

DIFFUSER DEVELOPMENT FOR A DIFFUSER AUGMENTED WIND TURBINE USING COMPUTATIONAL FLUID DYNAMICS

D.G. Phillips, P.J. Richards, R.G.J. Flay

Department of Mechanical Engineering
The University of Auckland
New Zealand

Abstract

Vortec Energy Limited and The University of Auckland have carried out significant work on developing the diffuser design for a Diffuser Augmented Wind Turbine (DAWT). The use of PHOENICS has been an essential element in this development, as it has allowed rapid investigation of a range of concepts. In addition wind-tunnel tests using both gauze screens and turbine blades have been performed to validate the use of Computational Fluid Dynamics (CFD) in the design of the diffuser.

This work is a continuation of the CFD modelling of the Vortec 7 prototype DAWT. The CFD model developed for the Vortec 7 was successfully used to predict the flow behaviour within the diffuser. Examination of proposed modifications to predict their potential performance improvement was carried out using PHOENICS and modifications exhibiting the most potential were then made to the prototype Vortec 7. With several key features of the diffuser design identified in the Vortec 7 analysis, development of an advanced diffuser began. A controlled contraction of the flow into the diffuser was incorporated, together with modifications to the inlet and secondary slots. Various inlet and exit to throat area ratios were examined over a range of turbine disc loading coefficients. The best design identified from this work was used in a series of wind-tunnel tests to evaluate the CFD modelling. Results from the wind-tunnel have shown up

deficiencies in the CFD model and have led to refinements of the model and the development of further design concepts.

It is anticipated that as the diffuser design progresses computational modelling will continue to play an important role alongside wind tunnel and prototype testing.



CFD comparison with Vortec 7 site data

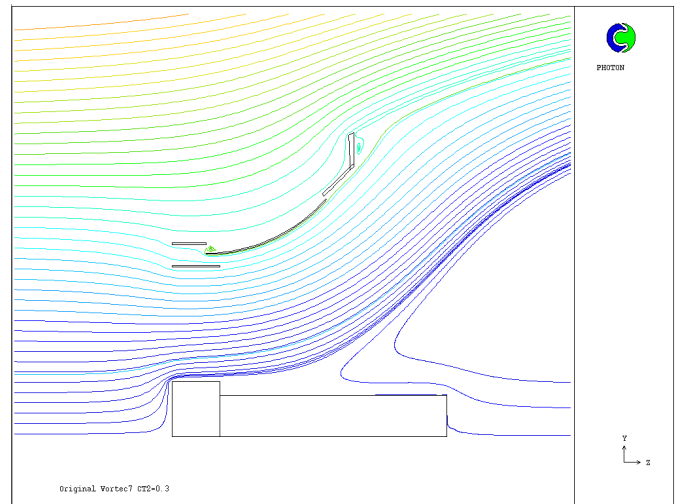
The initial CFD model was of the as-built Vortec 7 and the results obtained were compared with flow visualisation using smoke and spinnaker cloth tufts attached to the walls of the diffuser. The CFD model correctly predicted separation downstream of the nacelle and the diffuser outlet flange. Another important observation was the reversal of flow around the structural inlet ring

of the Vortec 7 which is shown in Figure 1(b). 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 modified 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 centre body 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, as illustrated in Figure 2.

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.



A full copy of this work is available by contacting CHAM.