International Journal of Academic Scientific Research
ISSN: 2272-6446 Volume 8, Issue 1 (February - March 2020), PP 23-34
www.ijasrjournal.org

Numerical Study of Flanged Diffuser Augmented Wind Turbine

Mohammad Youssef Al Hashem[1], Wafaa Fares[2]

[1,2] *Department of Power Engineering, Faculty of Mechanical Engineering, University of Aleppo*

ABSTRACT:

The aim of this work is to develop a wind turbine system that consists of a diffuser shroud with flanges at the inlet and the exit peripheries. The flanged-diffuser shroud plays a role of a device for collecting and accelerating the approaching wind. The flange generates a low-pressure region at the exit of the diffuser by vortex formation and draws more mass flow to the wind turbine inside the diffuser shroud especially at the throat and the front flange which make air easier to pass. To obtain the optimum diameter of the entry flanged diffuser height, three different diameters of entry flange were investigated, and the optimum diameter of the entry flange was found. Furthermore, the optimum entry flange was fixed and seven heights of rear exit flange were examined. As a result, the suggested wind turbine system demonstrated power augmentation for a given turbine diameter and wind speed by 16% compared to wind turbine without entry flange.

Keywords: *brim, CFD, diffuser-augmented wind turbine (DAWT), diffuser shroud with a broad-ring flange, flange type diffuser.*

1. Introduction

The need for an effective energy resource and limitation of fossil fuels is an obvious matter. Therefore, the search for alternative energy sources is an important subject. Additionally, due to concerns about environmental issues, the development and application of renewable clean energy sources are required. Wind energy technologies are considered to be as promising source of renewable energy that grow rapidly and play an important role in a new energy field. However, the percentage of wind energy currently produced is small in comparison with the total demand for energy. Therefore, the search for new wind technologies that produce higher power output is required. Wind energy generation is proportional to the cube of the wind speed and the speed is increased by improving the dynamic flue of air around the studied object and the concentration of the maximum speed at the turbine area will lead to a significant increase in energy output.

Even though, several studies regarding collecting wind energies for wind turbine have been presented by [1, 2, 3, 4, 5, 6, and 7]. The examination of a diffuser-augmented wind turbine (DAWT) was investigated by Gilbert et al. [2], Gilbert and Foreman [3], Igra [4] and others around 1980. In these studies, a diffuser with a large open angle was considered to concentrate wind energy and a boundary layer controlled with several flow slots was employed to recognize a flow that goes along the inside surface of the diffuser. Consequently, the method of boundary layer control prevents pressure loss by flow separation and increases the mass flow inside the diffuser.

In the present work, a ring-type plate, called "brim" has been added to the exit periphery of a diffuser. The addition of the plate causes vortices behind it and generates a low-pressure region behind the diffuser. Thus, the wind flows into a low-pressure region and the wind velocity is accelerated near the entrance of the diffuser as shown in Figure 1. A shrouded wind turbine equipped with a brimmed diffuser came into existence in this way. Furthermore, an inlet shroud was added to the entrance of the diffuser with a brim which makes wind easy to flow into the diffuser.

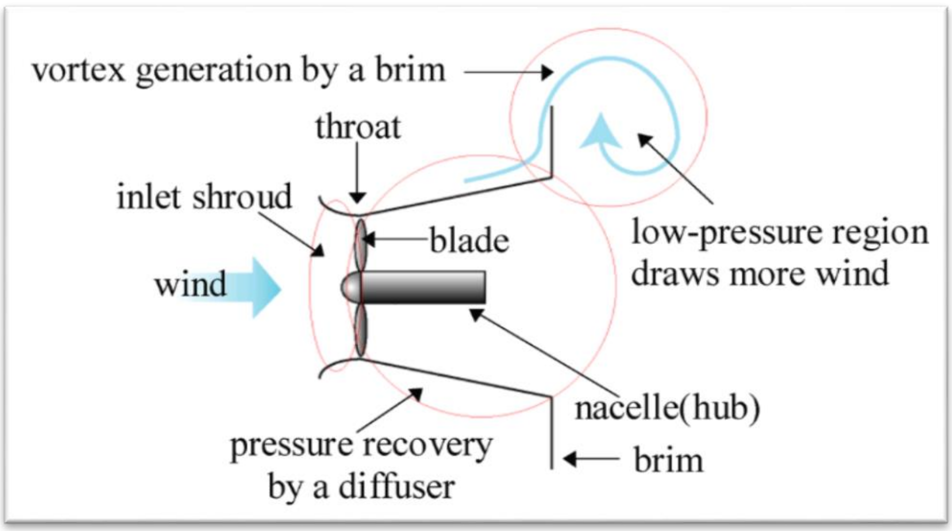

Figure 1. Flow around a wind turbine with brimmed diffuser.

2. Research objectives

The research includes a two-dimensional numerical study of flow through a diffuser with a horizontal axis wind turbine using ANSYS 16.1. The research objectives can be summarized in the following:

- Studying the effect of different diameters of the front entry flange.
- Studying the effect of different heights of the back-exit flange.

2.1 Engineering drawing:

The shape is a divergent diffuser, where the throat diameter is D, the total length of the diffuser is 1.5 D, the distance from the throat to the outlet is 1.25 D, and the inner angle of the diffuser is 12^{o}, the height of the flanged edge (turbulence generator) is h and the radius of the front entry flange is R .The diffuser is symmetrical, so the two-dimensional numerical study can be limited to half of the cover as shown in Figure 2.

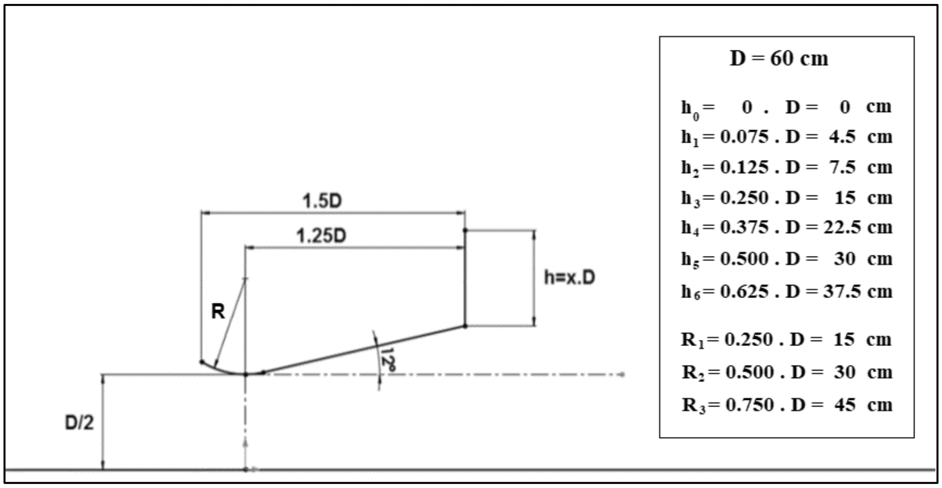

Figure 2. the geometry of the half-shroud.

Since the flow is external, the space through which air flows is defined as a rectangular dimension: length is 20 D, width is 10 D, Figure 3 shows the dimensions of the studied field.

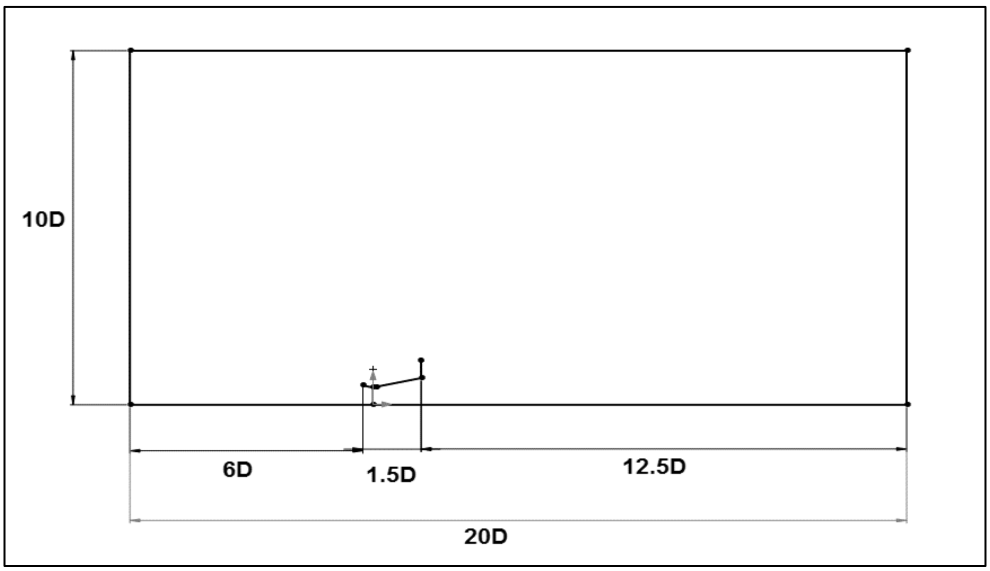

Figure 3. shows the dimensions of the field studied.

2.2. Meshing

To obtain valid simulation results, an appropriate mesh is necessary. Increasing the number of cells results in better accuracy as well as much more time for computing. Therefore, an optimal mesh is necessary to balance time consumed and the accuracy of the calculation. Three meshes were created for the mesh independence study. In order to observe the flow changes and clamp the boundary layer, the cell size decreases near the diffuser wall, at the diffuser entrance and exit, and at the throat region as shown in Fig 4. A summary of the three meshes is provided in Table 2.

Table 1. three meshes

Mesh (1)	31392cells
Mesh (2)	63840 cells
Mesh (3)	120400 cells

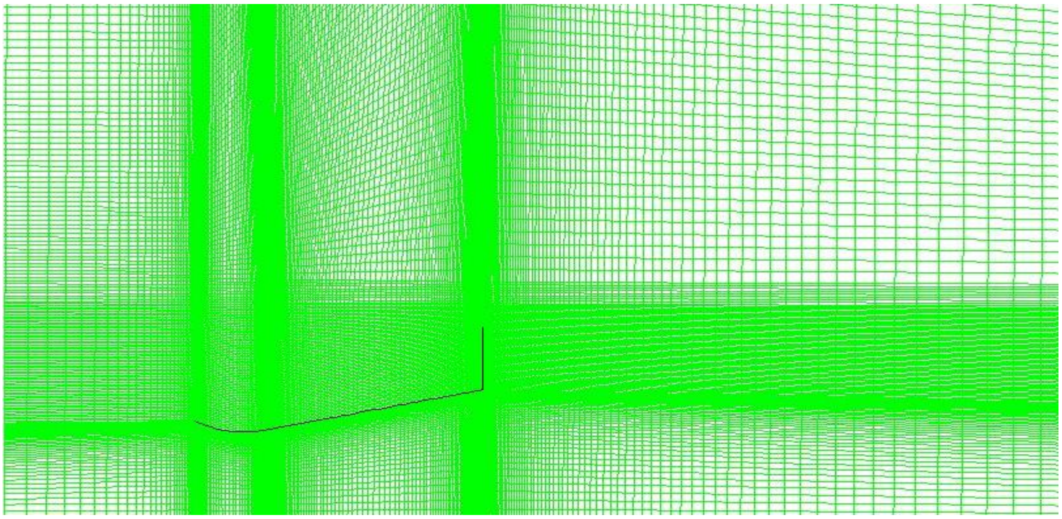

Figure 4. shows the meshing

Comparing the flow velocity within the diffuser and on its axis between the three meshes, the mesh (2) and mesh (3) were close, so mesh (2) was adopted as a valid mesh and its results will be used. Fig. 5 shows a comparison of flow velocity between the three meshes.

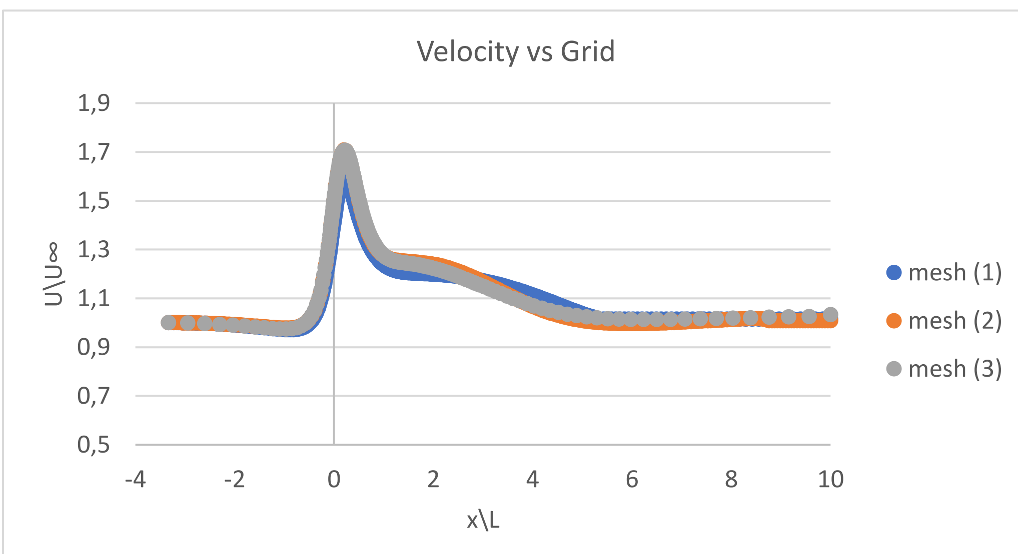

Figure 5. shows a comparison of flow velocity between the three meshes.

2.3. Numerical modeling within the Fluent program

Numerical modeling within the Fluent program was done according to the following conditions:

- 2D Axisymmetric Solver pressure based.
- Implicit – Steady -2nd order implicit
- Viscous model: $\boldsymbol{k-\omega\ SST}$
- Energy Equation: on
- Material: Fluid: air (ideal-gas, Viscosity: Sutherland)
- Operating Condition operating pressure = 101325 Pa.
- Boundary condition: Inlet: P=0,0 Pa, V= 6 m/s, T=288 k
  Outlet: Pressure outlet, P=0.0 Pa, T=288 k

3. Solution and results

3.1. The effect of the front entry flange radius

To obtain the optimum diameter of the entry flanged diffuser height, three different diameters of entry flange ($R_1 = ¼\,D = 15cm,\ R_2 = ½\,D = 30cm,\ R_3 = ¾\,D = 45\,cm$) were examined. Comparison between the flow velocities in the three cases was done. To obtain greater stability, the diameter of the entry flange that gave the highest velocity at the throat area and slightest disturbances around the diffuser was chosen. Velocity contours for the three entry flange radii are shown in Figures (6, 7, 8).

In Fig. 6, a considerable disorder in the velocity distribution has been noticed because of the large curvature of the entry flange which generates a disturbance behind the front flange. Subsequently, it will affect the overall diffuser performance. In Fig 7, acceleration of the flow and the increase in speed at the throat has been observed. In addition, there are an acceleration of the flow and an increase in the speed at the throat as shown in Fig (8), but with a value lower than the value of the speed at the R_2, due to the large relief of the entry flange, which canceled its effect.

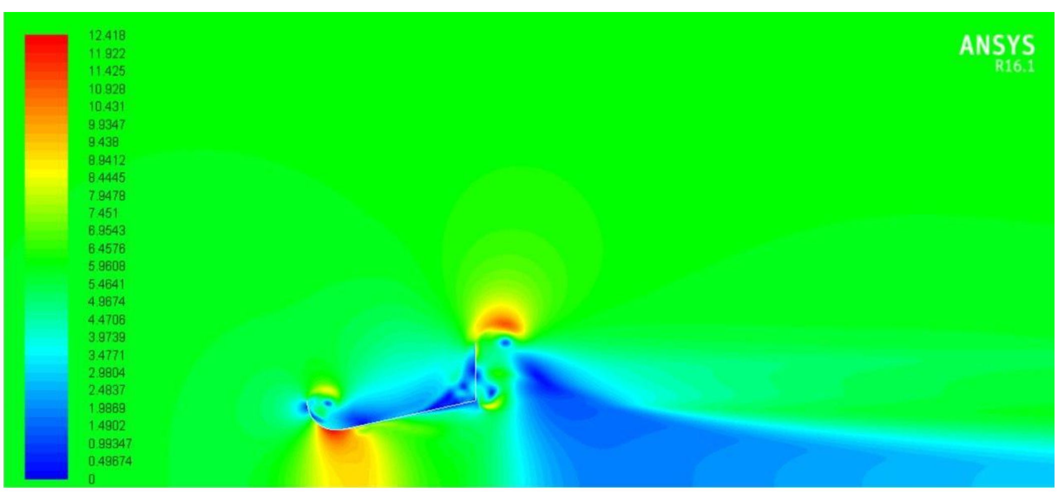

Figure 6. Shows the velocity contour of the entry flange radius $R_1 = ¼\,D = 15cm.$

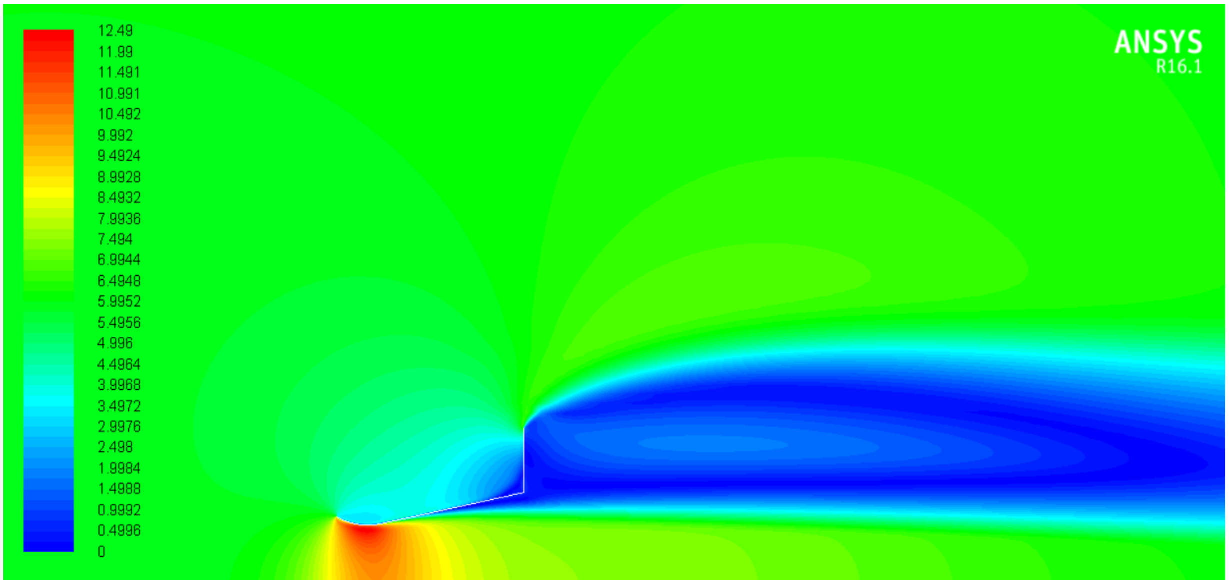

Figure 7. Shows the velocity contour of the entry flange radius $R_2 = ½\, D = 30cm$.

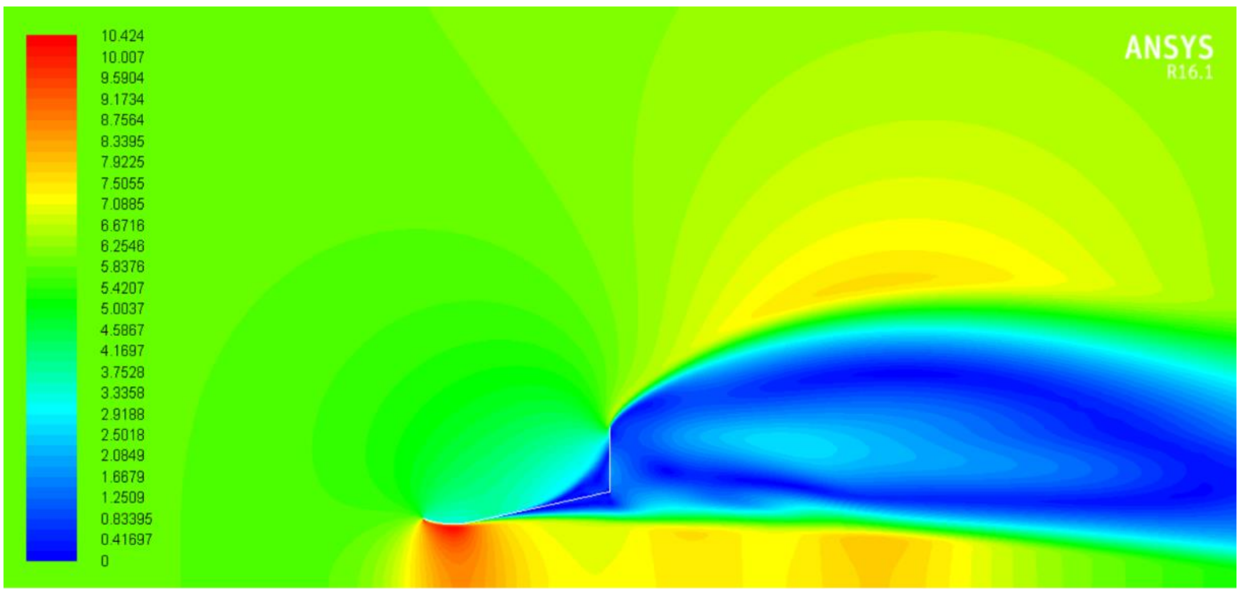

Figure 8. Shows the velocity contour of the entry flange radius $R_3 = ¾\, D = 45\ cm$.

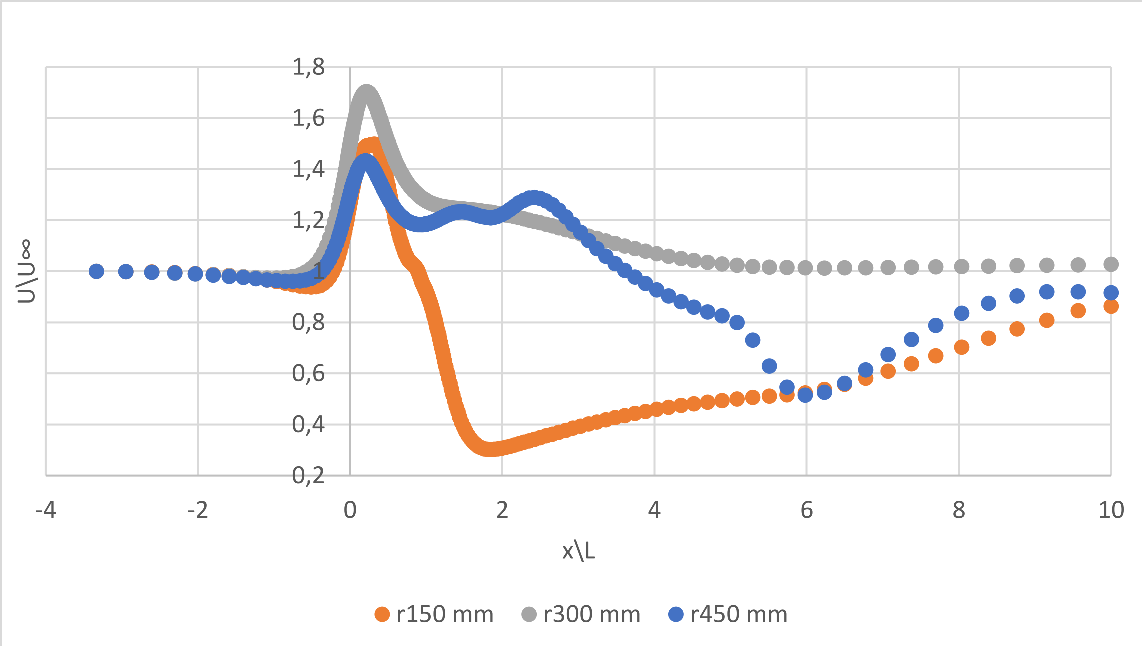

Figure 9. shows the flow velocity diagram in the three entry flanges.

It was found that the best diameter of entry flange is $D/2$ i.e. R= 300 mm, where it achieved the greatest speed and better stability. So it was adopted.

3.2. The effect of the height of the back flange (turbulence generator):

Seven different heights of the back flanged were examined:

$(h_0 = 0 * D = 0\ cm)$, $(h_1 = 0.075 * D = 4.5\ cm)$, $(h_2 = 0.125 * D = 7.5\ cm)$, $(h_3 = 0.25 * D = 15\ cm)$, $h_4 = 0.375 * D = 22.5\ cm)$, $(h_5 = 0.5 * D = 30\ cm)$, $h_6 = 0.625 * D = 37.5\ cm)$.

The most suitable rear back flanged height was tested integrally with the front flange that achieved the highest velocity at the throat.

Figs. 10 -11 -12 show the velocity contours of the back flange. Where we notice that the flow within the diffuser accelerates until the highest velocity value at the throat (is reached where the turbine is installed).

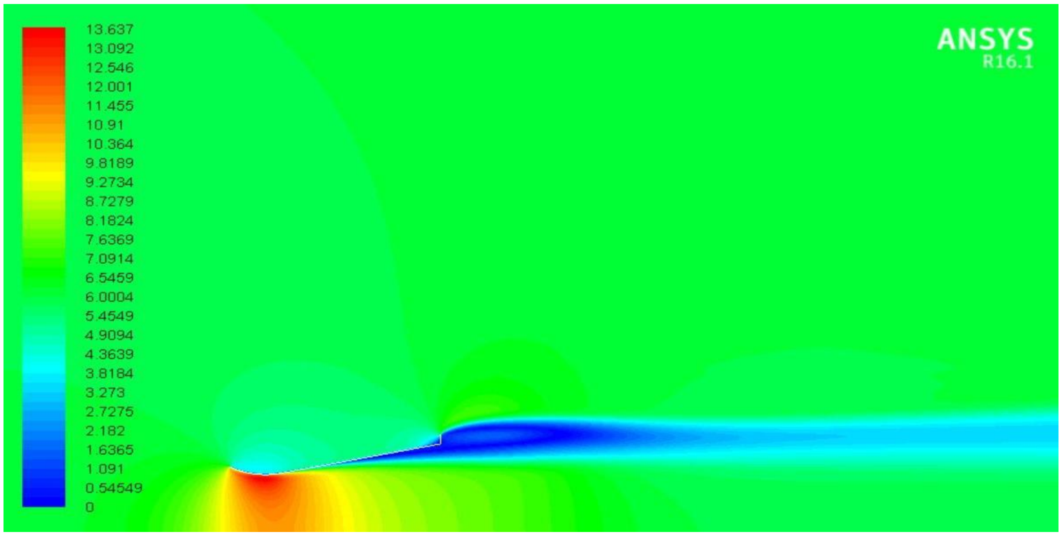

Figure 10. shows the velocity contour of the back flange (h1 = 0.075*D = 4.5 cm).

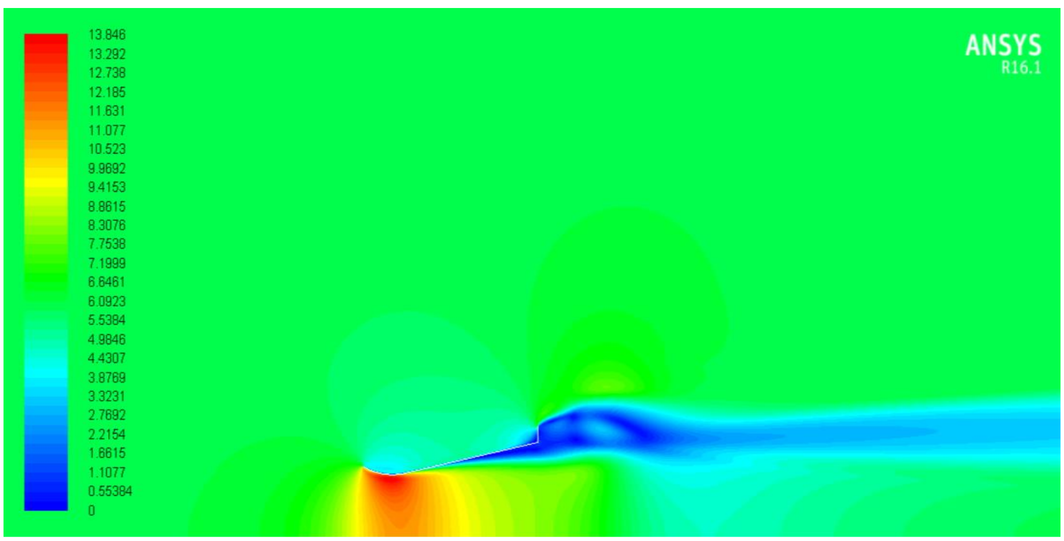

Figure 11. shows the velocity contour of the back flange (h_2 = 0.125*D = 7.5cm).

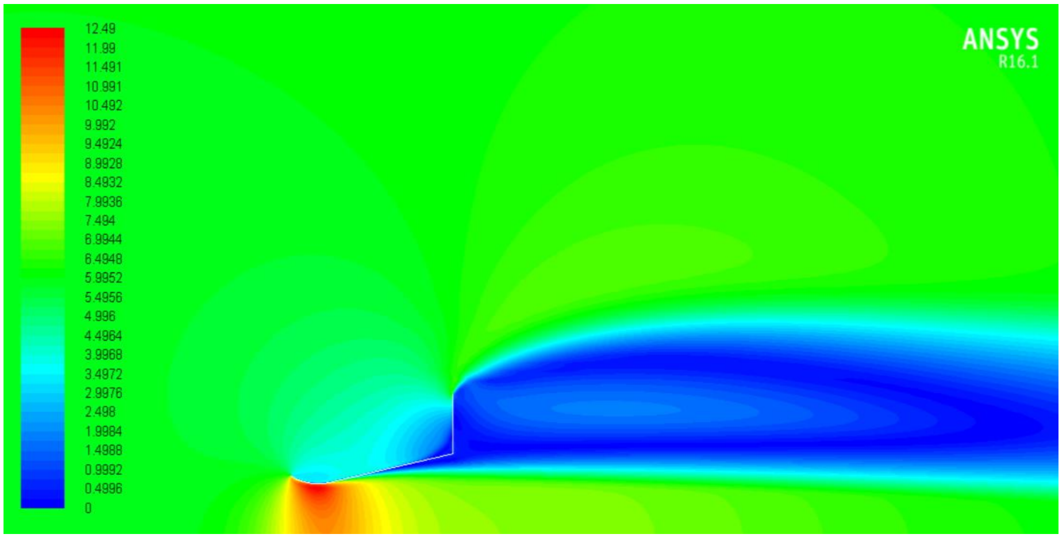

Figure 12. shows the velocity contour of the back flange (h_5 = 0.5*D = 30 cm).

A comparison between the wind velocities for the seven heights of the back flange is shown in Fig. 13. The velocity value increases with the increasing length of the back flange in the first three cases. Then, the velocity value decreases in the last four cases due to the height of the flange resulting from the effect of the vortices faraway behind the diffuser. The figure shows that the highest velocity value is 1.85 from the main flow velocity for a back flange whose height is equal to h_2 = 0.125*D.

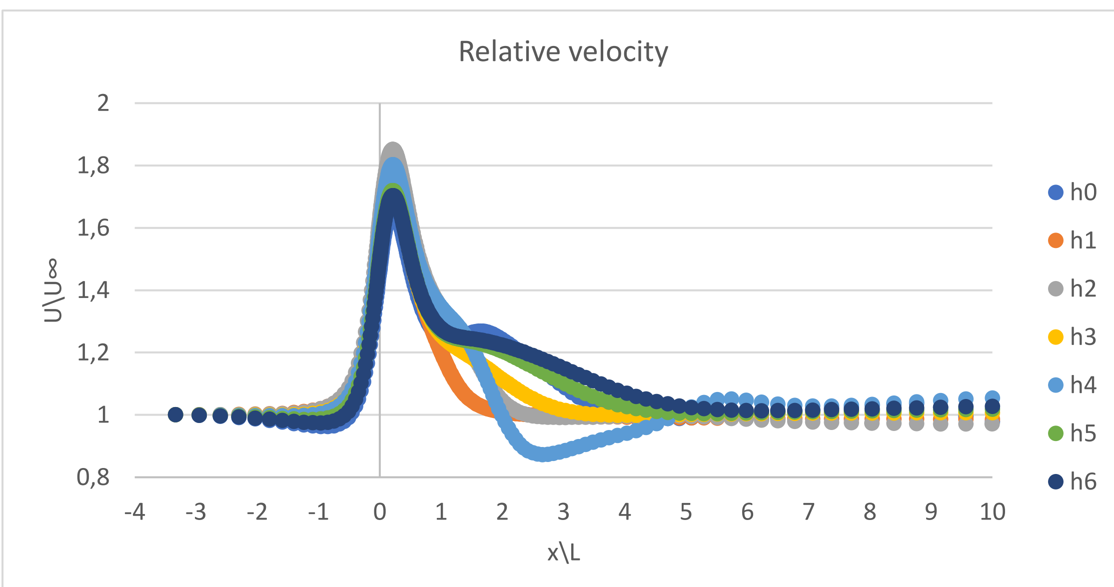

Figure13. relative velocity of the flow versus relative diffuser axis for the seven heights of the back flange.

Figure 14 -15-16 shows the pressure contours of the back flange. We notice an increase in the pressure difference between the entry and the exit area of the diffuser with an increase in the back flange in the first three cases, while the pressure difference decreases in the last four cases. We also notice that the highest value of pressure is on the external surface of the diffuser and it increases with an increase in the back flange, which is one of the disadvantages of this system.

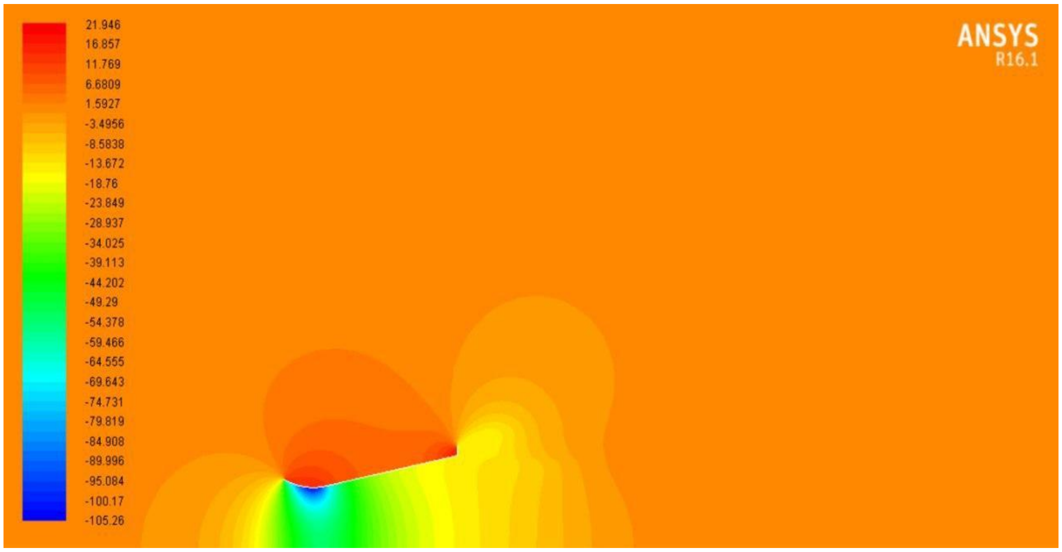

Figure 14. shows the pressure contour of the back flange (h1 = 0.075.D = 4.5cm).

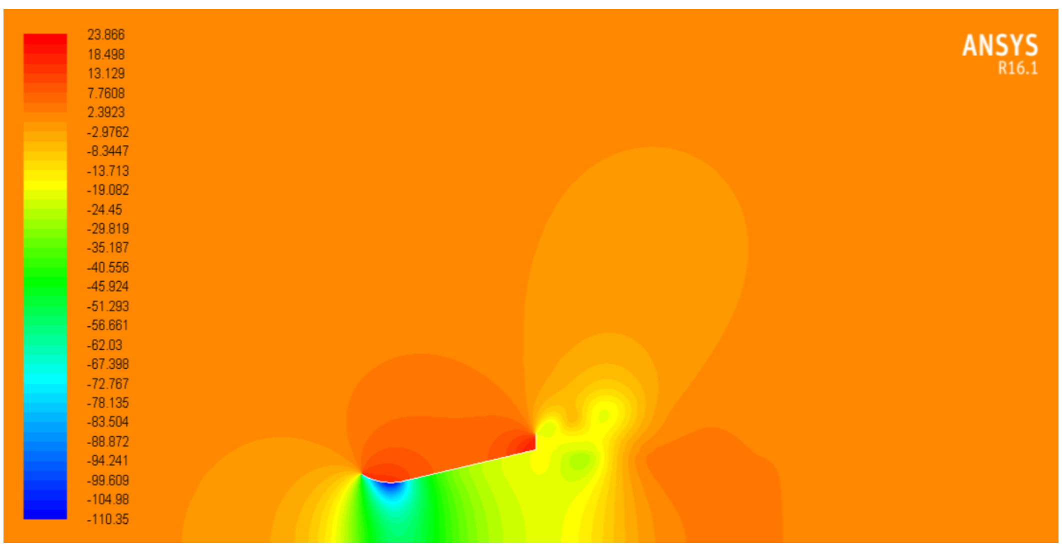

Figure 15. shows the pressure contour of the back flange (h_2 = 0.125.D = 7.5cm).

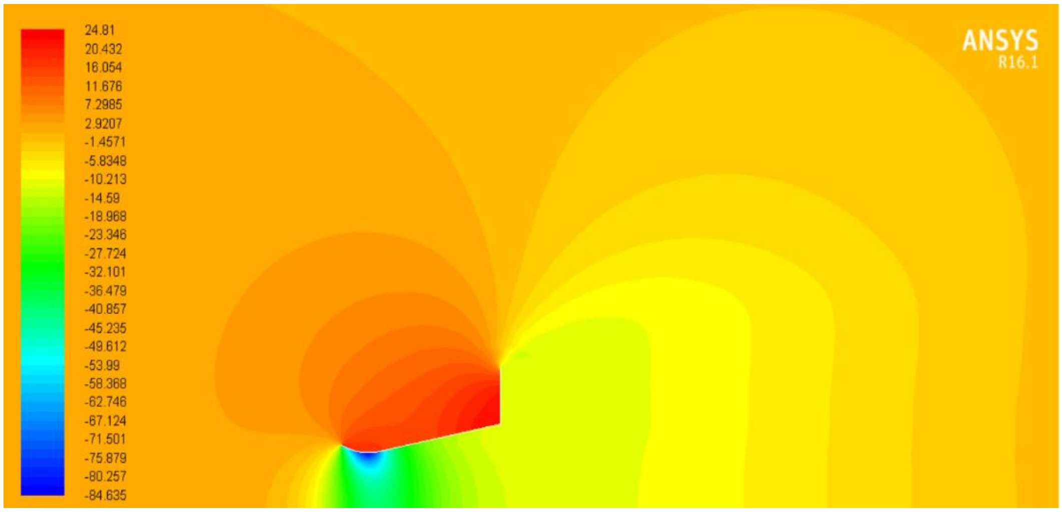

Figure 16. shows the pressure contour of the back flange (h_5 = 0.5.D = 30cm).

The figure17 shows a wind pressure coefficient comparison chart for the seven heights of the back flange. The figure shows that the wind pressure coefficient decreases at the throat in the first three cases while the pressure difference increases in the last four cases. The least pressure value is approximately -2.5 from the main flow pressure for a back flange whose height is equal to h_2 = 0.125*D.

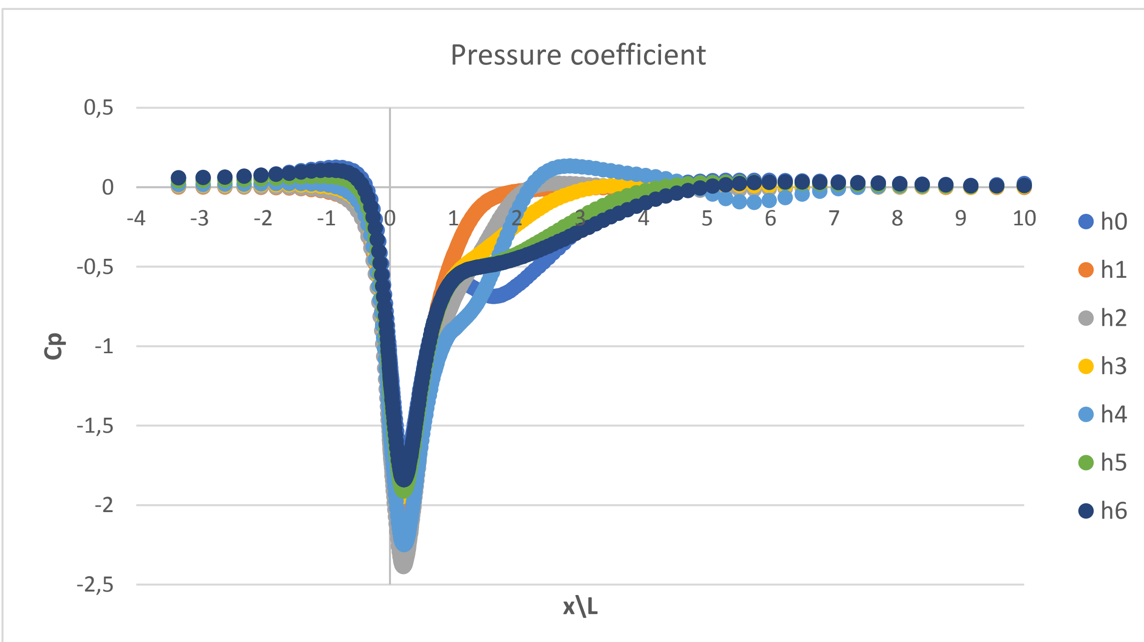

Figure 17. Scheme of comparing pressure coefficient of flow with the seven height of the back flange changing.

4. Conclusion

This project investigated approach to improve the utility of small wind turbines. By attaching a back flanged and entry flange diffuser shroud, the wind velocity at the blades is locally increased, so it will improve the energy production at lower wind speeds. The design of the shroud was accomplished through CFD modeling in ANSYS Fluent by isolating geometric factors and determining their influence on the performance

The optimum back flange height in the present study is h = 0.125*D, with a front flange of R = 0.5*D in radius within a diffuser of length L = 1.5*D, whereas without a front flange in the reference study, the height of the edge was h = 0.5 D [11].

The important features of this wind turbine equipped with a flanged diffuser shroud are as follows:

Four-fivefold increase in output power compared to conventional wind turbines due to concentration of the wind energy.

Significant reduction in wind turbine noise: Basically, an airfoil section of the turbine blade which gives the best performance in a low-tip speed ratio range is chosen. Since the vortices generated from the blade tips are considerably suppressed through the interference with the boundary layer within the diffuser shroud, the aerodynamic noise is reduced substantially [12].

Improved safety: The wind turbine, rotating at a high speed is shrouded by a structure. It is also safe against damage from broken blades.

5. Suggestions for Further Study

The immediate next step for this project would be the construction of a prototype in order to be tested in a wind tunnel.

References

[1] Lilley, G.M.; Rainbird, W.J. A Preliminary Report on the Design and Performance of Ducted Windmills; Report No. 102; College of Aeronautics: Cranfield, UK, 1956.
[2] Gilbert, B.L.; Oman, R.A.; Foreman, K.M. Fluid dynamics of diffuser augmented wind turbines. J. Energy 1978, 2, 368–374.
[3] Gilbert, B.L.; Foreman, K.M. Experiments with a diffuser-augmented model wind turbine. Trans. ASME, J. Energy Resour. Technol. 1983, 105, 46–53.
[4] Igra, O. Research and development for shrouded wind turbines. Energ. Conv. Manage. 1981, 21, 13–48.
[5] Phillips, D. G., Richards, P. J., Flay, R. G. J. CFD modelling and the development of the diffuser augmented wind turbine. Wind Struct., 2002, 5, 267–276.
[6] Phillips, D.G.; Flay, R.G.J.; Nash, T.A. Aerodynamic analysis and monitoring of the Vortec 7 diffuser augmented wind turbine. IPENZ Trans. 1999, 26, 3–19.
[7] Bet, F.; Grassmann, H. Upgrading conventional wind turbines. Renew. Energy 2003, 28, 71–78.
[8] HANSEN M O.L., 2008 - Aerodynamics Of Wind Turbine. Earthscan,2nded.,London,175 pages.
[9] PHILLIPS D . G.,2003 - An Investigation On Diffuser Augmented Wind Turbine Design. PhD Thesis, Department Of Mechanical Engineering, The university Of Auckland, New Zealand.
[10] BUSSEL V and GERARD J. W.DR.,2007 – The Science Of Making More Torque From Wind Diffuser Experiments And Theory Revisited. Journal of Phsyics,75, 1-12.
[11] ohya, Y., Karasudani, T., Sakurai, A., Abe, K. & Inoue, M. (2008), 'Development of a shrouded wind turbine with a flanged diffuser', Journal of Wind Engineering and Industrial Aerodynamics 96(5), 524 – 539.
[12] Abe, K.; Kihara, H.; Sakurai, A.; Nishida, M.; Ohya, Y.; Wada, E.; Sato, K. An experimental study of tip-vortex structures behind a small wind turbine with a flanged diffuser. Wind Struct. 2006, 9, 413–417.