Applied Thermal Engineering 135 (2018) 530–536

Contents lists available at ScienceDirect

Applied Thermal Engineering

journal homepage: www.elsevier.com/locate/apthermeng

Research Paper

Computational fluid dynamics analysis and field measurements on ice accretion on a cup anemometer support arm

Marie Cecilie Pedersen[a,b,*], Henrik Sørensen[b], Nigel Swytink-Binnema[c], Thomas Condra[b]

[a] Vattenfall Vindkraft A/S, Jupitervej 6, 6000 Kolding, Denmark
[b] Aalborg University, Department of Energy Technology, Pontoppidanstræde 111, 9220 Aalborg, Denmark
[c] TechnoCentre éolien, 70 rue Bolduc, Gaspé, Québec G4X 1G2, Canada

HIGHLIGHTS

- A CFD based icing model was developed in ANSYS Fluent.
- Ice was modelled over 1 h using experimental data as input conditions.
- The modelled ice thickness showed great agreement with the experimental data.
- The modelled ice was successfully included in the existing geometry by an iterative process in between every time step.

ARTICLE INFO

Keywords:
Ice model
Computational fluid dynamics
Ice accretion
Dynamic meshing

ABSTRACT

Ice growth on structures is a problem in cold climate regions. A method to model ice accretion on the cross section of a cup-anemometer support arm is presented in this study. The model was developed in ANSYS Fluent by implementing existing icing theory and by developing the dynamic meshing package to match ice accretion through user defined functions (UDFs). The Euler-Euler multiphase model was used to model in-cloud icing conditions and an impingement model was implemented to extract the ice deposit per time step. A surface boundary displacement model was implemented to determine the new surface contour after ice deposit and the surface boundary is displaced by an iterative process between each time-step. Icing was simulated over time by using measurements of the atmospheric conditions from a cold climate site in Canada. The numerical results were validated using experimental data and compare well with the experiments, when simulating 1 h of icing.

1. Introduction

Structures located in cold climate regions with high air densities, low temperatures and a high relative humidity are exposed to icing during the winter time. Such structures can be overhead power lines and conductors [20,15], meteorological masts (met masts), other tall structures and wind turbines [10,16]. Iced power lines can lead to power failure and the power line and power pole can collapse due to the additional ice load. Ice shedding from structures and wind turbines is a danger to accidental passers-by, service personnel or nearby buildings. The performance of wind turbines located in cold climates suffer from a loss of production and blade fatigue due to additional load on the blades. At some wind farms, the production losses due to icing are so high, that the total annual profit of the wind farm is threatened. The phenomenon leading to ice accretion is known as atmospheric icing. It covers the processes where water droplets- or drizzle, wet snow or rain will freeze and stick to an object and turn into ice. One normally makes a distinction between precipitation icing and in-cloud icing [14]. Precipitation icing covers ice accretion caused by wet snow, drizzle and freezing rain, whereas in-cloud occurs when super-cooled droplets in clouds or fog freeze upon contact with an object resulting in ice growth. With the expansion of wind power in cold climate[1] from around 500 MW in 2002 [16] to a cumulative capacity of 127 GW by the end of 2015 [9], knowledge and experience with operation and models have been requested by the related industry and academia to determine the potential icing risks of a site [5,16]. Simulating icing in a climatic wind tunnel is an expensive affair and does not provide definitive conclusions for wind power applications. Whilst wind power sites, are most often equipped with measurements equipment it is evident, and more cost-effective, to combine on-site measurements with numerical models for

* Corresponding author at: Aalborg University, Department of Energy Technology, Pontoppidanstræde 111, 9220 Aalborg, Denmark.
E-mail address: mcpe@et.aau.dk (M.C. Pedersen).

[1] Cold climate refers to sites, which experience severe periods of icing events and/or temperatures below the operation limits of a standard wind turbine [5]

https://doi.org/10.1016/j.applthermaleng.2018.01.086
Received 5 November 2017; Received in revised form 23 January 2018; Accepted 24 January 2018
1359-4311/

simulation of icing. Since a wind turbine is difficult to access during icing conditions, wind turbines are typically observed by a combination of measurement equipment installed on met masts and on the ground. It is not trivial to measure atmospheric icing, since the instruments are also influenced by the harsh conditions and can be affected by ice themselves [33]. However, studies have shown that combining measurements of the atmospheric parameters such as wind speed, temperature, pressure and relative humidity with observations by web cameras can provide a good picture of the atmospheric icing at a site and thereby the potential icing risks [32,16]. Observations from web cameras can be analysed using image analysis to provide additional icing information, such the ice thickness and the ice distribution, which are usually not available [1]. Thus, measurement equipment on met mast might just as useful for evaluating the ice risks at a site. And in the planning phase of the wind farm, met mast are often the only installations at the site.

A detailed icing model of the wind turbine blade provides information about the aerodynamic degradation of the blade during ice accretion. Detailed icing studies are seen in the aeronautics, which have primarily been developed for flight safety reasons [27]. Messinger (1953) [21] described the energy balance during ice accretion for an unheated surface, which has formed the basis of most icing models used today. An example is Makkonen's empirical model [19] and another example is the advanced icing models seen in the aircraft industry. Commercial ice models have been developed for aircraft icing [35,13,6], but those models are not likely to be accessible for wind power researchers and the model applications are tuned for in-flight conditions and not for icing near the ground. The aim of this study is to develop a framework for modelling icing using computational fluid dynamics (CFD), having icing on instruments as a starting point. The study is carried out based on the following objectives:

- The icing model will be developed in a commercial CFD software by; implementing existing icing theory and by modifying existing dynamic meshing package to match icing requirements, both by user defined functions (UDFs).
- The modelled ice thickness is compared to observed ice thicknesses obtained by image analysis.

More specifically, the intention is to develop a model, which runs in software ANSYS Fluent using built-in packages and user defined functions (UDFs). In this way, the user can easily modify and control the model and no other software is needed, thereby decreasing the computational time required. Other studies showed promising initial results using ANSYS Fluent [34,30]. And recently, for predicting ice accretion in relation to anti-icing, ANSYS Fluent was used to simulated in-cloud conditions, where the ice contour and generation of the new mesh were performed using MATLAB [31]. In this study, it is not the intention to compete with other highly specialised icing models from, for example, the aircraft industry. Rather it is to develop an easily accessible model for modelling ice accretion on structures near the ground, which runs as one unified model in ANSYS Fluent. In future studies the model framework can be extended for wind turbine blades, which will be interesting in terms of evaluating de-icing systems.

2. Materials and methods

2.1. On-site measurements of ice growth

Icing was simulated using measurements provided by TechnoCentre éolien from the Site Nordique Expérimental en Éolien Corus, Riviére-au-Renard, Quebec, Canada [32]. The measurements used were collected from a 126-m-high met mast, which has equipment installed at 16 heights including ground level. Icing was recorded over time at 103 m on a cup anemometer support arm with a diameter (d) of 0.0267 m, see Fig. 1. Ice thickness L was extracted using image processing [25], as seen in Fig. 2 (left). The camera was originally installed with the purpose of monitoring the mast equipment, but the pictures turned out to be of great value for observing ice accretion. The wind direction was shown to the perpendicular to the camera's axis, and the distance to the camera was fixed, which enabled image processing to derive the ice thickness [25].

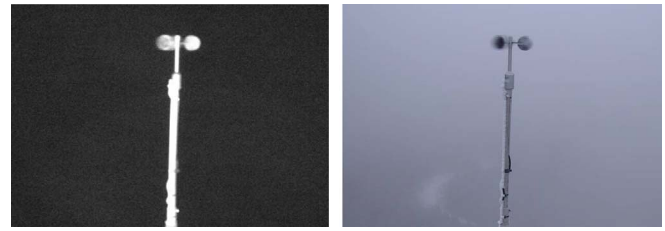

Fig. 1. The cup anemometer and support arm on the met mast in a Canadian wind farm. To the left at nighttime with no ice accretion and to the right at daytime during in-cloud conditions.

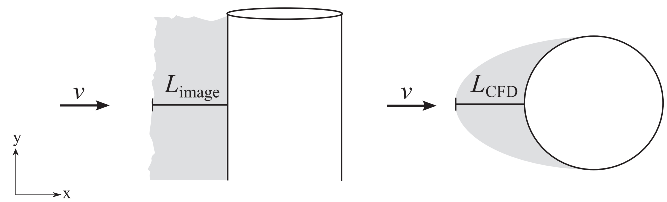

Fig. 2. Ice thickness L obtained from image analysis to the left and ice thickness L_{CFD} calculated from CFD icing model to the right. The wind free-stream velocity is given as v_∞.

2.2. Computational domain

The computational domain consists of a C-grid, with a structured O-grid surrounding the cylinder representing the cup anemometer support arm, and an unstructured grid far downstream, see Fig. 3. A structured O-grid was chosen due to the dynamic mesh update approach. The cylinder was discretized by 160 cells having the highest density around the stagnation point, marked by the black square in Fig. 3. The flow Reynolds number of the simulations was $2.77 \cdot 10^4$. Since ice accretion happens at the leading edge of the cylinder, a relatively low cell density, downstream and far away from the cylinder surface, was allowed. The domain consists of a total of 5454 mixed cells with dimensions of 36d upstream of the cylinder and 48d downstream of the cylinder. For the standard mesh design the first cell height was $1.33 \cdot 10^{-4}$ m. Since the simulations covers one hour of icing and future simulations are intended to cover a full icing event of several hours, such simulations are not suited to be dissolved with a very small timestep nor a very fine spacious dissolution. The simulation set-up was defined based on a

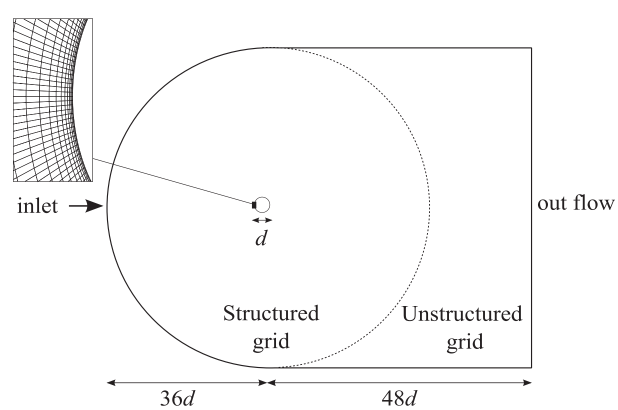

Fig. 3. Computational domain, as a C-grid with flow inlet and outlet from left to right. The black square on the front of cylinder illustrates the area of the zoom.