13th Australasian Fluid Mechanics Conference
Monash University, Melbourne, Australia
13-18 December 1998

COMPUTATIONAL MODELLING OF DIFFUSER DESIGNS FOR A DIFFUSER AUGMENTED WIND TURBINE

D.G. PHILLIPS[1], P.J RICHARDS[2], G.D. MALLINSON[3] AND 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

The paper outlines the use of computational fluid dynamic (CFD) code PHOENICS in the development of diffuser designs for a diffuser augmented wind turbine (DAWT). The purpose of the modelling is the rapid development of the diffuser design for improved air power coefficients. Comparison of CFD results and full scale site measurements of the Vortec 7 are described. Subsequent modifications made to the Vortec 7 after CFD modelling are discussed and results for it and a new development diffuser are presented.

INTRODUCTION

The Vortec 7 is the first full scale diffuser augmented wind turbine to be built. A DAWT has a duct which surrounds the wind turbine blades and increases in cross-sectional area further downstream. The resulting sub-atmospheric pressure within the diffuser draws more air through the blade plane, and more power can be generated compared to a "bare turbine" of the same rotor blade diameter.

The technology demonstration unit (Vortec 7) built by Vortec Energy Limited has a rotor blade diameter of 7.3m and is situated near the Franklin west coast, 120 km south of Auckland, New Zealand. The Vortec 7 design is based on work performed at the Grumman Aerospace Corporation in the 1970's and early 1980's (Foreman and Gilbert, 1983). The use of High Tensile Reinforced Fibrous Ferrocement (HT Ferro) enabled the production of the DAWT (Nash, 1997) which had previously not been economically viable. The optimal design found under K. M. Foreman, (Foreman, Maciulaitis and, Gilbert 1983) incorporated a short length-to-diameter diffuser with a large outlet-to-inlet area ratio. The use of bleed flow through two boundary layer control slots enabled the high speed external flow to energise the boundary layer inside the duct and prevent separation. The as-built Vortec 7 is shown in Figure 1 with the main geometric features shown in Figure 2.

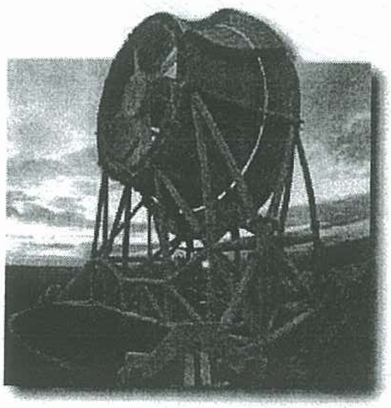

Figure 1 : The Vortec 7. Flow is from left to right. The curved boundary layer slot is seen near the outlet on the right.

A Technology for Business Growth (TBG) grant was awarded to Vortec Energy Limited in conjunction with Industrial Research Limited (IRL) and the University of Auckland (AU) for monitoring and optimising the demonstrator unit. Data from the Vortec 7 have been used for verification of the CFD model developed at AU.

The CFD model has been developed in order to test various geometric variations to the Vortec 7 diffuser. It has been used as 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) with significantly improved performance compared to the Vortec 7.

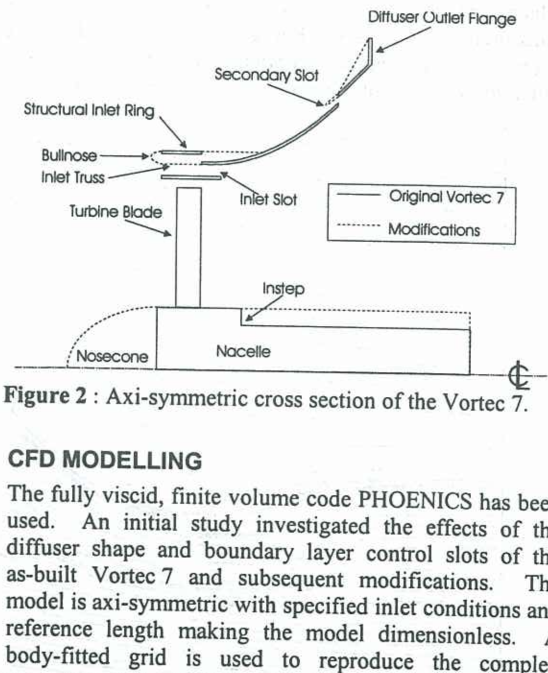

Figure 2 : Axi-symmetric cross section of the Vortec 7.

CFD MODELLING

The fully viscid, finite volume code PHOENICS has been used. An initial study investigated the effects of the diffuser shape and boundary layer control slots of the as-built Vortec 7 and subsequent modifications. 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 turbine modelled as a flow resistance. The turbine has a specified thrust coefficient which produces a pressure drop across the blade plane proportional to the local dynamic pressure. This is analogous to the use of a gauze screen in wind tunnel testing (Gilbert, Oman and Foreman, 1978). The flow is assumed to have no swirl with only the global effect of the blades having been modelled. Features such as tip vortices and flow swirl have been omitted in order to reduce the computation time. These assumptions allowed the use of the CFD modelling for rapid development of diffuser designs. The inlet truss at the Vortec 7 inlet 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. The k-ε turbulence model has been used with uniform inlet boundary conditions specified for k and ε. The values of k and ε have been calculated for the hub height and terrain in which a turbine would be situated (Richards and Hoxey, 1993). Turbulence levels comparable to those found in a wind tunnel have also been examined for comparison with future wind tunnel investigations.

CFD COMPARISON WITH VORTEC 7 SITE DATA

An initial CFD model using the as-built Vortec 7 geometry was developed for comparison (Phillips, Flay and Nash, 1998). The results obtained were compared with flow visualisation using smoke and spinnaker cloth tufts attached to the walls of the diffuser. At that time there were no data available for quantitative comparison. The CFD model correctly predicted separation downstream of the nacelle and the diffuser outlet flange. Another important observation was the reversal of flow around the structural inlet ring of the Vortec 7 which is shown in Figure 3. This was also seen during smoke testing of the Vortec 7. It was found that the CFD model also predicted the high velocity speed up through the inlet slot and the radial variation of flow speed across the blade plane which was observed in preliminary site measurements. Vortec Energy Limited modified the Vortec 7 by adding an elliptical nosecone, a bull-nose fitted to the inlet slot to prevent the flow reversal observed initially, and modifying the secondary inlet slot and diffuser to direct flow tangentially to the diffuser wall. These modifications were incorporated into the CFD model together with an instep in the centrebody to match the change in geometry from the circular spinner to the pentagon shaped nacelle. Both quantitative and qualitative results were compared between the CFD model and the Vortec 7 site data. Flow visualisation showed regions of separation downstream of the spinner along the nacelle. Intermittent separation was also observed on the trailing third of the primary diffuser with strong secondary slot flow adhering to the secondary diffuser wall. The separated regions agreed well with those predicted by the CFD model. The effect of the disc loading was studied and showed that at high disc loading the flow was forced out toward the diffuser and reduced the separated region on the diffuser. This effect was observed with the Vortec 7 by varying the rotational speed of the turbine. Velocity and directional anemometry sensors located on a boom at the diffuser exit clearly showed the separated region next to the nacelle and the high diffusion angles outside that region. These observations confirmed the streamline plots obtained with the CFD model.

The parameters of interest which characterise the performance of the DAWT are the velocity speed up at the turbine plane and the power coefficient. Comparison between the CFD model and site data of these parameters shows good agreement.

The inlet speed up found from the present CFD and Vortec 7 studies disagreed with the predictions of Grumman (Foreman and Gilbert, 1983, Foreman, Gilbert

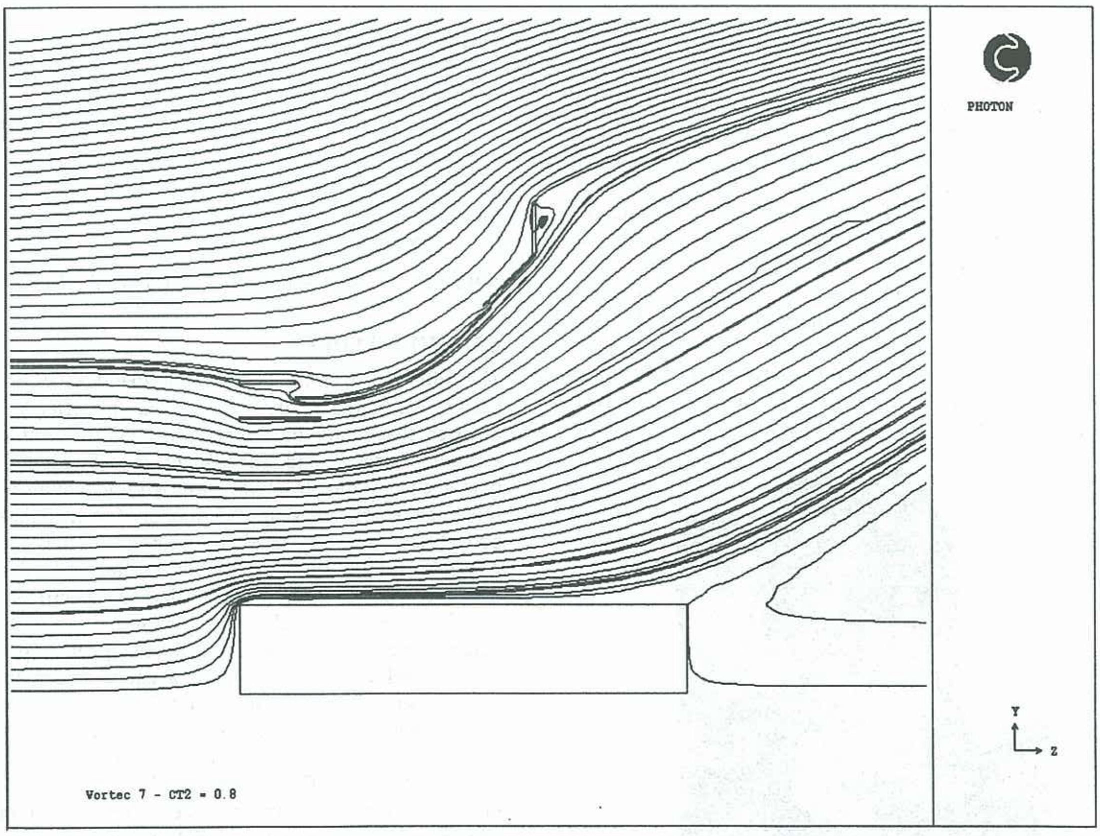

Figure 3 : Streamline plot showing flow reversal through inlet truss for the original Grumman design.

and Maciulaitis, 1983). To obtain the maximum speed-up for power production the flow velocity through the diffuser must decrease and leave the diffuser at a slightly sub-atmospheric pressure. With the large separation along the nacelle, the change in effective area through the diffuser is reduced and hence lower speed-ups resulted. To improve the diffusion of flow, streamlining of the nacelle was examined. A smooth centrebody was incorporated into the CFD model having the same radius as the spinner. The effect previously found due to varying the disc loading was also shown with this geometric configuration. For a local disc loading coefficient between 0.8 and 1.0, the flow remained attached to both the diffuser wall and nacelle. A doubling of the power coefficient at the optimal operating point was achieved by ensuring the flow did not separate. An increase in power was found for all the disc loadings modelled.

The Vortec 7 was modified to match the smooth nacelle tested in the CFD model by the addition of foam sections. The modified Vortec 7 had reduced separation along the nacelle. A radial variation in flow direction across the exit plane was observed with axial flow near the nacelle and a gradual turning in direction to be parallel to the diffuser at the wall. These agreed with the streamlines from the CFD model, and both the model and Vortec 7 exhibited similar degrees of separation for various disc loadings. The magnitudes of the power increase and inlet speed-up measured from the Vortec 7 agreed with that predicted by the CFD. The velocity field predicted for the Vortec 7 with smooth nacelle is shown in Figure 4. The inlet velocity is set at unity and the vector magnitudes represent the velocity speed up caused by the diffuser. A 73% increase in velocity can be seen at the inlet boundary layer control slot. A speed-up over the blade plane of 10% at the nacelle up to 20% at the blade tip has been found for the modified Vortec 7. This is a significant increase compared to a bare turbine where for an ideal Betz machine, the velocity at the blade plane is reduced to 66% of the free stream velocity. The effect of the secondary slot introducing high speed flow is evident, with the flow remaining attached along its entire length.

DEVELOPMENT OF THE DAWT

The development of a geometry for future designs is being undertaken using CFD modelling. The CFD model of the Vortec 7 has been the starting point for this development. The effect of the inlet slot, diffuser outlet flange, nacelle shape and size have been examined using the existing area ratio and diffuser exit angle. A new design has been developed which introduces controlled contraction of the flow into the blade plane. The primary diffuser has been extended upstream and the inner inlet ring removed. The area ratio and diffuser angle have been altered for this new geometry and the diffuser exit flange removed. Increases in power coefficient by a factor of approximately four compared with the original design have been found. A wind tunnel investigation of this new geometry is proposed to validate these promising results.

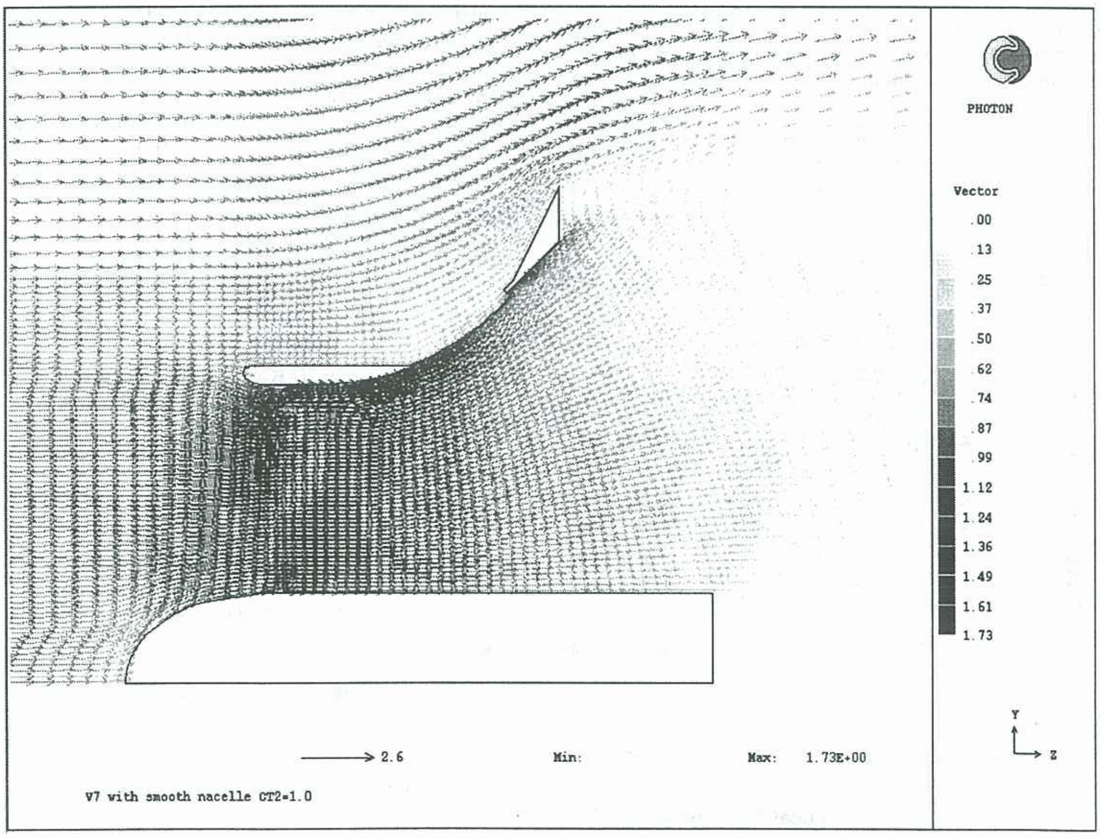

Figure 4 : Velocity field through modified Vortec 7 with smooth nacelle.

COMPARISON WITH ONE-DIMENSIONAL ANALYSIS

A simple one-dimensional model has been developed in conjunction with this research. It consists of a free isentropic contraction into the diffuser inlet, a specified pressure drop coefficient at the blade plane (analogous to the CFD turbine model), a specified diffuser efficiency, and a specified base pressure coefficient. This can be solved to determine the air power potential for turning wind turbine blades. Comparison of the CFD results for the as-built Vortec 7 with the one-dimensional analysis showed considerable difference in both the magnitude of the speed-up, the corresponding power coefficient, and the local disc loading at which optimal power was achieved. This comparison is shown in Figure 5. An optimal power coefficient around 2.0 is obtained with a local disc loading coefficient between 0.3 and 0.4 for the one-dimensional theory assuming a diffuser efficiency of 95%, a base pressure coefficient of -0.6 and an exit area ratio of 3. The CFD with modified Vortec 7 geometry predicts an optimal power coefficient of approximately 1.3 at a local disc loading between 0.8 and 1.0. An increase in power coefficient of over double was predicted from the modifications to the original Grumman design. The disc loading trend shown by the Vortec 7 geometry was not shown with the new configuration modelled. The new geometry with controlled inlet contraction displayed similar trends to those predicted by the one-dimensional analysis. Much larger inlet speed-ups were predicted with the optimal power coefficient around 1.9 occurring at a local disc loading between 0.3 and 0.4. This compares to the one-dimensional power coefficient prediction of 2.3 for the larger exit area ratio of 4. The development DAWT obtains approximately 85% of the power coefficient predicted by the one-dimensional analysis. This is an increase from the 65% obtained by the modified Vortec 7 in comparison with the one-dimensional theory.

CONCLUSIONS

The Vortec 7 is based on Grumman's optimal design developed by wind tunnel tests performed in the 1970's and early 1980's. A computational fluid dynamic (CFD) model has been developed to enable cost effective development of the Vortec 7. Vortec 7 site data and flow visualisation have been used to validate the CFD model. Modifications to the inlet slot and nacelle were made to the Vortec 7 after CFD investigations showed they would give improved performance. Variations in the nacelle size and shape, inlet slot size, inlet-to-outlet area ratio and new diffuser geometries have been investigated using the CFD model. A large increase in power coefficient compared with the original design is predicted from the CFD for a new development diffuser.

REFERENCES

FOREMAN, K.M., GILBERT, B.L. "Experiments with a Diffuser-Augmented Model Wind Turbine", *J. Energy Resour Technol Trans. ASME*, Vol 105, **3**, 46-53, 1983

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**, 368-374, 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.

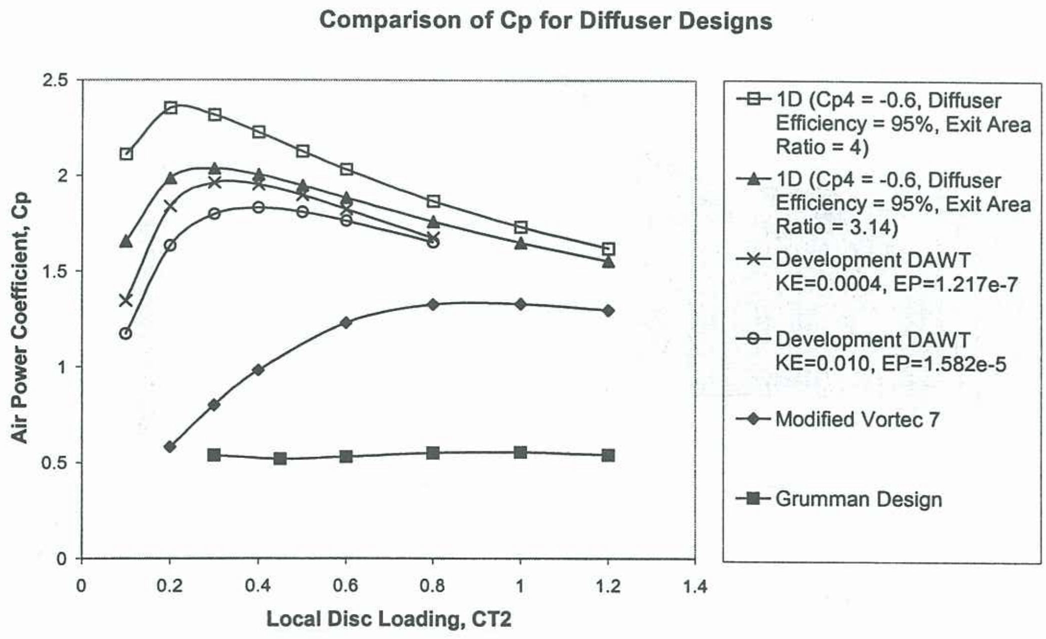

Figure 5 : Comparison of CFD results with one-dimensional analysis.