Development of Diffuser Designs for a Diffuser Augmented Wind Turbine

D.G. Phillips[1], P.J. Richards[2], R.G.J. Flay[3]
[1]Postgraduate Student, [2]Senior Lecturer and [3]Associate Professor
Department of Mechanical Engineering, The University of Auckland
New Zealand

Abstract

This paper describes the development of diffuser designs for a Diffuser Augmented Wind Turbine (DAWT) using computational fluid dynamics (CFD). A CFD model using PHOENICS has been developed to allow the assessment of various geometrical modifications for performance enhancement of the Vortec 7 DAWT. The investigations of diffuser designs have resulted in development of a new geometry. A wind tunnel model of this design has been made and initial testing has been performed. A comparison of the CFD models with field analysis and wind tunnel results is presented in this paper.

Introduction

A DAWT has a duct which surrounds the wind turbine blades and increases in cross-sectional area downstream of the blade plane. The increase in duct area downstream results in a reduction in the mean velocity of the flow through the diffuser due to conservation of mass. From Bernoulli's equation, an increase in static pressure through the diffuser must occur for isentropic flow. With the exit of the DAWT being on the leeward side of this obstruction to the flow, the resulting base or exit pressure can be assumed to be slightly sub-atmospheric. Therefore, at the narrower plane immediately downstream of the blades, the pressure will be even lower. More air is expected to be drawn through the blade plane due to this low pressure at the inlet in comparison with a bare turbine. The anticipated effect on the stream-tube passing through the blade plane (Foreman, 1979) can be seen in Figure 1.

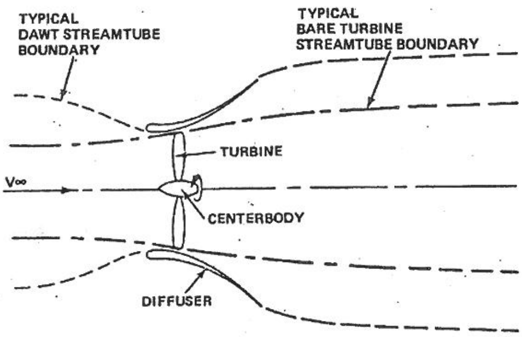

Figure 1 : Effect of diffuser on stream-tube passing through blade plane (Foreman, 1979)

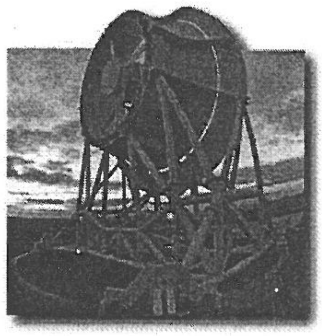

Figure 2 : The Vortec 7. Flow is from left to right. The curved boundary layer control slot is seen near the outlet on the right.

The design developed under K.M. Foreman in the 1970's and early 1980's enabled a substantial reduction in the diffuser size (Gibert et al. 1978). This was achieved by controlling the boundary layer within the diffuser and preventing separation along the diffuser wall. High speed external flow was injected along the diffuser wall through slots to energise the boundary layer. Foreman was able to produce a design which incorporated a large exit to inlet area ratio, high diffuser angle and a compact, short length to diameter, diffuser (Foreman et al. 1983). This optimal design was the basis of the first full scale DAWT built by Vortec Energy Limited. The Vortec 7 (Figure 2) has a blade diameter of 7.3m and was installed 110 km south of Auckland, New Zealand in early 1997 (Nash, 1997). With a Technolgy for Business Growth awarded to Vortec in conjunction with The University of Auckland (AU) and Industrial Research Limited (IRL) a program of monitoring and optimisation of the demonstrator unit began (Phillips et al. 1998).

A CFD model was developed at AU for comparison with the Vortec 7. Flow visualisation and data were used to validate the initial CFD model and it was then used to test various geometric variations to the Vortec 7 diffuser. It was found to be a cost effective tool for the improvement of the Vortec 7 performance and for the development of diffuser concepts for a Vortec 23 (blade diameter 23 m).

Computational Fluid Dynamic Model

The development of a CFD model at The University of Auckland was made in parallel with the Vortec 7 site testing. The fully viscid, finite volume code PHOENICS has been used. The model is axi-symmetric with specified inlet conditions and reference length making the model dimensionless. A body fitted grid is used to reproduce the complex geometry of the DAWT with the main features shown in Figure 3.

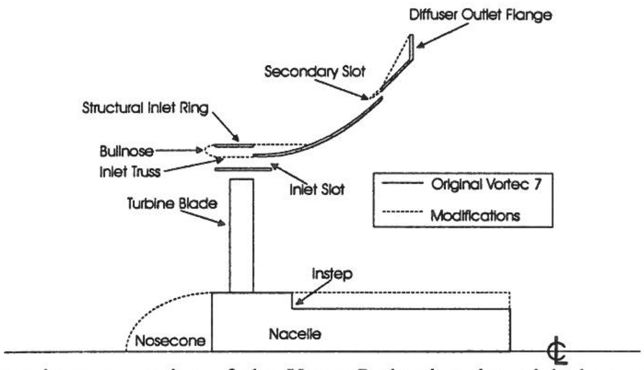

Figure 3 : Axi-symmetric cross section of the Vortec 7 showing the original geometry and subsequent modifications.

The k-ε turbulence model has been used with wall friction applied to all geometric surfaces. The values of k and ε have been calculated for the hub height of the Vortec 7 and terrain in which it is situated (Richards and Hoxey, 1993). For comparison with wind tunnel investigations, suitable levels of turbulence have also been examined for the development diffuser. The turbine has been modelled as a flow resistance analogous to the use of gauze screens in wind tunnel testing (Gilbert et al. 1978). The pressure drop across the blade plane is calculated from a specified turbine thrust coefficient and is proportional to the local dynamic pressure. Swirl and other features such as tip vortices are not incorporated into the model as the focus of the work was the rapid development and assessment of diffuser designs. The inlet truss on the Vortec 7 has been modelled by a shear stress acting in both the axial and radial directions over the cells in the inlet boundary layer control slot. The shear stress is scaled by the area ratio of the triangular geometry to the cell area in order to match the CFD model to the physical Vortec 7.

Modification of the Vortec 7

The CFD model of the original Vortec 7 showed areas of separation downstream of the nacelle and diffuser outlet flange as well as reversal of flow around the structural inlet ring. These areas of separation were observed from flow visualisation on the Vortec 7 using smoke testing and spinnaker cloth tufts. A bull nose was designed to prevent the flow reversal and improve the inlet slot flow. The secondary slot was modified with the addition of an inlet to direct flow parallel to the diffuser inner wall. These modifications showed improvements to the flow with the secondary slot flow adhering over the entire length of the secondary diffuser inner wall. Intermittent flow separation was still observed over the trailing third of the primary diffuser. CFD modelling of a nosecone for the nacelle and improved streamlining of the nacelle was performed. The radial flow resulting from the blunt nacelle was directed axially along the nacelle with the addition of a nosecone. Although improving the flow along the nacelle, separation was observed from flow visualisation. The step decrease in cross sectional area from the circular hub to the pentagon shaped nacelle provided a point for the flow to separate. This caused a region of tubulent flow adjacent to the nacelle and reduced the expansion of core flow through the blade plane is it moved downstream through the diffuser. The reduced expansion of the core flow meant that the inlet speed-up was reduced and hence a reduction in diffuser performance. The step change in geometry was smoothed with foam sections mounted to the outside of the nacelle. Although these modifications improved performance, the high cost of further retrofits meant that development of other diffuser concepts using the CFD modelling would be more cost effective.

Development of Diffuser Concepts

The modelling of the Vortec 7 and the flow visualisation on site identified the importance of both inlet and slot flow on the performance of a DAWT. Under field conditions the onset flow is turbulent with variation in magnitude and direction. The original secondary slot configuration of the Vortec 7 did not control the direction of the injected flow. With variation in the velocity and direction of the flow, the flow would at times blow the core flow away from the diffuser wall. Another area effected by the variation in onset flow was at the blade plane. With no control of the flow into the diffuser, it was found that there was substantial variation in the flow at the blade plane. Separation on both the primary diffuser wall and nacelle would result when the flow entered the diffuser at an angle. With these factors in mind, a new diffuser geometry was developed using the CFD model (Figure 4). A controlled inlet contraction of the flow into the blade plane has been introduced with a slot between the blade tip and primary diffuser wall. This gap allows the accelerated flow adjacent to the primary diffuser to continue downstream of the blade plane and keep the flow attached to the diffuser wall. A second slot directs flow tangentially along the secondary diffuser. An increase in exit area compared to the Vortec 7 has been examined with these modifications producing a substantial improvement in available air power. An aerodynamic profile for both the primary and secondary diffusers enables the controlled flow contraction and diffusion and provides structural benefits for design. The nacelle has been streamlined to prevent the separation of flow downstream as found on the Vortec 7. To validate the new diffuser geometry, a wind tunnel investigation has been undertaken for comparison with the CFD model.

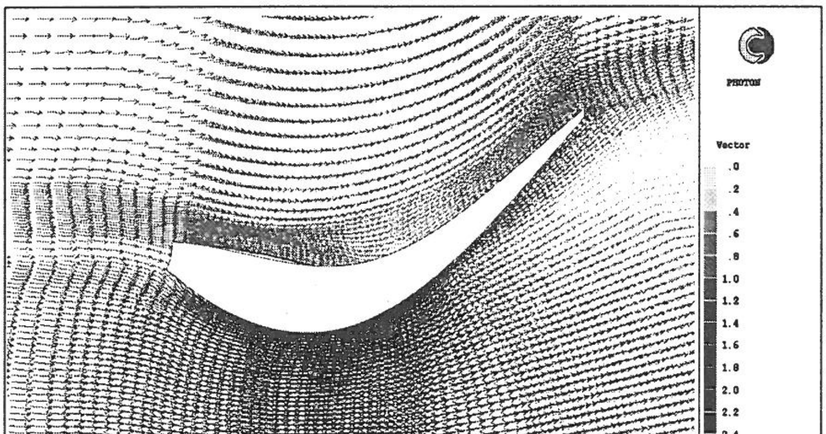

Figure 4: Velocity Filed through development DAWT. The geometry is hidden for proprietary reasons.

Wind Tunnel Testing and Comparison

A wind tunnel model with a 457 mm blade diameter has been built using the geometry developed from the CFD modelling at AU. A three dimensional ProEngineer model was used for the production of the wind tunnel model. Testing was performed in a closed section 3 by 2 m wind tunnel at Vipac Engineers and Scientist Limited, Melbourne. Gauze screens were used to produce a known pressure drop at the blade plane. The total pressure drop across the blade plane, the velocity speed-up and base pressure were measure using pitot tubes referenced to a pitot tube in the upstream ambient flow. A hot-wire anemometer was used to measure the velocity profile radially across the blade plane. The velocity speed-up pitot was located approximately 2/3 the way between the hub and blade tip with the total pressure pitots evenly spaced across the blade radius. These measurements provided the necessary data to characterise the performance of the DAWT and allow comparison with the CFD results. Figure 5 shows the variation in power coefficient, velocity speed-up at the blade plane and base pressure coefficients for various disc loadings.

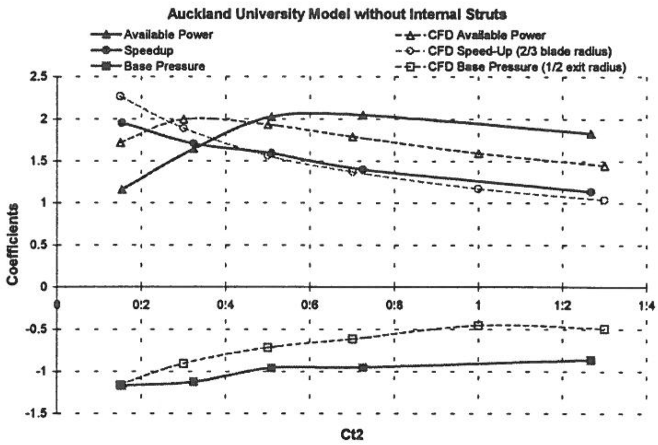

Figure 5 : Wind tunnel results and comparison with CFD power predictions.

It can be seen that large velocity speed-ups are achieved as the disc loading decreases. A lower than expected base pressure was recorded which was possibly due to testing in a closed section wind tunnel. The effect of this will be investigated by further testing in a larger wind tunnel. The magnitude of the power coefficient agrees with that predicted by the CFD with the slight increase resulting from the blockage effect of the wind tunnel. The peak performance occurred at a higher disc loading than was predicted by the CFD and further data analysis and testing will be necessary to determine the reason.

Conclusions

A computational fluid dynamic (CFD) model has been developed for the analysis of diffuser designs for a DAWT. The results for the CFD models show good agreement with flow visualisation of the Vortec 7. The use of the CFD model to examine variations to the Vortec 7 geometry enabled cost effect modifications to be made. A new diffuser geometry has been developed with increases in available air power predicted compared with the Vortec 7. A wind tunnel investigation of this new diffuser design shows velocity speed-ups and power coefficients similar to those predicted by the CFD modelling.

References

FOREMAN, K. M. "Cost Analysis of DAWT Innovative Wind Energy Systems", *Research Department Report RE-626AJ*, p532, 1979

FOREMAN, K.M., MACIULAITIS, A. and GILBERT, B.L., "Performance Predictions and Recent Data for Advanced DAWT Models", *ASME Solar Energy Division, Grumman Aerospace Corp.*, Bethpage, New York, April 1983

GILBERT, B.L, OMAN, R.A. and FOREMAN, K.M. "Fluid Dynamics of Diffuser-Augmented Wind Turbines", *J. Energy*, Vol 2, 6, pp368-374, 1978

NASH, T.A. "Design and Construction of the Vortec Seven", *NZ Wind Energy Association, First Annual Conference,* Wellington, June 1997

RICHARDS, P.J., HOXEY, R.P. "Appropriate boundary conditions for computational wind engineering models using the k-ε turbulence model", *J. Wind Engng and Ind. Aerodyn.*, 46 & 47, 145-153, 1993

PHILLIPS, D.G., FLAY, R.G.J., NASH, T.A. "Aerodynamic Analysis and Monitoring of the Vortec 7 Diffuser Augmented Wind Turbine", *IPENZ Conference*, Auckland, New Zealand, February 1998.