Numerical Study of Aerodynamic Characteristics of a Symmetric NACA Section with Simulated Ice Shapes

N. Tabatabaei, M.J. Cervantes, C. Trivedi, Jan-Olof Aidanpää

1 Fluid and Experimental Mechanics, Luleå University of Technology, Sweden
2 Norwegian University of Science and Technology, Norway

Abstract. To develop a numerical model of icing on wind turbine blades, a CFD simulation was conducted to investigate the effect of critical ice accretions on the aerodynamic characteristics of a 0.610 m chord NACA 0011 airfoil section. Aerodynamic performance coefficients and pressure profile were calculated and compared with the available measurements for a chord Reynolds number of 1.83×10^6. Ice shapes were simulated with flat plates (spoiler-ice) extending along the span of the wing. Lift, drag, and pressure coefficients were calculated in zero angle of attack through the steady state and transient simulations. Different approaches of numerical studies have been applied to investigate the icing conditions on the blades. The simulated separated flow over the sharp spoilers is challenging and can be seen as a worst test case for validation. It allows determining a reliable strategy to simulate real ice shapes [1] for which the detailed validation cannot easily be provided.

1. Introduction

Ice accretion on aerodynamic surfaces can lead to significant deterioration in blade performance and operation. Based on the detailed aerodynamic measurements taken on iced airfoils and wings since 1978, four types of ice shapes have been detected: (1) roughness, (2) horn ice, (3) streamwise ice, and (4) spanwise-ridge ice [2]. Ice accretion is a phenomenon where super-cooled water droplets impinge and accrete on a body. In another assortment, it has two types of ice shapes. The first is called ‘rime ice’ which is generated at very low temperature (less than –10°C). In rime ice conditions, the droplets in the air freeze instantly at the impingement point. The second is called ‘graze ice’ which is generated in the temperature range 0 to –10°C. On graze ice conditions, droplets gradually freeze while moving along the body (so-called runback). When an ice layer is formed on an airfoil, it affects the performance by increasing drag and reducing lift, and it may cause a serious accident [3].

The atmospheric ice accretion on the wind turbine blades can be simulated with the NASA panel code LEWICE [4]. It makes it possible to analyse the role of different atmospheric and system parameters on the predicted ice profile geometry and ice mass distribution [5]. In [6], for a wind speed and icing conditions, the LEWICE ice shapes were generated as the glaze ice accretions after 22.5 and 45 min. Glaze ice accretion is often characterized by the presence of large protuberances, commonly known as glaze horns, which can cause flow separation downstream of the horns [6]. On iced airfoils, the boundary layer separates near the top of the horn, due to the pressure gradient produced by the large discontinuity in the surface geometry. As the specification of the horn is determinative in the
subsequent vortices structure, it was desired to parametrize these horns. M. Papadakis et al. in 1999 [7] simulated glaze ice shapes by means of spoilers attached on the upper surface of an airfoil near the leading edge. The spoilers modeled in this work were sized to simulate 22.5 minutes glaze ice accretions for a 0.610-meter chord NACA 0011 airfoil. The 22.5 minutes ice accretion are often used in aircraft icing analysis and certification to evaluate aerodynamic performance degradation of critical aerodynamic surfaces in the event of an ice protection system failure. The simulated ice shapes were designed to rotate about a hinge axis so that variations in horn angle could be obtained with ease. In 2000, experimental data for the 0.610 meter and 0.305 meter NACA 0011 airfoil with simulated upper and lower glaze ice shapes were provided [6]. In 2004 [8], flow field measurements were carried out on the upper surface of a GLC-305 airfoil configured with glaze and rime ice-shape simulations. They confirmed the presence of the large separation bubbles downstream of the ice through velocity measurements.

The unsteadiness of the flow field about an iced airfoil frequently renders the prediction of the maximum lift coefficient by numerical means inaccurate. Based on the investigations provided in [9], the time-dependent pressure distributions indicated the movement of increased suction pressure over the iced airfoil surface. It may represent the motion of vortices in the separated shear layer near reattachment.

In 2012, F.Villalpando et al. [10] conducted a numerical simulation over a two-dimensional and ice-accreted NACA 63-415 airfoil at various angles of attack. They validated the calculated load coefficients with the experimental data and then calculated the modification of pressure distribution due to the ice accumulation numerically.

Power loss estimations for three different rime ice cases were determined for a 3 MW wind turbine in 2013 [11]. With an ice accretion simulation, the aerodynamic properties of the iced profiles were modelled using computational fluid dynamics (CFD). The power curves were generated for the clean wind turbine and for the same turbine with the different ice accretions. No validation with experimental data was available.

The work presented in this paper is a part of an ongoing research to determine aerodynamic performance of wind turbines in icing condition. One of the goals of this work is to validate the numerical method through an available experimental database on the effects of “critical ice accretions” on aircraft performance.

The following sections give a description of numerical methodology, modeled geometries and boundary conditions.

2. Approach and Methods

The geometry (Figure 1) and boundary conditions are from the experimental setup used in [7], [6]. The wind tunnel is a single-return closed circuit facility from which a section is chosen to be modeled. The test section is 7 feet high by 10 feet wide by 12 feet long.

![Figure 1. Simulated spoiler-ice shapes for NACA 0011 [3]](image-url)
To decrease the computational requirements, two-dimensional simulations are performed. In spite of the fact that turbulent structures are usually not symmetric and two-dimensional simulations are not completely feasible, a two-dimensional simulation, which is numerically affordable, is performed to see the main features of the flow.

Grid topology of C-type was used to generate high quality structured hexahedral grid by ICEM CFD15. The value of $y^+$ corresponding to the first grid point above the wall was below 1. Numerical simulations are performed using ANSYS CFX 15.0 solver [11]. Both steady state and transient simulations were conducted. High-resolution advection scheme was selected for the spatial discretization and ‘second order backward Euler’ scheme was applied for the temporal discretization of RANS equations. Shear stress transport model with automatic wall function was activated to model the turbulent flow. This model performs well to capture vortices in the separated flow. The convergence criterion was set to a root-mean-squared value less than $10^{-6}$. The simulations were performed for at least 4 cycles of lift variations, corresponding to 93,000 iterations or more in the case of $3.5 \times 10^{-5}$ s time step. The pressure variation were quasi periodic after 4 cycles. The numerical results are compared with the available experimental data of the test tunnel, including the pressure distribution over the blade surfaces, as the representatives of the airfoil performance in zero angles of attack and the selected spoiler install angles. The mean value of loads in the final steady situation is calculated as well as the accumulated value of the periodic acting loads in the transient process. The accumulated values are calculated assuming the convergence to periodic flow.

3. Clean airfoil

To have a preliminary evaluation of the simulation and effective parameters, a clean airfoil was firstly modelled. Although the dominant phenomenon in icing condition is separated flow, still modelling a smooth flow over a clean airfoil provides the opportunity to tune the boundary conditions and analyse the parameters of a numerical method.

3.1. Mesh analysis

Mesh refinement is performed step by step to approach a grid independent simulation by decreasing the uncertainty in the mesh. The discretization error by GCI method [12], which is based on Richardson extrapolation, is calculated in each case for transient simulations as well as the steady state simulations. Pressure coefficients $C_p$ in 3 different points (on the airfoil body) were chosen as critical parameters. Three grids were used for steady state simulations that contained 0.085 M, 0.67 M, and 2.3 M hexahedron elements. Once the mesh analysis was performed for the steady state simulation, it was performed for the transient simulation similarly with 0.67 M, 2.3 M, and 5.4 M hexahedron elements grids. The mesh discretization error is less than one percent in all investigated cases (Table 1).

| Simulation Type | Steady state | Transient |
|-----------------|--------------|-----------|
| Criteria Parameter | $C_p$ at $x/c=0.84$ (US) | $C_p$ at $x/c=0.77$ (LS) | $C_p$ at $x/c=0.84$ (US) | $C_p$ at $x/c=0.77$ (LS) |
| Numerical uncertainty in the fine-grid solution (%) | 0.2 | 0.00003 | 0.17 | 0.066 |

3.2. Resolving boundary layer

In mesh analysis, the first node size was fixed, and only the density of the mesh was studied. In this part, the effect of mesh refinement on the walls is discussed. Although the flow is smooth and there is no separation in zero angle of attack, still boundary layer of the airfoil body affects the pressure distributions. This distinction was seen comparing two simulations with $y^+=80$ and $y^+=0.8$. So, all the remaining simulations were performed with resolving the boundary layer over the airfoil surfaces with $y^+<1$. 

3
A similar analysis was implemented about the bottom and top walls boundary layers of the 2D modelled test case. Since the flow is attached to the airfoil everywhere, the effects of surrounding walls cannot be transmitted to the side walls of the tunnel; which was confirmed in the simulations. As expected from the stable character of an attached flow, no unsteadiness is seen in neither of 2-D and 3-D simulations; however, once the boundary layers of the surrounding walls are resolved, some slight effects of unsteadiness arise which actually don’t make significant changes in the whole pressure distributions.

3.3. Clean airfoil Results
The pressure coefficient distribution is shown for a clean airfoil in Figure 2; ‘x/c’ is the dimensionless axial distance from the leading edge. Pressure coefficient is defined as the pressure value divided by the dynamic pressure in the free stream fluid. Since the zero angle of attack is considered, the pressure distribution is almost the same on upper and lower surfaces. Regarding the agreement between the simulated and measured pressure values, lift and drag forces were in good agreement too (Table 2).

| Table 2. Load acting on the clean airfoil calculated from a steady state simulation |
|----------------------------------|----------------|
| Lift Coefficient | Drag Coefficient |
| experiments | 0.005 | 0.01          |
| Simulation     | 0.002 | 0.01          |

Figure 2. Pressure distribution on the upper and lower surfaces of a clean airfoil

4. Iced airfoil
The computational domain consists of the same components of the test model including the spoilers both on the upper and lower surfaces of the blade. The spoiler angle is set to 40° and 0° for upper and lower ones, respectively. With symmetry boundary assumption in the span direction, two spoilers are defined as no-slip walls as well as the top and bottom walls of the wind tunnel. Flow is assumed to have a uniform velocity at inlet and is allowed to move backward at exit as the opening boundary is set.

4.1. Mesh analysis
A multi-blocking mesh consisting of hexahedral elements is generated. The mesh is dense on the spoilers and the airfoil surface in order to resolve the boundary layer formed on these parts. More studies are accomplished to analyse the effects of resolving boundary layers on the surrounding walls, spoilers and the airfoil. The mesh density effects are also studied to achieve a mesh independent solution.

A preliminary mesh containing of 0.2 M hexahedral elements was generated and refined around the airfoil and spoilers to maintain y’<1. It is seen that both sides of each spoiler need a different level of refinements. As the dimensionless distance, y’ depends on the wall shear stress as well as the size of
the first cell $y$. The wall shear stress value is found different at each side of the spoilers as they are the upstream and downstream of the separated flow. Resolving the boundary layer on the airfoil and spoilers, the mesh resolution sensitivity is studied. Load coefficients have been considered as the key parameters (Table 3). Three grids were used consisting of 1.18, 1.47, and 3.7 million elements.

Table 3. Discretization error for iced airfoil in a steady state simulation

| Simulation Type     | Steady state (2D)    |
|---------------------|----------------------|
| Criteria Parameter  | Lift Coefficient     |
| Numerical uncertainty in the fine-grid solution (%) | 0.076                |
|                     | Drag Coefficient     |
|                     | 0.054                |

Then it is seen that the error due to the mesh size is negligible in the performed steady state simulation. It should be noted that the steady state convergence cannot be reached if a coarser mesh is used.

A similar study for transient simulation was performed with three grids of 0.94, 1.18 and 3.7 million elements. Regarding the method suggested in [12], the numerical uncertainty due to the mesh increases in the case of transient simulation (Table 4). It means that a denser mesh is required for a time-dependent simulation; while more refinement is not affordable in this study due to the computational costs.

Table 4. Discretization error for iced airfoil in a transient simulation

| Simulation Type     | Transient (2D)     |
|---------------------|-------------------|
| Criteria Parameter  | Lift Coefficient  |
| Numerical uncertainty in the fine-grid solution (%) | 8.15               |
|                     | Drag Coefficient  |
|                     | 3.4               |

4.1.1. Side walls boundary layer effects. In the mentioned mesh study, the average value of $y^+$ for the test section walls above and below the blade was around 600. To see the effect of the boundary layer at the walls, the mesh was refined on that walls up to $y^+<1$. No change was observed in the results while enough number of nodes were provided inside the domain to keep a satisfactory resolution of the mesh downstream. It means that the wind tunnel walls have been designed properly not to have any considerable effect on the flow over the blade, or the effect may not be captured through a steady state simulation.

4.2. Steady state results
Considering the possibility of flow separation due to the sharp spoilers, the flow is expected to be highly transient. The vortices location and shape are expected to strongly vary with time. On the other hand, it is known that in a steady state simulation for all the properties of the system, the partial derivative with respect to time is assumed to be zero. In the case of the dominant transient character of the flow, a steady state simulation can be representative of a mean situation of the flow. In a steady-state simulation, the physical system is assumed to move towards an equilibrium or steady-state solution [13]. From a numerical point of view, the CFX-Solver applies a false time step as a mean of under-relaxing the equations as they iterate towards the final solution. Because the solver formulation is robust and fully implicit, a relatively large time scale can typically be selected, so that the convergence to steady-state is as fast as possible [14].

The challenge of a steady state simulation in this study is to damp the strong flow oscillations numerically, which are the physical transient effects of the flow, and obtain a mean behaviour of the flow through a steady state simulation. Numerical effort to damp the time-dependent effects is necessary. Convergence of this steady state simulation is highly dependent on a proper initial value. The results are shown in Figure 3 which can be addressed as the mean vortices. The vortices are confined on the airfoil surface, so no bubble is seen in the downstream region. Also, the sizes of the upper and lower vortices are similar in the steady situation which means the vertical acting forces are...
similar in both directions, i.e., a lift value around zero (excluding the forces on the spoilers themselves). More details can be seen in a closer snapshot in the right-hand side of Figure 3. The flow separates at the tip of both spoilers with the high velocity; while the low speed rotating flows fill the region downstream of the spoilers. Obviously, the axial flow which was supposed to act on the airfoil surfaces to generate the lift force doesn’t meet the airfoil surfaces at all since the airfoil is covered thoroughly by the vortices. With the existing horn height, the designed curvature of the NACA0011 does not create any lift, as the flow is following the path formed at the boundary of the vortices.

![Figure 3. Streamlines in a steady state flow](image)

Pressure distribution on both sides of the airfoil is plotted in Figure 4 besides the experimental data. The gap in between is expected as the transient effects have been skipped and only the mean value has been considered. Regarding the similar vortices on upper and lower surfaces in steady state, the pressure distributions on both surfaces are close together; while there is a distinct difference between the pressure values of upper and lower surfaces of the airfoil in the measured data.

![Figure 4. Pressure Coefficient distribution: steady state simulation and experimental results](image)

Regarding the acting loads, lift and drag are overestimated by 8.5% and 4.7% respectively. They are reported with the experimental values at zero angle of attack (Table 5).

### Table 5. Load acting on the iced airfoil calculated from a steady state simulation

|                | Steady state simulation | experiments |
|----------------|-------------------------|-------------|
| Lift Coefficient | 0.151                   | 0.138       |
| Drag Coefficient  | 0.231                   | 0.220       |
4.3. Transient effects
In addition to the turbulence modelling and the applied mesh properties, an appropriate time step size needs to be chosen in the case of transient simulation. Transient effects are considered after a time step size sensitivity analysis as well as a mesh analysis which was previously described.

4.3.1. Time step size sensitivity analysis. The time-dependent behavior for a transient simulation is specified through the ‘time step’ as a numerical time discretization. The time step option provides a way for the numerical tool to track the progress of real time during the simulation. To specify the appropriate time step size, the effect of its variations was studied on the resultant lift and drag force. Regarding the inlet wind velocity, the number of time steps that it takes for the flow to pass the test case length was considered as a parameter (assuming a uniform velocity and a straight path). With an initial time step of 1.4 e-4 s, 500-time steps are necessary to propagate through the test case length. It was refined finally to 3.5e-5 s. Comparing to the final selected case, as it is shown in Figure 5, using 500 time steps causes a 44.7% reduction in the lift force. The lift value reduces by 12.9% as the number of time steps is halved (1000-time steps). Increasing the number of time steps from 2000 to 4000, no significant changes were observed. So, the case of 2000 time steps was selected.

![Figure 5](image)

**Figure 5.** Accumulated value of lift and drag force in transient simulations with different timestep sizes

4.3.2. Transient results. Usually, flow field studies of separation bubbles are focused on the time-averaged characteristics, however, the bubble flow fields are known to have strong unsteady characteristics that also play a role in the aerodynamics [2]. In this paper, these unsteady features are discussed besides the time-averaged characteristics. The separated flow downstream the sharp spoilers contains large separation bubbles challenging to capture with CFD. For the chosen position of the spoilers, the vortex movement behind the airfoil cannot be captured through a steady state simulation; while a transient simulation reveals more details of the flow. The streamlines over the iced airfoil are shown in 4 different selected times (Figure 6).
Figure 6 illustrates the vortices formation and movement over the airfoil surface. Consequently, the pressure around the airfoil changes with time as the vortices are formed and convected. It causes the lift to oscillate from positive to negative repeatedly that can lead to a dynamic load. Moreover, there are some variations downstream of the airfoil due to the shape and location of the vortices. Higher velocity at the region of separation occurrence leads to more waves downstream (at 3.29003 s). As an example, for a wind farm in which the distance between the wind turbines is designed, considering the vortex wake flow downstream, icing effect should be taken into account as it makes time-varying instabilities downstream of the rotor.

It is observed in Figure 6 that the tunnel sidewalls, above and below the airfoil, are enough far from the airfoil since the effects of the vortices are faded around the airfoil and the flow near the walls are unaffected.

The accumulated mean values of the forces on the iced airfoil are calculated with the periodic variations over a period of time. Due to the lift force fluctuations in time, a large number of iterations for the convergence of the accumulated value is needed (Figure 5). Finally, the time-averaged value of the forces is presented in Table 6. There is a difference between the calculated value of the lift and the measured data. As the RMS value for the measurements is not available, it is not possible to come to a clear conclusion. In addition to the prescribed mesh uncertainty, in the case of transient simulations, the influence of three dimensional phenomena may be more effective.

| Table 6. Load acting on the iced airfoil calculated from different types of simulations |
|---------------------------------|------------------|-----------------|------------------|
| Lift Coefficient                | Steady state     | transient       | experiments      |
| 0.151                           | 0.084            | 0.138           |
| Drag Coefficient                | 0.231            | 0.231           | 0.220            |

4.4. The effect of geometry modelling

Although the previous studies [6] have shown that the height, location, and angle of the horns are the important features of ice shape which can affect airfoil aerodynamic performance, we investigated the effect of the spoilers thickness too. Two cases were considered including the real geometry of the test

---

1 Refer to (9) for more information.
model and the case in which the thickness of the spoilers is assumed zero. The thickness was 1.3 mm for each spoiler. The transient operation was simulated.

The pressure distributions for both cases are plotted in Figure 7. No difference is seen between the two cases. It should be noted that this distribution curve doesn’t include the spoiler surfaces, but it shows only the pressure on the surface of the airfoil, excluding the parts on which spoilers are placed. Since the flow is separated on top of the spoilers, the airfoil surface downstream the spoilers locations is covered thoroughly by vortices (Figure 3). In fact, the pressure distribution in this area is dictated by the separation zone behind the spoiler. Whereas it is not affected by the thickness of the upstream spoilers, it shows the thickness of the spoilers, doesn’t influence the formed vortices on the airfoil significantly.

**Figure 7.** The effect of the spoilers geometry on the pressure distribution on the airfoil surface (not including the parts spoilers are located)

Some differences appear in the total forces on the whole iced airfoil (Figure 8). Regarding the accumulated values of the forces, ignoring the spoiler thickness causes an 86% reduction in the drag force and 118% reduction in the lift value. Also, the direction of the lift force is different in the no-thickness one with a negative resultant lift.

**Figure 8.** The effect of the spoilers thickness on the acting forces on the iced airfoil

Since the pressure distribution on the main body of the airfoil (excluding the spoilers locations) is similar in both cases, it can be concluded that the difference is attributed to the spoilers. To investigate the portion of the spoilers in the total load on the iced airfoil, Figure 9 is plotted. It shows the loads on the airfoil parts excluding the spoilers surfaces; as well as the total load on the iced
The accumulated value out of a transient simulation is plotted for both thick and flat spoilers. As an example, it is seen that 79% of the total drag force is on the spoilers for the thick case and 78% for the lift force. It is similar for a flat spoiler case. Therefore, details of the ice shape may significantly affect the lift and drag.

\begin{figure}[h]
\centering
\includegraphics[width=0.8\textwidth]{force_accumulated_value.png}
\caption{The spoilers portion in the acting forces on the iced airfoil}
\end{figure}

5. Conclusions
In order to develop a numerical strategy to simulate the icing condition of a wind turbine, this study is considering some wind tunnel tests [6], [7]. They were conducted with upper and lower spoiler-ice shapes as an investigation of the effect of glaze ice features on the aerodynamic performance of an NACA 0011 section. The 1.5-in spoiler was sized to approximate the horn heights of 22.5 min glaze ice-accretions. The experimental results provided detailed information for numerical validation such as the pressure distribution over the blade body.

First, the clean airfoil was simulated to check the preliminary requirements of the numerical model. An agreement with the experimental data was achieved even through a steady state simulation and the transient properties don’t seem to be very effective on loads as there is no separation in the flow; while still resolving the boundary layer is necessary on the airfoil surfaces.

For an iced airfoil with spoilers installed on upper and lower surfaces, a steady state simulation was performed to capture the mean flow character skipping the time-dependent properties of the transient flow in the separated region. It was concluded that a simulated mean flow cannot be a reliable model to calculate loads in the case of iced airfoil; however, it can give an idea about the approximate shape of the vortices and the separation points; while the downstream wakes cannot be captured at all.

Then the transient simulation was conducted to see the vortices structure and resultant periodic flow. Regarding the different vortices situations at different times, it was seen that their distributions on upper and lower surfaces of the iced airfoil is changing. It causes the lift switches from positive to negative repeatedly that can lead finally to a dynamic load. Also, there is not consistent situation for vortex wake flow downstream which can be important in wind farm design at icing condition.

Calculated accumulated values of loads through a transient simulation, were underestimated compared to the experimental data. It can be due to the skipping of the three dimensional effects, or the shortage of the shear stress transport model which has been used as the turbulence modelling.

Mesh density and time step size were investigated, for both steady state and transient simulations of iced airfoil, to make sure of the independence of the solution from the numerical features. To capture the transient effects, a high resolution of the mesh is needed. As an example, the same mesh that provides a consistent steady state solution, imposes 8% discretization error in a transient simulation even with 2.5 times more refinement. So the transient simulation of iced airfoil is very expensive though it is necessary to estimate the acting loads, downstream wakes, etc.

Furthermore, it was concluded that the thickness of the leading edge glaze ice should be taken as an effective parameter of an ice shape. Although it doesn’t change the formed vortices on the airfoil downstream of the spoiler tip, it can affect the acting forces on the spoilers. On the hand, it was...
investigated that in an iced airfoil the main part of the total force is inserted on the spoilers themselves; therefore, details of the ice shape may significantly affect the lift and drag. Focusing on zero angle of attack, the pressure distribution followed the close trend to the experiments; Drag force accumulated value was close to the experimental data; while the approximation of a time-averaged value of the lift force needs more effort both in numerical and experimental study to clarify more details of the fluctuating properties of the separated flow on the airfoil downstream of the spoilers.

6. References
[1] A. P. Broeren, M.B. Bragg and H.E. Addy 2006 Flowfield measurements about an airfoil with leading-edge ice shapes  J Aircraft 43 1226.
[2] M.B. Bragg, A.P. Broeren and L.A. Blumenthal 2005 Iced-airfoil aerodynamics Prog Aerosp Sci 41 323-62
[3] T. Matsuura, M. Suzuki and M. Yamamoto 2012 Numerical simulation of ice accretion phenomena on rotor blade of axial blower J Therm Sci 21 322–26.
[4] W.B. Wright 1995 User Manual for the NASA Glenn Ice Accretion Code LEWICE 1.6 NASA Lewis Research Center
[5] M. Etemaddar 2013 Load and response analysis of wind turbines under atmospheric icing and controller system faults with emphasis on spar type floating wind turbine (Norwegian University of Science and Technology)
[6] M. Papadakis, S. Alansatan and S. Wong 2000 Aerodynamic characteristics of a symmetric NACA section with simulated ice shapes 38th Aerospace Sciences Meeting and Exhibit
[7] M. Papadakis, S. Alansatan and M. Seltmann 1999 Experimental Study of Simulated Ice Shapes On a NACA 0011 Airfoil 37th AIAA Aerospace Sciences Meeting and Exhibit (Reno,NV)
[8] A.P. Broeren, M.B. Bragg and H.E. Addy 2006 Flowfield measurements about an airfoil with leading-edge ice shapes J Aircraft 43 1226-34
[9] H.M. Gurbacki and M.B. Bragg 2002 Unsteady aerodynamic measurements on an iced airfoil 40th AIAA Aerospace Sciences Meeting & Exhibit, Aerospace Sciences Meetings
[10] V. Fernando, R. Marcelo and I. Adrian 2012 Numerical study of flow around iced wind turbine airfoil Eng Appl Comp Fluid 6 39-46.
[11] V. Turkia, S. Huttunen and T. Wallenius 2013 Method for estimating wind turbine production losses due to icing (Espoo:VTT Technical Research Centre of Finland)
[12] I.B. Celik, U. Ghia, P. Roache and C. Freitas 2008 Procedure for estimation and reporting of uncertainty due to discretization in CFD applications J. Fluids Eng 130
[13] 2013 ANSYS CFX-Solver Modelling Guide Release 15.0 (ANSYS, Inc.)
[14] 2013 ANSYS CFX-Solver Theory Guide Release 15.0 (ANSYS, Inc.)