PO.150

CFD analysis of a 5kW portable Marine Wind Turbine

Dr. Cian Desmond, Jason McCorkell, Dr. Paul Leahy, Rachel Chester
University College Cork, MaREI

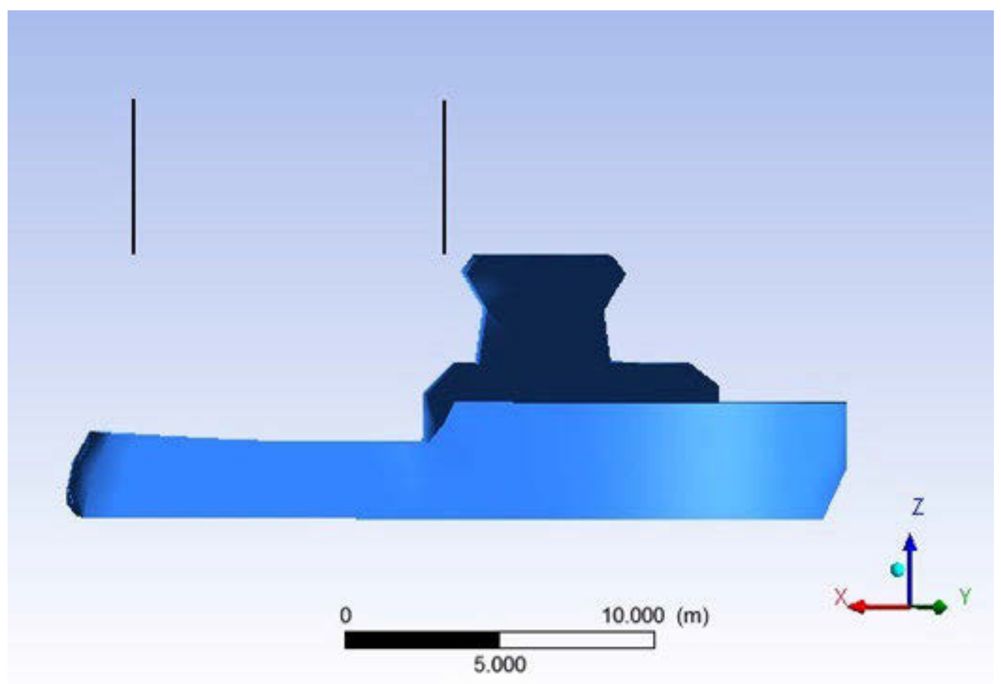
Coláiste na hOllscoile Corcaigh, Éire
University College Cork, Ireland

Abstract

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.

Method

- **ANSYS CFX** software package was used to simulate various offshore wind conditions using the **RANS** turbulence closure[1]
- Considered a 5 kW wind turbine with a **5m rotor diameter** Swept areas of the turbine are illustrated by the black lines above the vessel geometry
- To correlate with vessel downtime, simulations were made for regular wind speeds **12.5 m/s to 25m/s** using 2.5m/s increments.

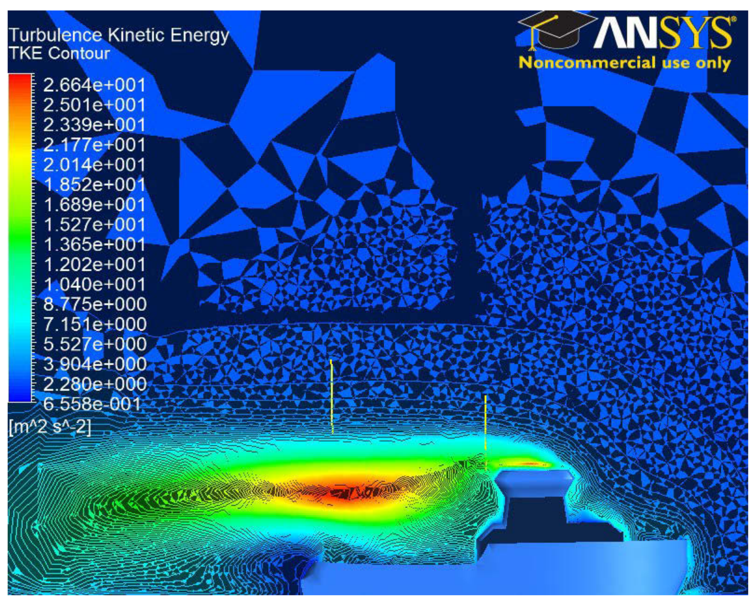


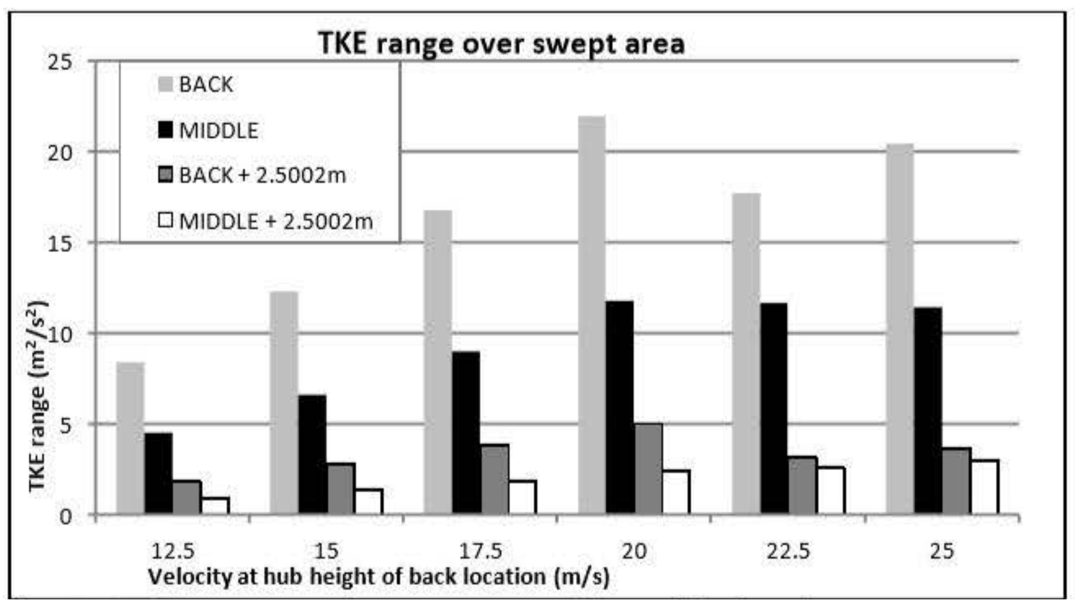
- **Two vessel orientations** considered:
  - Scenario 1: Front of vessel facing into the wind
  - Scenario 2: Rear of vessel facing into the wind
- **Four turbine positions** considered:
  1. Rear of the vessel with hub height 10m above deck
  2. Middle of the vessel with hub height 10m above deck
  3. Rear of the vessel with hub height 12.5m above deck
  4. Middle of vessel with hub height 12.5m above deck

Results

The position of the turbine is critical in order to **optimize generation and avoid damage** to the turbine and/or vessel. A turbine located within an area of turbulent flow, caused by the presence of the vessel, increases dynamic fatigue loading on the turbine and the structure on which it is held. **To improve survivability and maximize performance**, the turbine needs to be in a position where **turbulent air flow is minimized**.

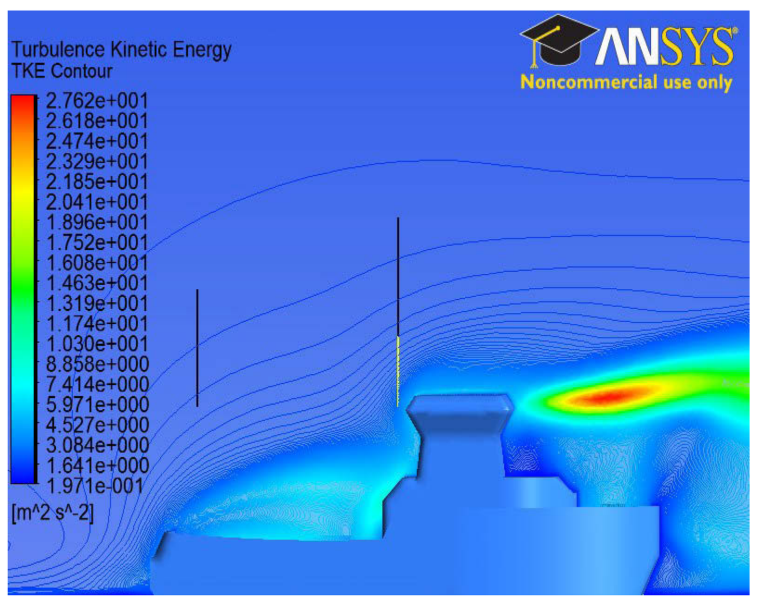
Scenario One – Front of vessel facing into the wind

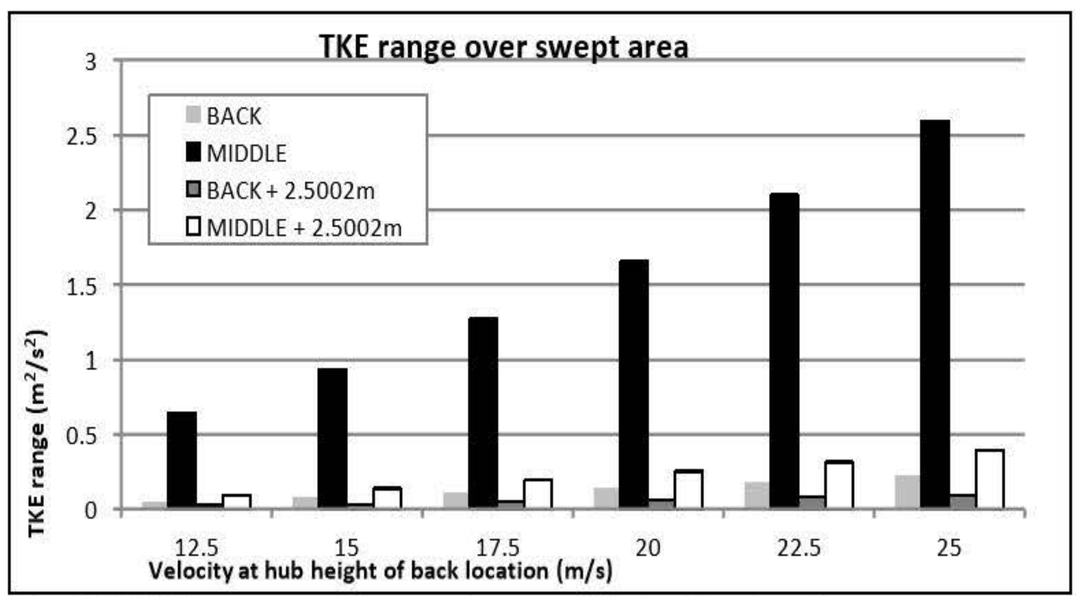




The rear and middle locations of the 10m turbine are located within the turbulent flow caused by the presence of the vessel. Raising the hub height to 12.5m, 2.5m above the highest point on the vessel, places the swept area within less turbulent flows.

Scenario Two – Rear of vessel facing into the wind





At the rear of the vessel, the hub height has little significant effect on the state of wind flow experienced by the turbine.

However, the middle location remains in an area of turbulent flow caused by the presence of the vessel. Similarly to scenario one, raising the height of the turbine to 12.5m can reduce the turbulence experienced and places the turbine in an area of increasingly consistent wind flow.

Conclusions

In both scenarios investigated it was found that the **middle location for the 10m high turbine is unsuitable** due to its proximity to the vessel's highest point causing turbulent flow to pass over the swept area. Similarly, the 10m high turbine located at the rear position is also within turbulent flow during scenario one. Raising the position to be 2.5m above the highest point of the vessel **raises the swept area into much steadier air flows** with lower turbulence. However, in **scenario two there is no great advantage in raising the height** of the turbine 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.

References

[1] C. Desmond, "The consideration of forestry effects in wind energy resource assessment."