CFD Analysis On The Performance Of Wind Turbine With Nozzles

Chunkyraj Kh, C. D. Hampali, Anand S. N.

Abstract: In this paper, an effort has been made in dealing with fluid characteristic that enters a converging nozzle and analysis of the nozzle is carried out using Computational Fluid Dynamics package ANSYS WORKBENCH 14.5. The paper is the continuation of earlier work: Analytical and Experimental performance evaluation of Wind turbine with Nozzles. First, the CFD analysis will be carried out on nozzle in-front of wind turbine where streamline velocity at the exit, volume flow rate in the nozzle and pressure distribution across the nozzle will be studied. Experiments were conducted on the Wind turbine with nozzles and the corresponding power output at different air speed, and different size of nozzles were calculated. Different shapes and dimensions, with special contours and profiles of nozzles were studied. It was observed that the special contour nozzles have superior outlet velocity and low pressure at nozzle exit, the design has maximum Kinetic energy. These indicators conclude that the contraction designed with the new profile is a good enhancing of the nozzle performance.

Keywords: CFD, Nozzle, Wind power, Wind turbine.

1. Introduction

In present world, enormous amount of effort has been carried out in overcoming the increasing demand of energy, Renewable energy being one of the main player. Wind turbines work on the principle by converting the Kinetic energy in the wind into rotational Kinetic energy (Mechanical energy) in the turbine and finally to electrical energy. The energy for the conversion mainly depends on the wind speed and the swept area of the turbine. So if the wind speed can be increased with external apparatus, there is increase in the output power. Here in this analysis, nozzle is used to increase the velocity of the wind. With this set-up, the output power can be increased marginally. Performance evaluation by varying the dimensions of nozzles is also be carried out. This paper is the continuation of the earlier work entitled “Analytical and Experimental performance evaluation of Wind turbine with Nozzles”. In this paper, an effort has been made in dealing with fluid characteristic that enter the converging nozzle and analysis of the nozzle is carried out using CFD package ANSYS WORKBENCH 14.5. The streamline velocity at the exit, volume flow rate in the nozzle and pressure distribution throughout the nozzle will be observed. Experiments were conducted on the Wind turbine with nozzles and the corresponding Power output at different air speed, and different size of nozzles were calculated. Md. Abu Abrar et al.,[1] studied the similarity between experimental and CFD analysis for output power of horizontal axis mini wind turbine.

- *Chunkyraj Kh. is currently pursuing Masters degree program in Machine Design and Dynamics in Reva University, Bengaluru, India. E-mail: chunky_kh@yahoo.com*
- *C. D. Hampali is currently working as a Sr. Associate Professor in the department of School of Mechanical Engineering in Reva University, Bengaluru, India. E-mail: cdhampali@revainstitution.org*
- *Anand S.N. is currently working as an Associate Professor in the department of School of Mechanical Engineering in Reva University, Bengaluru, India. E-mail: snagowda1974@gmail.com*

Ahmad Atieh et al. [2] showed that the performance of a novel wind tunnel with conical shape and elevated structure have better compared to straight conical shape with no elevation, for numerical analysis. Maria Rodriguez Lastra et al. [3] conducted experimental and numerical analysis on four different contraction profiles of wind tunnel, where the new contraction design showed better performance. Special contoured nozzles were numerically analyzed for the effect of nozzle geometries on the discharge coefficient and turbulence characteristics. It was observed that the geometry greatly effect on the sonic lines and discharge coefficient and turbulence [4], [5]. S. Zaghi et al. [6] investigated the blockage effect for wind tunnel experiments and simplified boundary conditions to be applied on numerical simulations. It was observed that the use of nozzle increases the velocity of the fluid leaving it. And the power generation of wind energy depends largely on the wind speed. The shape and size of the nozzle plays a vital role in proper selection and efficiency of the nozzle. The streamline velocity of the fluid, volume flow rate of the fluid and the pressure distribution across the nozzle at various inlet speed will be studied using CFD analysis on ANSYS WORKBENCH 14.5. An optimized size and shape of nozzle will be studied with required specifications.

2. Re-visiting the previous work

2.1. Analytical Approach

The Extractable power in the wind is given by the expression:

$$P_{avail} = \frac{1}{2} C_p \rho A V^3 \text{ (Watt)} \quad (1)$$

Where, ρ is Density of air (in kg/m^3); A is Area of Swept (in m^2); V is Wind speed (in m/s); C_p is Power coefficient; C_{pmax} = 0.59 according to Betz limit. The swept area of the turbine can be calculated from the length of the turbine blades using the equation for the area of circle:

$$A = \pi r^2 \text{ (m}^2\text{)} \quad (2)$$

Where, r is radius of the rotor blade (in m)

The above equation (1) is used to calculate the generated power in both the cases (i.e. without nozzle and with nozzle). Based on the different speed of wind, the variation in the

Power is obtained and calculated. In power generation with nozzle, the wind velocity is increased due to nozzle before hitting the turbine blades. This increase in velocity increases the Power generation of the turbine.

Nozzle:

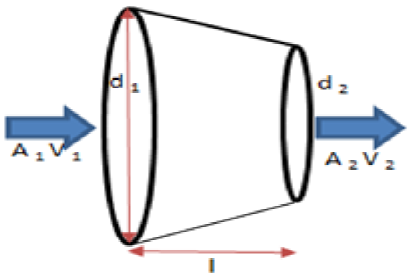

Fig. 1. *Schematic diagram of a Converging nozzle*

Assuming Steady flow and conservation of masses:

$$A_1V_1 = A_2V_2 \quad (3)$$

Where, A_1 & A_2 are Inlet and Outlet area of nozzle (in m^2)

V_1 & V_2 are Inlet and Outlet wind speed (m/s)

Table 1 *Nozzle dimensions*

Sl. No.	Inlet diameter,d_1 (m)	Outlet diameter,d_2 (m)	Length of Nozzle, l (m)
Nozzle no.1	0.33	0.22	0.35
Nozzle no.2	0.44	0.22	0.35

2.2. Experimental Approach

The experimental approach consists of a small rotor with blades, nozzles (two nos.), an external air blower, Anemometer, Ammeter, Voltmeter and Led bulbs. The Power generated by the rotor is compared with the power obtained from analytical approach. The schematic arrangement of the experimental Set-up is shown below.

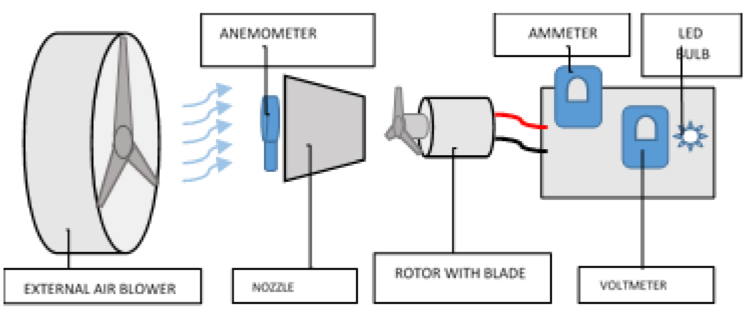

Fig. 2. *Schematic diagram of the Experimental set-up*

The rotor with the blades mounted is fitted with a PCB on the back side where the generated Power can be calculated by using the measured Current and Voltage that develops in the circuit (across the led bulb). Initially the experiment is carried out without the nozzle. The wind speed is measured by an anemometer at a distance of 50cm from the rotor blades. The developed current and voltage across the led bulb is measured by two multi-meters (one acting as Ammeter and another as Voltmeter).

Finally the Power is calculated based on the product of Current and voltage.

$$P_E = V \times I \text{ (Watt)} \quad (4)$$

Where, V is Voltage (in V), I is Current (in A)

The next arrangement is with the nozzle where the nozzle is mounted in-front of the rotor blade.

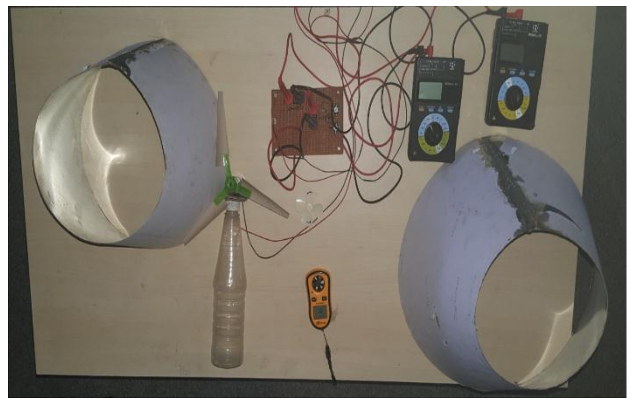

Fig. 3. *Experimental Set-up*

The experimental observation and Power variation curve with different wind speed is discussed below.

2.3. Results and Discussion:

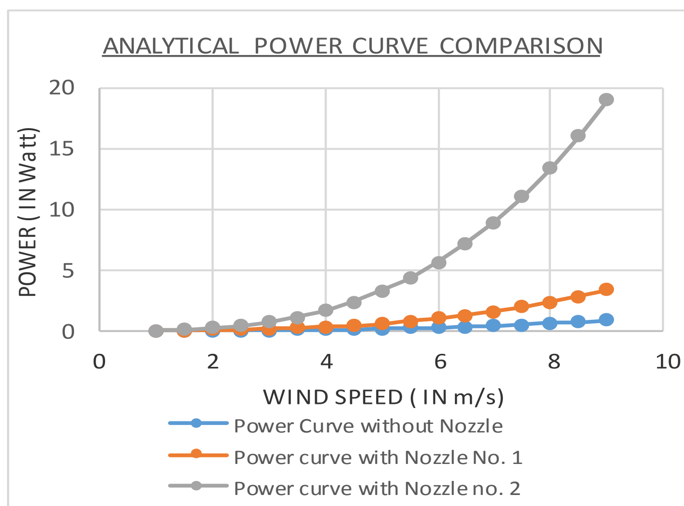

Fig. 4. *Analytical Power curve comparison*

Analytical Result:
It was observed that the power generation increases with the increase in wind speed. And the Power curve follows a parabolic curve with the increase in wind speed. It was also observed that the use of nozzle increases the power generation for the same wind speed. With the increase in diameter ratio (d_1/d_2) of the Nozzle, the Power generation also increases.

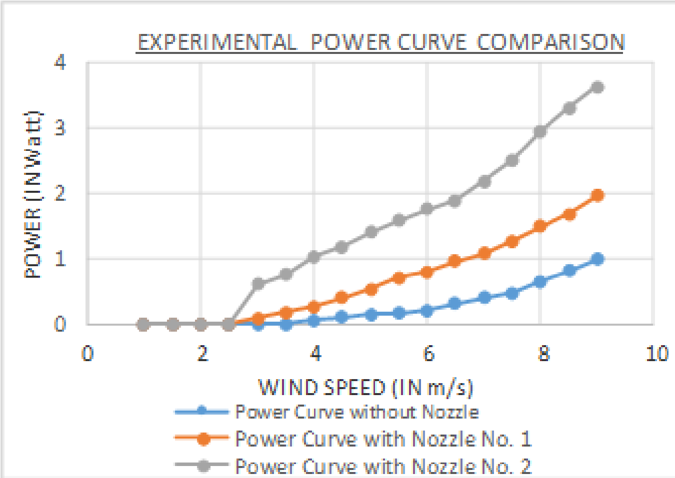

Fig. 5. *Experimental Power curve comparison*

Experimental Result:
It was observed that the power generation increases with the increase in wind speed. It was also observed that the use of nozzle increases the power generated for the same wind speed. With the increase in diameter ratio (d_1/d_2) of the Nozzle, the Power generation also increases.

3. Current work & Methodology

- To prepare the solid model of nozzles in ANSYS WORKBENCH 14.5.
- To obtain the Streamline velocity, volume flow rate and pressure distribution across the nozzle for different inlet speeds.
- To obtain the Streamline Velocity, Volume Flow rate and pressure distribution across the nozzle for:
  - Different inlet-to-outlet diameter ratio.
  - Different length of the nozzle.
- Results are tabulated for outlet velocity of Nozzle, Relative Pressure distribution at Nozzle exit.
- To study the pressure distribution across the nozzle at different inlet speeds.
- Find the optimum shape of the nozzle at various inlet speed.

Data considerations and assumptions:

MESH: Physical Preference - CFD
Solver Preference - CFX
Sizing & type- Hexagonal with fine mesh
Set up: Fluid domain: Air at 25oC (Continuous Fluid)
Reference Pressure 1 atm.
Non Buoyant model and Stationary domain
Fluid model: Isothermal (25^oC)
K-Epsilon turbulence model

3.1. Models:

4 Normal shaped Nozzles:

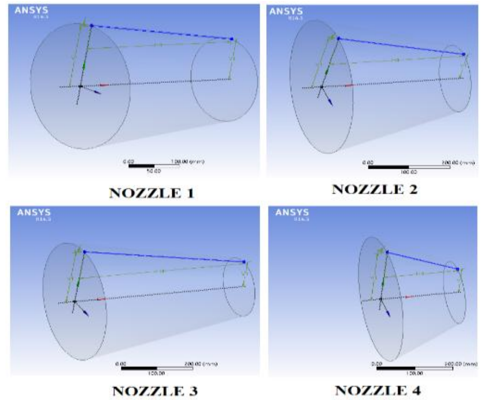

Fig.6 *Shape of Normal shaped nozzles*

Table 2: *Dimensions of Normal shaped nozzles*

Sl. No.	Inlet diameter,d_1 (mm)	Outlet diameter,d_2 (mm)	Length of Nozzle, l (mm)	Thickness, k (mm)
Nozzle 1	330	220	350	2
Nozzle 2	440	220	350	2
Nozzle 3	440	220	500	2
Nozzle 4	440	220	200	2

By analyzing the results between Nozzle 1 and 2, we can conclude the effect of inlet-to-outlet diameter ratio on nozzle. Also by analyzing the results between Nozzle 2, 3 and 4, we can come to conclusion of the effect of length on nozzle.

4 Special shaped Nozzles:

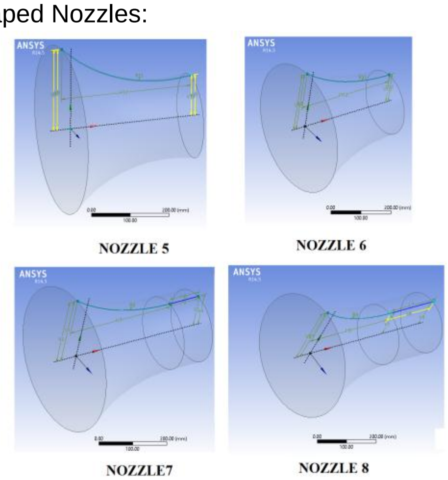

Fig.7 *Shape of Special shaped nozzles*

Table 3. *Dimensions of Special shaped nozzles*

Sl. No.	Inlet dia.,d_1 (mm)	Outlet dia.,d_2 (mm)	Length of Nozzle, l (mm)	Radius of curvature, r (mm)	Cylindrical length (mm)
Nozzle 5	330	220	350	400	-
Nozzle 6	440	220	350	600	-
Nozzle 7	440	220	500	600	100
Nozzle 8	440	220	200	600	200

By analyzing the results between Nozzles 5, 6, 7 and 8, we can come to conclusion of the best solution and optimized shape of the nozzle.

4. Observations

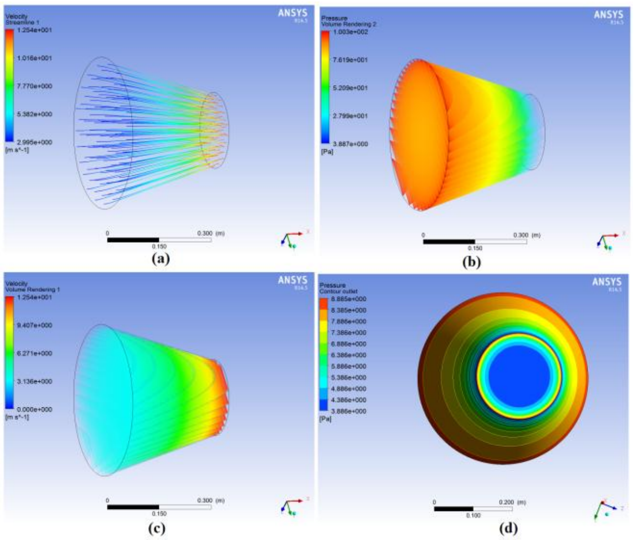

Fig.8 *(a) Streamline velocity, (b) Pressure-volume flow across the nozzle, (c) Velocity-volume flow and (d) Relative pressure distribution at nozzle exit of Nozzle 2 at 3 m/s inlet speed.*

The above figures show the streamline velocity, Pressure-volume flow across the nozzle, Velocity-volume flow and relative pressure distribution at nozzle exit 0f Nozzle 2 at the inlet speed of 3 m/s. It was observed that similar trend were shown in the analysis of the normal shaped nozzles. The observations of normal shaped nozzles are illustrated below. In all the four cases (viz. Nozzle 1, 2, 3 and 4), it is observed that the velocity at the nozzle exit is always higher than that of the inlet .As the inlet velocity of wind increases, the outlet velocity is also increased proportionately. It is also observed that the Pressure at the inlet is higher, and it gradually decrease at the exit. By the conservation of Energy, this loss in Pressure is converted into Kinetic Energy, where the speed of the fluid is increased at the exit. In the above four cases, the Kinetic Energy developed by Nozzle 4 (shortest length) is highest and in very short period, which leads to too much turbulence at the exit. But as length increases, the turbulence nature of the fluid flow can be checked, as well there is increase in frictional loss (or eddy loss). So it is important to design the nozzle with maximum Kinetic Energy at the exit, with minimal losses. For the analysis, the specifications of Nozzle 2 is selected for Maximum Kinetic Energy where the turbulence and losses characteristics will be discussed in the next four cases (viz. Nozzle 5, 6, 7 and 8). The design of special shaped nozzles (viz. Nozzle 5, 6, 7 and 8) is based on Nozzle 2, where maximum Kinetic Energy was considered for this nozzle. With the special profile, it is observed that the Kinetic Energy is further increased. . It is observed that the high velocity region extends to almost half the length of nozzle, without compromising the Kinetic Energy. The pressure distribution across the nozzle near the exit is almost uniform. This shows that the turbulence nature near the exit is checked and controlled. Cylindrical exit in Nozzles 7 and 8 is provided to suppress the turbulent nature of the fluid.

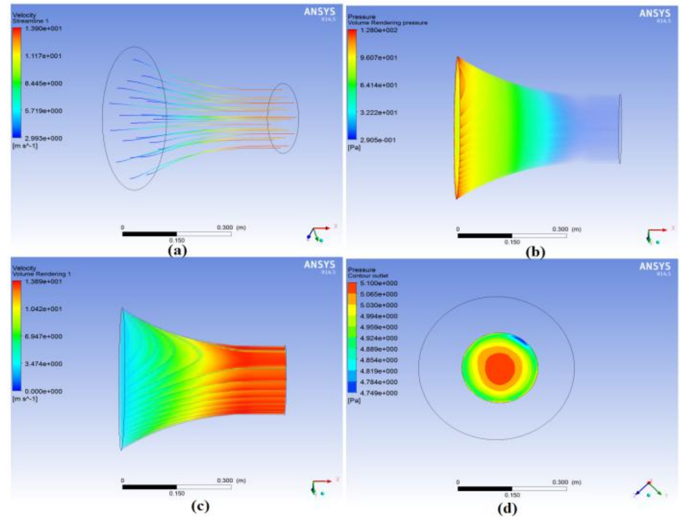

Fig.9 *(a) Streamline velocity, (b) Pressure-volume flow across the nozzle, (c) Velocity-volume flow and (d) Relative pressure distribution at nozzle exit of Nozzle 7 at 3 m/s inlet speed.*

The above figures show the streamline velocity, Pressure-volume flow across the nozzle, Velocity-volume flow and relative pressure distribution at nozzle exit 0f Nozzle 7 at the inlet speed of 3 m/s

5. Results and Discussion

5.1 Outlet Speed and Discussion:

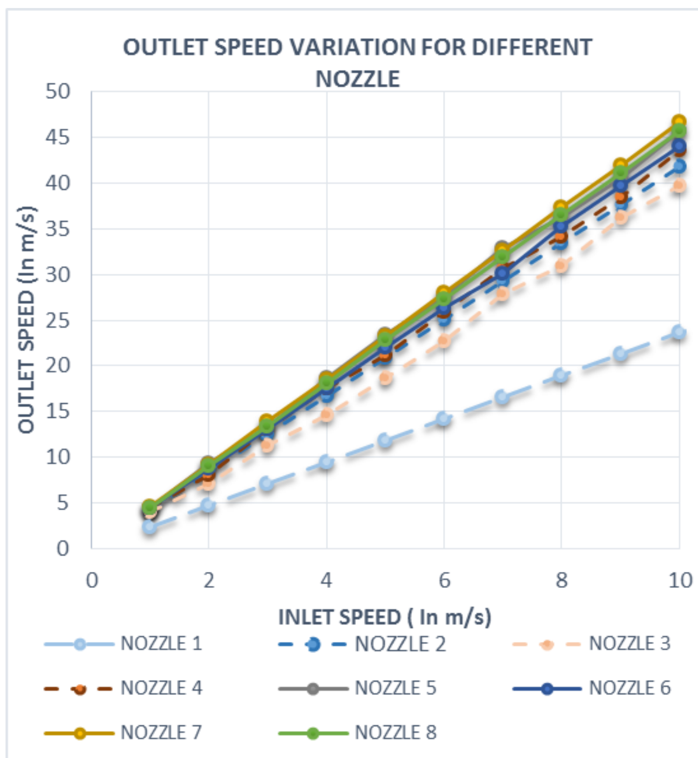

Fig.10 *Outlet Speed variation for different nozzles at different inlet speed.*

The above Fig. 10 shows the outlet speed of different nozzles at different inlet speed. It is observed that the outlet speed of the nozzles increase with the increase in inlet speed of the air. Outlet speed of the Special shaped nozzles are superior to that of Normal shaped nozzles. Of the Normal shaped nozzles, the nozzle with the shortest length (Nozzle 4) have highest outlet speed provided the inlet-to-outlet diameter ratio remains constant. Of the Special shaped nozzles, the nozzle 7 have the highest outlet speed provided the inlet-to-outlet diameter ratio remains constant.

5.2. Relative Pressure distribution at nozzle exit:

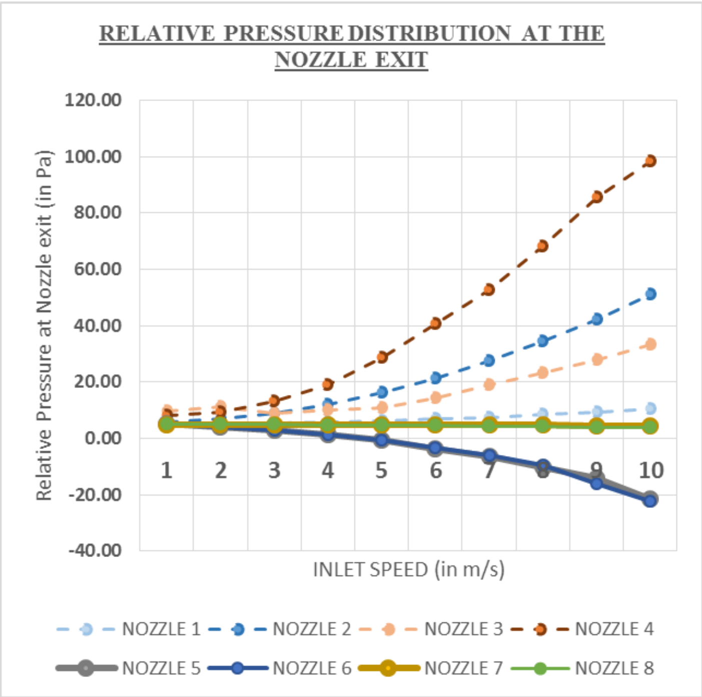

Fig.11 *Relative pressure distribution at Nozzle exit for different inlet speed.*

The above Fig. 11 shows the Relative pressure distribution at nozzle exit for different nozzles at different inlet speeds. It is observed that for normal shaped nozzles, increase in inlet-to-outlet diameter ratio increase the pressure generated at nozzle exit for increasing inlet speed. It is also observed for normal shaped nozzles that nozzle with shorter length experience higher Pressure with increase in inlet speed of air. Of the Special shaped nozzles (viz. Nozzles 5 & 6), it is observed that Pressure distribution at nozzle exit decrease with increase in inlet speed. This phenomenon is due to the shape of the contour which creates a negative pressure near the corner of the nozzle exit when fluid passes at high velocity. Of the Special shaped nozzles (viz. Nozzles 7 & 8), it is observed that the Pressure remains almost constant with increasing inlet speed. This phenomenon is due to the fact that the straight cylindrical part at the nozzle exit provides a uniform outlet pressure. It also restrict the negative pressure generation at the exit.

6. Conclusion

After studying the above figures and tables, it can be concluded that:

1) The outlet speed of the nozzle increases with the increase in inlet-to-outlet diameter ratio provided the length of the nozzle remains constant.
   But Pressure at nozzle exit also increases with increase in inlet-to-outlet diameter ratio.
2) The outlet speed of the nozzle increases with the decrease in nozzle length provided the inlet-to-outlet diameter of the nozzle remains constant.
   But as the length of the nozzle decreases the Pressure at nozzle exit increases, and the Kinetic Energy is achieved at very short interval of time, which creates turbulence at the exit.
   Thus an optimized shape and size of the nozzle should be selected.
3) After the study, Nozzle 7 has the best outlet speed and Kinetic Energy compared to others and the turbulence created at the exit is suppressed due to the cylindrical end. And Pressure generated at the nozzle exit is also almost constant.
   Nozzle 7 has the Optimized shape and size for the given specification.

References:

[1] Md. Abu Abrar, AM Ishtiaque Mahbub, Mohammad Mamun, "Design Optimization Of A Horizontal Axis Wind Turbine Through Development Of CFD Model And Experimentation". Procedia Engineering 90(2014)333-338.

[2] Ahmad Atieh, Samir Al Shariff, Naseer Ahmed, "Novel wind tunnel", Sustainable Cities and Society (2016)

[3] Maria Rodriguez Lastra, Jesus Manuel Fernandez Oro, Monica Galdo Vega, Eduardo Blanco Marigorta, Carlos Santolaria Morros, "Novel design and experimental validation of a contraction nozzle for aerodynamic measurements in a subsonic wind tunnel" , J. Wind Eng. Ind. Aerodyn.118 (2013)35–43

[4] M.M.A. Alam, T. Setoguchi, S. Matsuo, H.D. Kim, "Nozzle geometry variations on the discharge coefficient", Propulsion and Power Research 2016; 5(1):22–33

[5] Y. Yu, M.Shademan, R.M.Barron, R.Balachandar, "CFD study of effects of geometry variations on flow in a nozzle", Eng. Appl. Comput. FluidMech. 6(3) (2012)412–425.

[6] S. Zaghi, R. Muscari, A. Di Mascio, "Assessment of blockage effects in wind tunnel testing of wind turbines", J. Wind Eng. Ind. Aerodyn.154 (2016)1–9