wind-blowing angle was investigated, and in the third step, the effect of dust on the Invelox turbine was studied. The results are presented in the form of contours and graphs, and the following results are a summary of the results:

- Validation investigations were carried out in two stages with continuity equation and experimental work of Alai and Andreopoulos and the results showed 5.13 and 5.66 errors between the available data and the obtained numerical data respectively, and these values are desirable.
- The pressure contours show that the pressure increases on the sides of the wall and also forms in the throat due to the flow being thrown at the sharp upper edge of the wake area, which reduces the pressure value.
- The x-component of the velocity shows that the flow is not symmetrical, and the reason for this is the curved path of the flow, which creates a pressure difference between the walls.
- The velocity increases in the center of the route, which is the bottleneck point, and decreases again when passing through the bottleneck. The reason for this is the reduction of the cross-section in the throat and the law of conservation of mass.
- Considering the average velocity in the venturi range where the power generation turbine is located, mode 2 can be operated in a residential building with a velocity of 6.54 m/s and a difference of 5% from the other two modes.
- Based on the data obtained from the analysis of the influence of the wind angle, it is concluded that in the design, an attempt should be made to make the direction of the flow velocity at the inlet closer to the angle of 90 degrees, because the velocity values in the venturi throat at 90 degrees are more than 100 The percentage is greater than the velocity values at an angle of 30 degrees with the same velocity value at the inlet.

The results show that the oscillation frequency of the vanes increases with the increase in rotational velocity. In the case of not considering dust particles, this value increases by 25%, while considering dust particles as 0.1% as an example, it increases up to 300%, and this can cause irreparable damage to the Turbine as well as the power generation function of the imported system.

CONFLICT OF INTEREST

The authors have no conflicts of interest to declare. All co-authors have seen and agree with the contents of the manuscript and there is no financial interest to report certify that the submission is original work and is not under review at any other publication.

REFERENCES

1. Shahnazi, S., Mehrdad, S., Hijazi, M. and Rahimi Aryai, A., 2016. New design of residential apartment in Isfahan using renewable energy and modeled on traditional Iranian architecture. Master of Architecture thesis, Sepehr-Isfahan Institute of Higher Education, Isfahan, Iran
2. Naqib Al-Zhakrin Bafghi, M. A., Kargar Sharif Abad, H. and Talebi, M. H., 2016. Investigation of natural ventilation due to the use of wind deflectors with different geometries using Design Builder software. Master thesis, School of Mechanical Engineering, Department of Mechanics, Yazd Science and Art University, Technical, Yazd, Iran.
3. Göltenbott, U., Ohya, Y., Yoshida, S. and Jamieson, P., 2017. Aerodynamic interaction of diffuser augmented wind turbines in multi-rotor systems, *Renewable Energy,* 112, pp. 25-34. Doi:10.1016/j.renene.2017.05.014
4. Gavade, A., Mulla, A., Ransing, A. and Sane, N., 2018. Design and manufacturing of INVELOX to generate Wind power using nonconventional energy sources, *International Journal of Advanced Research in Science and Engineering*, pp. 588-592.
5. Anbarsooz, M., Amiri, M. and Rashidi, I., 2019. A novel curtain design to enhance the aerodynamic performance of Invelox: A steady-RANS numerical simulation, *Energy,* 168, pp. 207-221. Doi:10.1016/j.energy.2018.11.122
6. Venkatramakrishnan, S. R., Pandey, J. K., Mondal, A. K. and Karn, A., 2020. Low Speed Wind Turbines for Power Generation: A Review, *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences,* 67(1), pp. 146-169. Available at: https://www.akademiabaru.com/submit/index.php/arfmts/article/view/2843
7. Allaei, D. and Andreopoulos, Y., 2013. INVELOX: a New Concept in Wind Energy Harvesting, Proceeding of ASME 2013 7th international Conference on energy Sustainability & 11th Fuel Cell Science, Engineering and Technology Conference ES-Fuel Cell, Minneapolis, MN, USA, July 14-19, No.18311.
8. Allaei, D. and Andreopoulos, Y., 2014. INVELOX: Description of a New Concept in Wind Power and Its Performance Evaluation, *Energy,* 69, pp. 336-344. Doi:10.1016/j.energy.2014.03.021
9. Kumar, N. M., Subathra, M. and Cota, O. D., 2015. Design and wind tunnel testing of funnel based wind energy harvesting system, *Procedia Technology,* 21, pp. 33-40. Doi:10.1016/j.protcy.2015.10.006
10. Allaei, D., Tarnowski, D. and Andreopoulos, Y., 2015. INVELOX with multiple wind turbine generator systems, *Energy,* 93, pp. 1030-1040. Doi:10.1016/j.energy.2015.09.076

11. Khadka, N., 2015. Efficient use of wind energy to generate electrical power by means of wind tunnel effect and Coanda effect. Bachelor Thesis, Charles Darwin University, Australia.
12. Raeisi Mahdi Abadi, P., Vahdati Daneshmand, S. and Sharifi, R., 2016. Development and Economical Evaluation for Wind Power Plant in "Chabahar in Sistan and Baluchestan Province-Iran, *Journal of Renewable Energy and Environment*, 3(1), pp. 17-24. Doi:10.30501/jree.2016.70074
13. Georgescu, M.-R., Vladut, A. C., Cosoiu, C. I. and Georgescu, A.-M., 2017. Numerical study of the flow inside a wind trapping system, 2017 International Conference on Energy and Environment (CIEM): IEEE, pp. 330-334, Doi:10.1109/CIEM.2017.8120846
14. Anbarsooz, M., Hesam, M. S. and Moetakef-Imani, B., 2017. Numerical study on the geometrical parameters affecting the aerodynamic performance of Invelox, *IET Renewable Power Generation*, 11(6), pp. 791-798. Doi:10.1049/iet-rpg.2016.0668
15. Venters, R., Helenbrook, B. T. and Visser, K. D., 2018. Ducted wind turbine optimization, *Journal of Solar Energy Engineering*, 140(1), pp. 011005. Doi:10.1115/1.4037741
16. Asl, H. A., Monfared, R. K. and Rad, M., 2017. Experimental investigation of blade number and design effects for a ducted wind turbine, *Renewable Energy*, 105, pp. 334-343. Doi:10.1016/j.renene.2016.12.078
17. Farahpour, H., Saniei, P., Khatibzadeh, H. and Radfar, P., 2017. INVELOX wind turbine power curve modeling based on field data in Manjil, Iran, Proceedings of the 3rd International Conference on IEA, Tehran, Iran.
18. Dighe, V. V., de Oliveira, G., Avallone, F. and Van Bussel, G., 2018. On the effects of the shape of the duct for ducted wind turbines, 2018 Wind Energy Symposium, 8–12 January 2018, pp. 0997, Doi:10.2514/6.2018-0997
19. Hodgkin, A., Deskos, G. and Laizet, S., 2023. On the interaction of a wind turbine wake with a conventionally neutral atmospheric boundary layer, *International Journal of Heat and Fluid Flow*, 102, pp. 109165. Doi:10.1016/j.ijheatfluidflow.2023.109165
20. Taghinezhad, J., Alimardani, R., Masdari, M. and Mahmoodi, E., 2021. Performance optimization of a dual-rotor ducted wind turbine by using response surface method, *Energy Conversion and Management: X*, 12, pp. 100120. Doi:10.1016/j.ecmx.2021.100120
21. Zadehbagheri, M., Kiani, M. J., Pirouzi, S., Movahedpour, M. and Mohammadi, S., 2023. The impact of sustainable energy technologies and demand response programs on the hub's planning by the practical consideration of tidal turbines as a novel option, *Energy Reports*, 9, pp. 5473-5490. Doi:10.1016/j.egyr.2023.04.377
22. Zhang, X., 2014. Analysis and optimisation of a novel wind turbine. PhD, University of Hertfordshire, UK [Online]. Available at: https://uhra.herts.ac.uk/handle/2299/13897
23. Mahmoud, N., El-Haroun, A., Wahba, E. and Nasef, M., 2012. An experimental study on improvement of Savonius rotor performance, *Alexandria Engineering Journal*, 51(1), pp. 19-25. Doi:10.1016/j.aej.2012.07.003

Persian Abstract

چکیده

پژوهش حاضر اینولاکس را به صورت سه بعدی و پایدار شبیه سازی می‌کند. رژیم جریان، جریان آشفته است از یک شبکه سازمان نیافته با تعدادی سلول ۳۵۰۰۰۰ است. این کار به بررسی مدل‌سازی اینولاکس با ابعاد متعارف و چهار اندازه مختلف در قالب چهار حالت برای استفاده در یک ساختمان مسکونی پرداخته است. داده‌های عددی با خطای کمتر از ۶ درصد با داده‌های تجربی و تحلیلی موجود مطابقت خوبی دارند. نتایج نشان می‌دهد که با در نظر گرفتن سرعت متوسط حالت ۲ با سرعت ۶/۵۴ متر بر ثانیه و اختلاف ۵ درصدی با دو حالت دیگر، قابلیت بهره‌برداری در ساختمان مسکونی را دارد. شایان ذکر است که در این تحقیق علاوه بر این، اثر گرد و غبار بر عملکرد توربین مورد ارزیابی قرار گرفت. نتایج نشان می‌دهد که فرکانس نوسان تیغه‌ها با افزایش سرعت چرخش افزایش می‌یابد. در صورت در نظر نگرفتن ذرات گرد و غبار روی پره‌ها، این مقدار ۲۵ درصد افزایش می‌یابد، در حالی که با در نظر گرفتن ذرات گرد و غبار با مقدار ۰/۱ درصد، تا ۳۰۰ درصد افزایش می‌یابد و این می تواند آسیب‌های جبران ناپذیری به توربین و همچنین عملکرد تولید برق سیستم به همراه داشته باشد.