CFD analysis of a 5kW portable Marine Wind Turbine

Dr. Cian Desmond, UCC, Dr. Paul Leahy, UCC, Mr. Jason McCorkell, UCC.

INTRODUCTION

DARE Technology Limited are developing a portable marine wind turbine with the aim of reducing emissions from marine vessels by providing an alternative source of energy generation when the vessels are experiencing downtime. As part of the development of the solution it is necessary to determine the expected wind characteristics that will be experienced by the turbines mounted on the vessels. To this end, CFD analysis has been conducted in which we consider a typical marine vessel on which a portable marine wind turbine will be deployed. This analysis is used to determine the suitability of a number of possible mounting locations for the turbine. Four turbine locations and two vessel orientations were investigated.

APPROACH

The concept being developed by DARE D involves attaching a 5kW portable marine wind turbine with a 5m rotor diameter to a fixture on amarine vessel when the vessel is experiencing down-time. The portable marine wind turbine will then contribute to the running of the vessel at anchor. The hub height of the turbine will be dependent on the type of vessel and the available equipment onboard. The turbine will then provide the vessel with an alternative source of power generation whilst the wind turbine is within its operating range.

A typical anchored vessel will orientate itself into the wind with either the front or the rear of the vessel facing into the wind, so this research will consider only these two orientations of the vessel, scenario 1 and scenario 2 respectively.

Typically, marine service vessels cannot undertake lifting or crew transfer operations when wave heights are greater than 2m or in wind speeds greater than 12m/s, measured at 20m above sea level directly above the vessel [2], [5]. For the turbine considered in this investigation that warrants an operational window of wind speeds in conditions from 12m/s at 20m above sea level, up to 25m/s at hub height at which point the maximum operational wind speed is reached [2].

MAIN BODY

The software package ANSYS CFX was used to simulate various wind conditions over a typical small-medium sized marine services vessel, using the Reynolds averaged forms of the Navier-Stokes equations.

Simulations in each of the two scenarios in the investigation were made for wind speeds of 12.5m/s – 25m/s at the hub height of the rear position for the portable marine wind turbine. Simulations were run for wind speeds within these limits in 2.5m/s increments, totalling six simulations for each mesh generated. 12.5m/s was set as the lower limit for the investigation as in both scenarios wind speeds at one increment lower (10m/s) where found to not equate to wind speeds greater than or equal to 12m/s at 20m above sea level (see Appendix B & C), and so would not constitute downtime for a marine services vessel.

Emphasis was placed on the location to the rear of the vessel as this is targeted by DARE as the most suitable location given that there are generally less obstacles disturbing the flow. This can be seen in Fig. 1 with a side on view of the swept areas for the turbines highlighted by the two black lines with hub heights of 10m above the deck as suggested by DARE. The central location for the portable marine wind turbine is positioned very close to the highest point of the vessel at just over 7.5m above the deck. This is expected to disturb the flow over the swept area of the middle position, particularly in the scenario with the wind direction coming head-on to the vessel.

The positions of the wind turbine swept areas relative to the vessel shown in Fig. 1 were kept throughout the investigation for both scenarios.

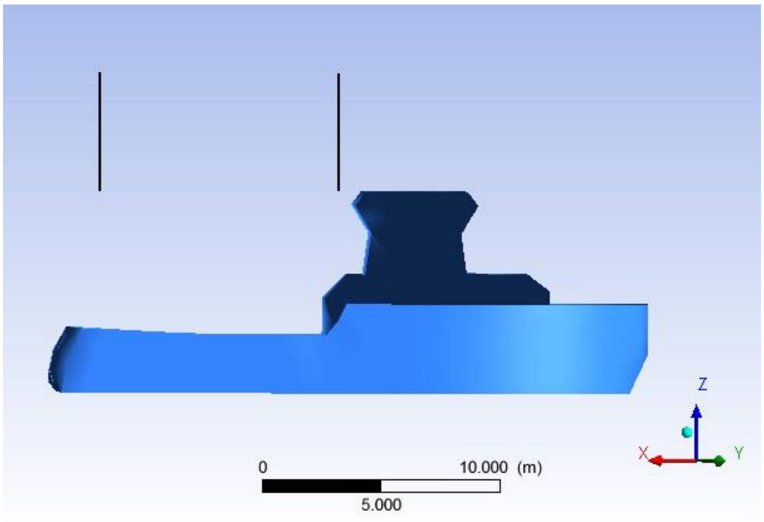

Fig.1

This investigation concentrated on setting wind speeds at the hub height of the turbine situated 10m above the deck at the rear of the vessel for analysis.

A mesh sensitivity analysis will be presented in the full paper along with a detailed assessment of the effect of the turbine mounting on the available wind resource. Example figures are given below.

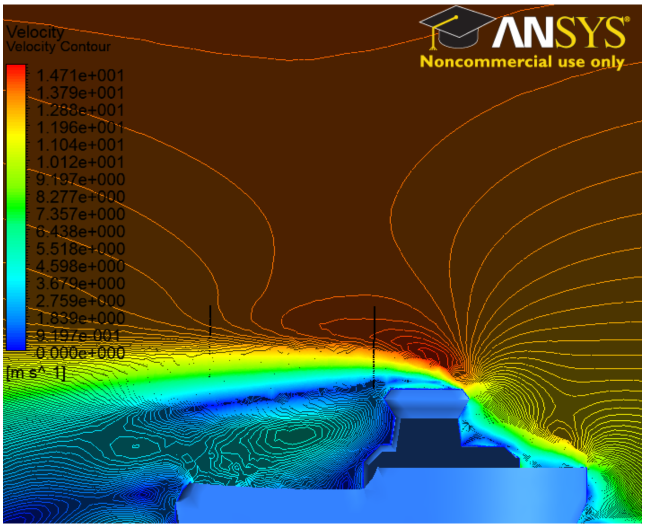

Fig. 2. Velocity contour plot.

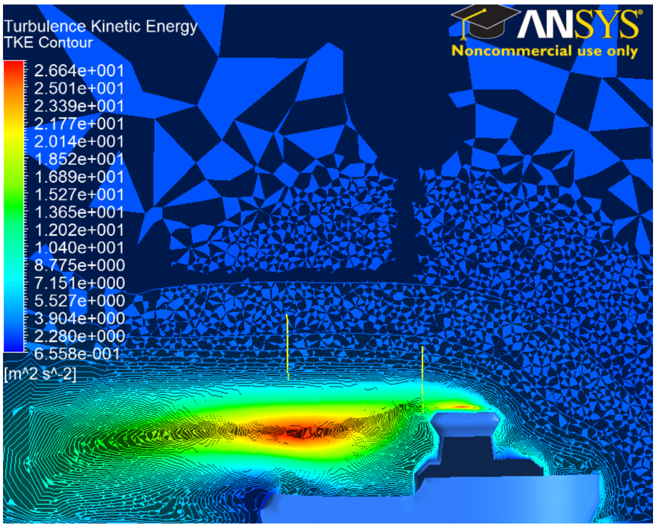

Fig. 3. Turbulent Kinetic Energy contour plot.

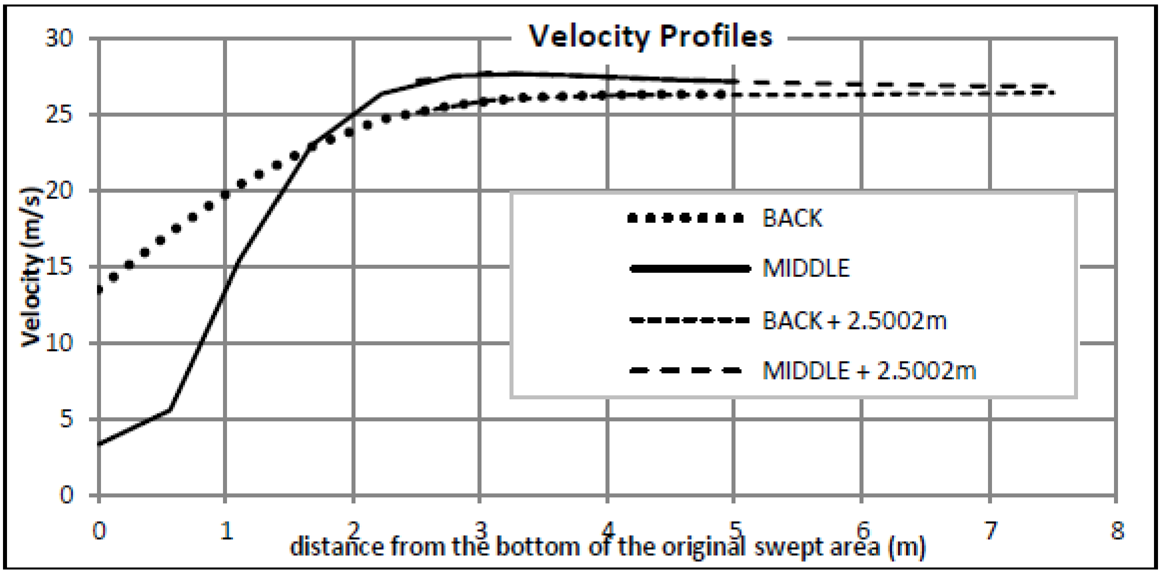

Fig. 4. Velocity profiles for each of the four locations considered with the velocity set at 25m/s at the hub height of the rear turbine.

CONCLUSION

The location for a portable marine wind turbine on a marine services vessel needs careful consideration. In this investigation it was found that locating the turbine too close to the vessel results in turbulent air flow and large velocity ranges over the swept area.

In both scenarios investigated it was found that the middle location for the turbine is unsuitable due to its proximity to the vessel's highest point causing turbulent flow to pass over the swept area. Raising the middle position to be 2.5 m above the highest point of the vessel raises the swept area into much steadier air flows with lower turbulence.

In scenario 1, the rear location for the turbine is located within the turbulent flow caused by the presence of the vessel. Raising the location 2.5m above the highest point of the vessel raises the turbine out of the turbulent flow into much steadier flows. However in scenario 2, there is no great advantage to moving the turbine location higher above the vessel in terms of finding significantly steadier flow.

The results from this analysis are representative of the flow experienced over the vessel but the simplification of the vessel geometry and flow conditions reduced the accuracy in the results obtained. Further and more comprehensive analysis is required to obtain realistic results that could be validated with data obtained from test conditions. Simulations more in line with actual conditions would include modelling unsteady flow conditions with wave motion included.

Also, the geometry of the vessel could include more of the fixtures and fittings present on an actual vessel to more accurately simulate the flow over such obstacles.

LEARNING OBJECTIVES

The learning objectives of this paper are to show conference attendees how CFD can be used to optimise deployment of a real world technology and also to introduce an alternative wind energy application.

REFERENCES

[1] J. Skea, “The renaissance of energy innovation,” *Energy Environ. Sci.*, vol. 7, no. 1, p. 21, 2014.

[2] P. Grehan and D. Hayes, “DARE Technology market analysis.”

[3] European Parliament and Council of the European Union, “Directive 2008/50/EC of the European Parliament and of the Council of 21 May 2008 on ambient air quality and cleaner air for Europe,” *Off. J. Eur. Communities*, pp. 1–43, 2008.

[4] Equasis Statistics, “The world merchant fleet in 2013,” 2014.

[5] G. J. W. Van Bussel and W. a a M. Bierbooms, “Analysis of different means of transport in the operation and maintenance strategy for the reference DOWEC offshore wind farm,” 2003.

[6] H. K. V. and W. Malalasekera, *An Introduction to Computational Fluid Dynamics*, vol. M. 2007.

[7] T. Strategy, D. Using, F. Method, T. F. Method, and B. Conditions, “1. Introduction to CFD.”

[8] C. Desmond, “The consideration of forestry effects in wind energy resource assessment.”

[9] W. Zuo, “Introduction of Computational Fluid Dynamics.”

[10] J. Franke, C. Hirsch, and A. Jensen, “Recommendations on the use of CFD in wind engineering,” *Cost Action C*, no. August 2015, pp. 1–11, 2004.

[11] P. Flows and U. Flows, “Solution methods for the Incompressible Navier-Stokes Equations.”

[12] L. Chen, “Navier-stokes equations for fluid dynamics,” pp. 1–10, 2014.

[13] B. E. Launder and D. B. Spalding, “Lectures in mathematical models of turbulence,” 1972.

[14] X. Zhang, *CFD simulation of neutral ABL flows*, vol. 1688, no. April. 2009.

[15] P. . Richards and R. . Hoxey, “Appropriate boundary conditions for computational wind engineering models using the k-ϵ turbulence model,” *J. Wind Eng. Ind. Aerodyn.*, vol. 46–47, pp. 145–153, Aug. 1993.

[16] O. Førde, “Analysis of the Turbulent Energy Dissipation,” no. June, 2012.

[17] J. Wu, “Wind stress and surface roughness at air-sea interface,” *Geophys. Res.*, vol. 74, no. 2, pp. 444–445, 1969.