Numerical Simulation of Flow past an Airfoil with Ice Accretion on Leading Edge

Boyu Wang
Washington University in St. Louis

Follow this and additional works at: https://openscholarship.wustl.edu/eng_etds

Part of the Aerodynamics and Fluid Mechanics Commons, and the Mechanical Engineering Commons

Recommended Citation
https://openscholarship.wustl.edu/eng_etds/573

This Thesis is brought to you for free and open access by the McKelvey School of Engineering at Washington University Open Scholarship. It has been accepted for inclusion in McKelvey School of Engineering Theses & Dissertations by an authorized administrator of Washington University Open Scholarship. For more information, please contact digital@wumail.wustl.edu.
Numerical Simulation of Flow past an Airfoil with Ice Accretion on Leading Edge

By

Boyu Wang

A thesis presented to the McKelvey School of Engineering of Washington University in St. Louis in partial fulfillment of the requirements for the degree of Master of Science

May 2021

St. Louis, Missouri
Table of Contents

Table of Contents.................................................................................................................................................... i

List of Tables ........................................................................................................................................................................ iii

List of Figures........................................................................................................................................................................ iv

Nomenclature........................................................................................................................................................................ vi

Acknowledgements .............................................................................................................................................................. vii

Abstract................................................................................................................................................................................... ix

Chapter 1: Introduction........................................................................................................................................................... 1

1.1: Background........................................................................................................................................................................ 1

1.2: Objectives and Technical Approach .............................................................................................................................. 3

Chapter 2: Physical Model and Mesh.................................................................................................................................... 6

2.1: Physical Model and Mesh of Clean Surface Airfoil ....................................................................................................... 6

2.2: Physical Model and Mesh of Airfoil with Ice .................................................................................................................. 8

Chapter 3: Governing Equations and Numerical Method .................................................................................................. 12

3.1: Governing Equations ....................................................................................................................................................... 12

3.2: Turbulence Models ......................................................................................................................................................... 14

3.2.1: SA Turbulence Model.................................................................................................................................................. 14

3.2.2: WA Turbulence Model.............................................................................................................................................. 15

3.3: ANSYS Fluent Numerical Setup.................................................................................................................................... 16

Chapter 4: Flow past NACA23012 Clean Airfoil .............................................................................................................. 18

4.1: Flow Conditions .......................................................................................................................................................... 18
4.2: Simulation Results and Validations ................................................................. 18
4.3: Pressure and Velocity Contours ................................................................. 20
  4.3.1: Pressure Contours at Various Angles of Attack ...................................... 20
  4.3.2: Velocity Contours at Various Angles of Attack ...................................... 22
  4.3.3: Velocity Vectors at Various Angles of Attack ........................................ 24
Chapter 5: Flow past NACA23012 Airfoil with Ice Accretion near the Leading Edge ... 27
  5.1: Flow Conditions ............................................................................................ 27
  5.2: Simulation Results and Validations ............................................................... 27
  5.3: Pressure and Velocity Contours ................................................................. 30
    5.3.1: Pressure Contours at Various Angles of Attack ..................................... 30
    5.3.2: Velocity Contours at Various Angles of Attack ..................................... 32
    5.3.3: Velocity Vectors at Various Angles of Attack ........................................ 34
    5.3.4: Zoomed-in View of Velocity Contours and Velocity Vectors at Leading Edge .... 35
    5.3.5: Zoomed-in View of Velocity Contours and Velocity Vectors at Trailing Edge .... 37
Chapter 6: Conclusions and Future Work .......................................................... 40
  6.1: Conclusions ................................................................................................. 40
  6.2: Future Work ............................................................................................... 42
References ............................................................................................................ 43
Vita ....................................................................................................................... 45
List of Tables

Table 1: Simulation results for clean surface airfoil case using SA model .........................18
Table 2: Simulation results for clean surface airfoil using WA 2017m model .....................19
Table 3: Simulation results for iced airfoil using the SA model........................................29
Table 4: Simulation results for iced airfoil using WA model............................................29
List of Figures

Figure 1: Examples of accumulation of ice on wing’s leading edge in flight .........................1
Figure 2: Qualitative description of relative aerodynamic effect of various ice shaped airfoils [10] .................................................................................................................................................4
Figure 3: EG1164 horn shape ice accretion on an airfoil[7] ....................................................4
Figure 4: Geometry of NACA23012 airfoil .........................................................................6
Figure 5: Structured mesh in the computational domain.......................................................7
Figure 6: Zoomed-in mesh near the airfoil............................................................................7
Figure 7: Mesh quality under determinant 2×2×2 criterion ................................................8
Figure 8: Geometry of NACA23012 airfoil with leading edge glazed with ice generated in AutoCAD .................................................................................................................................................9
Figure 9: Zoomed-in ice shape on the leading edge generated in Auto CAD ....................9
Figure 10: Mesh in the computational domain around airfoil with horn shape ice ..........10
Figure 11: Unstructured grid around the airfoil with ice shape near the leading edge ......11
Figure 12: Zoomed-in mesh near the horn ice shape near the leading edge ....................11
Figure 13: Comparison of computed lift coefficient with experiment data using SA and WA turbulence models .................................................................................................................................19
Figure 14: Comparison of computed drag coefficient with experiment data using SA and WA turbulence models .................................................................................................................................20
Figure 15: Pressure contours around the airfoil at different angles of attack ..................22
Figure 16: Velocity contours around the airfoil at various angles of attack .......................24
Figure 17: Velocity vectors around NACA23012 airfoil at $\alpha = 0^\circ$ .................................24
Figure 18: Velocity vectors around NACA23012 airfoil at $\alpha = 6^\circ$ .................................25
Figure 19: Velocity vectors around NACA23012 airfoil at $\alpha = 12^\circ$ ...............................25
Figure 20: Comparison of computed lift coefficient using SA and WA turbulence models with 3D casting experimental data .................................................................................................................................29
Figure 21: Comparison of computed drag coefficient using SA and WA turbulence models with 3D casting experimental data ..........................................................30
Figure 22: Pressure contours around the airfoil with ice accretion at different angles of attack ..32
Figure 23: Velocity contours around the airfoil with ice accretion at different angles of attack...33
Figure 24: Velocity vectors around the airfoil with ice accretion at different angles of attack.....35
Figure 25: Zoomed-in velocity contours and velocity vectors around the horn ice at $\alpha = 0^\circ$ ....36
Figure 26: Zoomed-in velocity contours and velocity vectors around the horn ice at $\alpha = 6^\circ$ ....36
Figure 27: Zoomed-in velocity contours and velocity vectors around the horn ice at $\alpha = 12^\circ$ .....37
Figure 28: Zoomed-in velocity contours and velocity vectors near the trailing edge at $\alpha = 0^\circ$ ....38
Figure 29: Zoomed-in velocity contours and velocity vectors near the trailing edge at $\alpha = 6^\circ$ ....38
Figure 30: Zoomed-in velocity contours and velocity vectors near the trailing edge at $\alpha = 8^\circ$ ....38
Figure 31: Zoomed-in velocity contour and velocity vectors around the trailing edge at $\alpha = 10^\circ$38
Figure 32: Zoomed-in velocity contours and velocity vectors near the trailing edge at $\alpha = 12^\circ$..39
Figure 33: Comparison of lift coefficient between clean surface airfoil and iced airfoil cases ....41
Figure 34: Comparison of drag coefficient between clean surface airfoil and iced airfoil cases..41
Nomenclature

\( \alpha = \) angle of attack (AOA)  
\( c = \) airfoil chord length  
\( CFD = \) Computational Fluid Dynamics  
\( Cl = \) lift coefficient  
\( Cd = \) drag coefficient  
\( Cp = \) pressure coefficient  
\( \rho = \) density of air  
\( \mu = \) viscosity of air  
\( Re = \) Reynolds number  
\( Ma = \) Mach number  
\( \Delta Cl = \) difference in Cl between simulation and experiment  
\( \Delta Cd = \) difference in Cd between simulation and experiment  
\( v = \) velocity of airflow  
\( RANS = \) Reynolds-Averaged Navier-Stokes  
\( SA = \) Spalart-Allmaras  
\( WA = \) Wray-Agarwal  
\( UDF = \) user-defined function  
\( y^+ = \) non-dimensional wall distance
Acknowledgements

I would like to express my sincere gratitude to those who helped me during my research in Computational Fluid Dynamics laboratory at Washington University in St. Louis.

I am extremely grateful to Dr. Agarwal for his continuous help, guidance and encouragement in my research. Through my master research, I learned what CFD simulation is and how to do CFD simulation. Dr. Agarwal guided me in aerodynamics and fluid mechanics which was very helpful in my research project. In addition, his rich experience in CFD helped me overcome one difficulty after another during my Master’s research.

Secondly, I would like to express my appreciation to Leilei Ji, Siyuan Chen, Li Shan and Bryce Thomas for their time and efforts to help me. The patient instruction by Siyuan Chen helped me to know how to use ANSYS ICEM and ANSYS Fluent. Leilei Ji and Li Shan helped me to successfully improve the quality of the mesh, compile the user defined function to simulate using WA 2017m turbulence model in Fluent, and reduce the errors in simulations.

Thirdly, I would like to thank my thesis defense committee members, Dr. David Peters and Dr. Swami Karunamoorthy for serving on the committee and providing helpful advice.

Finally, I would like to thank all group members in the CFD lab for suggestions and help in my study and research. The harmonious communication environment in CFD lab allowed me to keep learning and making progress.

Boyu Wang

Washington University in St. Louis

May 2020
Dedicated to my mother Ling Jin and my father Wenjie Wang.
Abstract

Numerical Simulation of Flow past an Airfoil with Ice Accretion on Leading Edge

By

Boyu Wang

Master of Science in Mechanical Engineering

Washington University in St. Louis, 2021

Research Advisor: Professor Ramesh K. Agarwal

The focus of this research is on aerodynamic simulation of flow past NACA 23012 airfoil with clean surface and with ice accretion on its leading edge by using the commercial CFD solver ANSYS Fluent. Reynolds-Averaged Navier-Stokes (RANS) computations are performed using Spalart-Allmaras (SA) and Wray-Agarwal (WA) turbulence models. ANSYS mesh package ICEM is used to model the geometry and generate the mesh. The computations are performed at 0, 2, 4, 6, 8, 10, and 12 degrees angle of attack which are compared with experimental data. For the case of ice accretion at the leading edge, the physical geometry becomes more complex; therefore, AutoCAD is used first for geometry modelling and then ANSYS ICEM is used to generate an unstructured mesh. Again, ANSYS Fluent is used to conduct simulations at 0, 2, 4, 6, 8, 10, and 12 degrees angle of attack, and SA and WA turbulence models are employed. All cases are run at chord Reynolds number of 1.8 million and a Mach number of 0.18. It is shown that the recently developed WA model can be used to obtain accurate results and should be considered as an alternative turbulence model for computing such complex flows.
Chapter 1: Introduction

1.1: Background

The understanding of environmental conditions of an aircraft flight is an important part of aviation safety; the nature of atmospheric conditions and weather can lead to flight accidents. Among some weather-related atmospheric conditions such as high turbulence, ice formation on an aircraft surface is an area of major concern in aircraft safety. In flight icing has resulted in many catastrophic accidents in the aviation history. Many accidents have occurred due to undetected ice accretion or ineffective ice removal methods. Ice accretion on wings and engines can alter the performance characteristics and can result in sudden loss of stability and control [1, 2]. Figure 1 shows some examples of ice accumulation in flight on wing’s leading edge.

![Figure 1: Examples of accumulation of ice on wing’s leading edge in flight](image)

The results of the research related to the inflight icing accidents are meaningful for the improvement of aviation safety. Therefore, there have been both experimental and numerical investigations on the effect of ice accretion on the aerodynamic performance of airfoils and wings. Currently, there are two areas of interest in icing research. The first one is to investigate
the ice formation process under different atmospheric conditions. The other one is to investigate
the effect of ice accretion on the aerodynamic properties of the wings[3]. Aircraft test data with
ice accretions are very difficult to obtain because of the paucity of facilities and cost. Generally,
all three methods -- the flight tests, icing wind tunnel tests, and numerical simulation are
currently employed to investigate the effect of ice accretion. Among these, numerical simulations
have become very popular among the three because of the least cost and time required; however
it must be assured that the simulation data is accurate and reliable.

In theory, the aerodynamic performance of a full-scale airfoil with ice accretion can be
represented by a subscale airfoil with ice accretion. The only thing required is to reproduce the
ice geometry and match the Reynolds number and Mach number for subscale airfoil. However, it
is very difficult to achieve this goal. It is nearly impossible to reproduce the geometry at a
reduced scale and it is also difficult to match Reynolds number and Mach number in the wind
tunnel facility properly[4]. Therefore, generally the small scale geometry may have some
differences, and the matched Reynold number and Mach number may also have error. In both
simulation and wind tunnel experiments, ice shape castings are used to evaluate the aerodynamic
performance since ice is so easy to melt at standard temperature in the wind tunnel. If the ice
melts, the geometry of the shape will be different from its original shape, therefore the
aerodynamic performance will be uncertain. Ice shape casting creates the original ice shape using
another material by molding technique. The highest fidelity simulation and testing requires a 3D
casting of the original ice shape; however, it is very expensive and also requires a lot of time. As
a result, a 2D simulation of subscale model is often used to obtain a similar aerodynamic
performance as would be for a 3D model. Many studies have shown that this method can provide
accurate results if the 2D models are designed appropriately and the Reynolds number and Mach
number are properly matched. For simulations, several different Navier-Stokes flow solvers have been used by the researchers including OVERRFLOW, TURNS, and GT-Hybrid[5, 6].

Some research on the effects of ice accretion on helicopter rotors has also been conducted by simulation. The effect of ice accretion on rotors both in hover and in forward flight is very different than that on a fixed wing aircraft.

1.2: Objectives and Technical Approach

The objective of this study is to conduct numerical simulations of flow past an airfoil with ice accretion on its leading edge to study the effect of ice accretion on the aerodynamic performance characteristics of the airfoil. For this purpose, NACA23012 airfoil is chosen for simulation since the validation data for this airfoil is available in a NASA report [7] and in an AIAA paper[4]. Computational Fluid Dynamics (CFD) simulations are conducted using the commercial CFD solver ANSYS Fluent. Reynold-Averaged Navier-Stokes (RANS) equations are solved using the two one-equation linear eddy viscosity turbulence models, namely the Spalart-Allmaras (SA) model [8] and Wray-Agarwal (WA) model [9]. Simulations are performed for both the clean airfoil without ice and airfoil with ice accretion at various angles of attack and are compared with experimental data given in the NASA report[7]. Several kinds of ice shapes can occur on the airfoil surface, such as ice roughness, stream-wise ice shape, span-wise-ridge ice shape, and horn ice shape, etc. Figure 2 provides a qualitative picture of the four classifications and their geometry and their relative aerodynamic effect. The horn shape is the classic shape typical of glaze-type ice accretion, which occurs at the leading edge and has a very complex pattern. The horn shape ice can usually form through longer exposures under glaze and mixed icing conditions. The shape of the horn ice is generally characterized by its height, the angle it makes with the chord line, and its location indicated by s/c (the nondimensional surface
length along the airfoil profile)[10]. Figure 3 shows the EG1164 horn ice shape in the NASA report [7].

Figure 2: Qualitative description of relative aerodynamic effect of various ice shaped airfoils [10]

Figure 3: EG1164 horn shape ice accretion on an airfoil[7]

The irregular geometry of the ice accretion makes the computational simulations very challenging. Autodesk and ANSYS tools are employed to address this problem. Because of the complex geometry, AutoCAD is first used to model the geometry, which is then imported into
ANSYS ICEM. Unstructured grids are generated and Fluent is used to obtain the pressure and velocity distributions on the airfoil and aerodynamic coefficients such as lift and drag.

The objectives of research are:

1. Explore accurate numerical methods to simulate clean surface NACA23012 airfoil and prove the accuracy of the method by comparing the computations with the experimental data and controlling the difference between the two within 5%.

2. Use the selected numerical method for clean airfoil case to simulate the subscale airfoil with ice accretion to verify that the aerodynamic performance of a subscale airfoil at low Reynolds number can be directly related to that of a full-scale airfoil with ice accretion at high Reynolds number.

3. Evaluate the accuracy and efficiency of recently developed WA turbulence model compared to SA turbulence model.
Chapter 2: Physical Model and Mesh

2.1: Physical Model and Mesh of Clean Surface Airfoil

As mentioned in the last chapter, the selected airfoil is NACA23012 airfoil. The geometric model of NACA23012 airfoil (Figure 4) is constructed from the NACA 5 digit airfoil generator on the website ‘Airfoil Tools’. After obtaining a large number of airfoil coordinate points, the curve of the airfoil was generated in ANSYS ICEM. The actual chord length in the experiment is 18 inches (0.4572m). The chord length of the airfoil modeled in ANSYS ICEM is 1m, which is later set to the experimental value of 18 inches in ANSYS Fluent. The computational domain consists of a semi-circle ahead of the airfoil with a radius of 20m and the rectangular part of the domain at the rear of the airfoil is 40m in height and 20m in width. Mesh generation is also done using ICEM. The structured grid is generated in the computational domain. Due to the turbulent boundary layer near the surface of the airfoil, mesh in this region is refined and is much denser than the mesh in the far field. The computations are performed on a series of meshes so that it can be ensured that the solution is mesh independent and y+ is less than 1 for first grid point away from the surface of the airfoil[11]. Figure 5 shows the final mesh around the airfoil in the entire computational domain and Figure 6 shows the zoomed-in view of the mesh close to the airfoil. The number of cells is 217875. The number of faces is 436625. The number of nodes is 218750.

Figure 4: Geometry of NACA23012 airfoil
Figure 5: Structured mesh in the computational domain

Figure 6: Zoomed-in mesh near the airfoil

Figure 7 shows the pre-mesh quality under determinant 2×2×2 criterion. The model is a clean surface NACA23012 airfoil, so the geometry is much simpler than the model with ice. It is relatively easy to generate structured mesh to get more accurate results. Structured mesh has
higher quality than unstructured mesh, therefore structured meshes as shown in Figure 5 is used in the simulation. Furthermore, based on lot of numerical experiments in many applications, it has been found that the WA turbulence model is more sensitive than SA turbulence model to mesh quality especially close to the surface. Therefore, mesh with higher quality is necessary for accurate simulations. Determinant 2×2×2 criterion is one of the widely used methods to show the quality of the mesh. From Figure 7, it can be seen that even the minimum determinant is between 0.9 and 1; thus the quality of the mesh is very high. Because of the high quality of mesh, the simulation results calculated by both the SA and WA turbulence model are very close to the experiment data as shown in Chapter 4. The differences between computations and experiments is under 5%, for all angles of attack, and is less than 3% for some angles of attack.

![Figure 7: Mesh quality under determinant 2×2×2 criterion](image)

### 2.2: Physical Model and Mesh of Airfoil with Ice

For the ice accretion model, the geometry is much more complex near the leading edge. The horn shape ice $\text{EG1164}$ is shown in Figure 3. A set of coordinate points of the ice shape were extracted from figure 3. Then the ice shape and NACA23012 airfoil were combined in AutoCAD to get the complete geometry shown in Figure 8. Figure 9 shows the exact ice shape drawn in
AutoCAD. Compared to the geometry in the NASA report [7], it can be seen that the number of coordinate points are enough to fit an identical geometry. The final simulation results also show that the geometry is very close to the experimental geometry.

Figure 8: Geometry of NACA23012 airfoil with leading edge glazed with ice generated in AutoCAD

Figure 9: Zoomed-in ice shape on the leading edge generated in AutoCAD

The computational domain again consists of a semi-circle ahead of the airfoil with a radius of 20m and a rectangular part in the rest of the domain with 40m height and 20m width. The chord length of the airfoil in ICEM is again set as 1, and the size of the ice shape is related to the value of x/c. Because of the complex shape at the leading edge, unstructured triangular mesh was
generated since the EG1164 horn ice shape is so irregular that it is difficult to generate structured grids of high quality. Again, the grids near the surface of the airfoil are much denser than the grids in the far field, especially around the area of the ice accreted leading edge. Figure 10 shows the complete unstructured mesh in the computational domain. Figure 11 shows the mesh around the airfoil with ice accretion. Figure 12 shows the zoomed-in grids near the ice shape. The total number of cells is 584178. The number of faces is 877495. The number of nodes is 293317.

After importing the mesh in ANSYS Fluent, simulations are performed at various angles of attack. After a number of trials for generating high quality structured mesh, the difference between the simulation results and experimental data was again controlled close to 5% using both WA and SA turbulence models.

Figure 10: Mesh in the computational domain around airfoil with horn shape ice
After the description of both the geometry generation and the mesh generation, the next few chapters describe the numerical method and simulations in ANSYS Fluent.
Chapter 3: Governing Equations and Numerical Method

3.1: Governing Equations

In general, fluid flows contain laminar flow, transitional flow, and turbulent flow. In external flows when Reynolds numbers is larger than few millions of $O(10^6)$, the flows are turbulent, while those at low Reynolds numbers below $O(10^5)$ are usually remain laminar for streamlined bodies. Flows with Reynolds numbers between $O(10^5)$ and $O(10^6)$ are typically regarded as transitional flows. All these three kinds of flows can be described by the laws of conservation of mass, momentum, and energy which are expressed by the continuity equation, momentum equations, and energy equation. In this thesis, in order to express the velocity and pressure distribution in the flow field, the conservation of mass and momentum equations are solved numerically in the computational domain on a mesh with specified boundary conditions[12]. Equation (1) shows the governing mass conservation equation for incompressible flow, and equation (2) shows the governing momentum conservation equation for incompressible fluid.

$$\frac{\partial u_j}{\partial x_j} = 0$$

(1)

$$\rho \frac{\partial u_i}{\partial t} + \rho u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} (2\mu S_{ij})$$

(2)

The momentum equation can be obtained by the application of Newton’s second law in both integral and differential forms. Using the Stokes’ constitutive relation for Newtonian flow, the viscous forces can expressed explicitly in terms of the appropriate flow-field variables and the
final momentum equations are called the Navier-Stokes equations. Equation (3), (4) and (5) shows the incompressible Navier-Stokes equations in x, y, and z direction in Cartesian coordinates. To analyze incompressible viscous flow, these three equations and continuity equation are sufficient to solve [13].

\[
\frac{\partial u}{\partial t} + \rho \frac{u}{\partial x} + \rho v \frac{\partial u}{\partial y} + \rho w \frac{\partial u}{\partial z} = -\frac{\partial p}{\partial x} + \frac{1}{\rho} \left( \frac{\partial (\mu \nabla \cdot V)}{\partial x} + \frac{\partial}{\partial y} \left[ \mu \left( \frac{\partial u}{\partial x} + \frac{\partial u}{\partial y} \right) \right] + \frac{\partial}{\partial z} \left[ \mu \left( \frac{\partial u}{\partial z} + \frac{\partial u}{\partial x} \right) \right] \right) + \frac{\partial}{\partial y} \left[ \mu \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial y} \right) \right] + \frac{\partial}{\partial z} \left[ \mu \left( \frac{\partial u}{\partial z} + \frac{\partial v}{\partial z} \right) \right]
\]

(3)

\[
\frac{\partial v}{\partial t} + \rho \frac{u}{\partial x} + \rho v \frac{\partial v}{\partial y} + \rho w \frac{\partial v}{\partial z} = -\frac{\partial p}{\partial y} + \frac{1}{\rho} \left( \frac{\partial (\mu \nabla \cdot V)}{\partial y} + \frac{\partial}{\partial x} \left[ \mu \left( \frac{\partial v}{\partial x} + \frac{\partial u}{\partial x} \right) \right] + \frac{\partial}{\partial z} \left[ \mu \left( \frac{\partial v}{\partial z} + \frac{\partial v}{\partial y} \right) \right] \right) + \frac{\partial}{\partial x} \left[ \mu \left( \frac{\partial v}{\partial x} + \frac{\partial w}{\partial x} \right) \right] + \frac{\partial}{\partial z} \left[ \mu \left( \frac{\partial v}{\partial z} + \frac{\partial w}{\partial z} \right) \right]
\]

(4)

\[
\frac{\partial w}{\partial t} + \rho u \frac{\partial w}{\partial x} + \rho v \frac{\partial w}{\partial y} + \rho w \frac{\partial w}{\partial z} = -\frac{\partial p}{\partial z} + \frac{1}{\rho} \left( \frac{\partial (\mu \nabla \cdot V)}{\partial z} + \frac{\partial}{\partial x} \left[ \mu \left( \frac{\partial w}{\partial x} + \frac{\partial u}{\partial x} \right) \right] + \frac{\partial}{\partial y} \left[ \mu \left( \frac{\partial w}{\partial y} + \frac{\partial v}{\partial y} \right) \right] \right) + \frac{\partial}{\partial x} \left[ \mu \left( \frac{\partial w}{\partial x} + \frac{\partial v}{\partial x} \right) \right] + \frac{\partial}{\partial y} \left[ \mu \left( \frac{\partial w}{\partial y} + \frac{\partial w}{\partial y} \right) \right]
\]

(5)

However, it is well known that the Navier-Stokes equations cannot be solved analytically for any practical flow. Furthermore, for 3D turbulent flows at high Reynolds numbers, they require lot of computational power using the Direct Numerical Simulation (DNS) or even Large Eddy Simulation (LES). Therefore, Reynolds-Average Navier-Stokes (RANS) equations are employed for computation of practical/industrial turbulent flows. The RANS equations are time-averaged Navier-Stokes equations for describing the motion of turbulent fluid flow. The time-averaged Navier–Stokes equations or RANS equations can be expressed in Cartesian coordinates as [14]:

\[
\rho \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = \rho \bar{f}_i + \frac{\partial}{\partial x_j} \left[ -\bar{p} \delta_{ij} + \mu \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \rho \bar{u}_i \bar{u}'_j \right]
\]

(6)

where \( f_i \) is a vector representing the external forces. The term \( \rho \bar{u}_i \bar{u}'_j \) is called the “Reynolds
“Stress” which needs to be modeled. Boussinesq approximation is used to model this term which describes the stress strain relationship in terms of eddy viscosity. The modeling of Reynolds stress is called “Turbulence Modeling.” There are many turbulence models that have been developed over a century. The turbulence models used in this thesis are described in the next section.

3.2: Turbulence Models

Two turbulence models have been used in this thesis. One is the Spalart–Allmaras (SA) one equation turbulence model and the other is the Wray Agarwal (WA) 2017m one-equation turbulence model. One of the objectives of this research is to compare the accuracy of the two turbulence models.

3.2.1: SA Turbulence Model

The Spalart–Allmaras (SA) model is a one-equation linear eddy viscosity model which solves a modeled transport equation for the kinematic eddy turbulent viscosity. The Spalart–Allmaras model was designed specifically for aerospace applications involving wall-bounded flows and has been shown to give good results for boundary layers subjected to adverse pressure gradients [8]. The SA model is usually used for high Reynolds numbers, and therefore is widely used to compute turbulent flows. In SA model, a variable $\tilde{\nu}$ which is proportional to the eddy viscosity is used. Equation (7) shows the transport equation for turbulent kinematic eddy viscosity used in SA model[15]. The details are given in Reference [8].

\[
\frac{\partial \tilde{\nu}}{\partial t} + u_j \frac{\partial \tilde{\nu}}{\partial x_j} = c_{b1}(1 - f_{l2})S\tilde{v} - \left[ c_{w1}f_w - \frac{c_{b1}}{\kappa^2} f_{l2} \right] \left( \frac{\tilde{v}}{d} \right)^2 \\
+ \frac{1}{\sigma} \left[ \frac{\partial}{\partial x_j} \left( (\tilde{v} + \tilde{\nu}) \frac{\partial \tilde{\nu}}{\partial x_j} \right) + c_{b2} \frac{\partial \tilde{\nu}}{\partial x_i} \frac{\partial \tilde{\nu}}{\partial x_i} \right]
\]
3.2.2: WA Turbulence Model

The Wray-Agarwal (WA) model is also a one equation linear eddy viscosity model. The WA model was derived from two-equation k-omega closure and has been shown to give excellent results for a wide variety of wall-bounded and free shear flows[9]. A new variable $R$ is defined as $k/\omega$ in the WA model transport equation (8).

$$
\frac{\partial R}{\partial t} + \frac{\partial u_j R}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ (\sigma_R R + v) \frac{\partial R}{\partial x_j} \right] + C_1 S + f_1 C_{2k\omega} \frac{R}{S} \frac{\partial R}{\partial x_j} \frac{\partial S}{\partial x_j} - (1 - f_1) C_{2k\varepsilon} R^2 \left( \frac{\frac{\partial S}{\partial x_j} \frac{\partial S}{\partial x_j}}{S^2} \right) 
$$

(8)

The turbulent eddy viscosity is given by equation (9).

$$
\mu_t = \rho f_\mu R 
$$

(9)

$S$ is defined as the mean strain in equations (10) and (11).

$$
S = \sqrt{2S_{ij} S_{ij}} 
$$

(10)

$$
S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) 
$$

(11)

Equation (12) is a damping function that accounts for the wall blocking effect.

$$
f_\mu = \frac{\chi^3}{\chi^3 + C_\omega^3} 
$$

(12)

$$
\chi = \frac{R}{v} 
$$

(13)

$$
v = \frac{\mu}{\rho} 
$$

(14)

Equations (15) and (16) describe the switching function. In order to get good stability in computations, the value of $f_i$ should be no more than 0.9.
\[ f_1 = \min(tanh(arg_1^2), 0.9) \]  \hspace{1cm} (15)

\[ arg_1 = \frac{1 + \frac{d\sqrt{RS}}{v}}{1 + \left[ \frac{\max(d\sqrt{RS}, 1.5R)}{20v} \right]^2} \]  \hspace{1cm} (16)

where \( d \) is the minimum distance to the nearest wall.

The constants used in WA model are shown given in equations (17) – (20).

\[ C_1 = f_1(C_{1k\omega} - C_{1ke}) + C_{1ke} \]  \hspace{1cm} (17)

where \( C_{1k\omega} = 0.0829, C_{1ke} = 0.1127 \).

\[ \sigma_R = f_1(\sigma_{k\omega} - \sigma_{ke}) + \sigma_{ke} \]  \hspace{1cm} (18)

where \( \sigma_{k\omega} = 0.72, \sigma_{ke} = 1.0 \).

\[ C_{2k\omega} = \frac{C_{1k\omega}}{\kappa^2} + \sigma_{k\omega} \]  \hspace{1cm} (19)

\[ C_{2ke} = \frac{C_{1ke}}{\kappa^2} + \sigma_{ke} \]  \hspace{1cm} (20)

where \( \kappa = 0.41, C_w = 8.54 \)

All the information about WA model comes from the NASA website (https://turbmodels.larc.nasa.gov/wray_agarwal.html) and an AIAA paper[16].

### 3.3: ANSYS Fluent Numerical Setup

The double precision, pressure-based solver is used to simulate all cases in ANSYS Fluent. Both Spalart-Allmaras (SA) and Wray-Agarwal (WA2017m) turbulence models are used with the incompressible Reynolds-Averaged Navier-Stokes (RANS) equations. WA model was developed at Washington University by Dr. Ramesh Agarwal; therefore a User-Defined-Function (UDF) file needs to be imported into ANSYS Fluent since it is not included in Fluent like the SA model. All other parameters are set as default in ANSYS.
SIMPLE and SIMPLEC scheme with second-order discretization are chosen for the solution algorithm. Steady-state solvers are used in all cases. When changes in results of C_d and C_l are less than 1×10^{-4}, the calculations are considered converged.

As to the boundary conditions, the entire boundary which contains the semi-circle and the two connecting parallel horizontal lines of the computational domain is set as inlet, where the inlet boundary condition is set as the velocity-inlet. The selected model is a subscale NACA23012 airfoil with EG1164 horn ice, so the matched Reynolds number is 1.8×10^6 and the matched Mach number is 0.18 which gives the free stream velocity as 61.25 meters per second. The right boundary of computational domain which is a rectangle is set as the outlet, and it is set as pressure-outlet. The model is set as no-slip wall boundary condition.

As to the reference values, “compute from the inlet edge” is chosen. In order to match the size of the subscale geometry, the chord length is set as 18 inches or 0.4572 meters. Other values are set as default.
Chapter 4: Flow past NACA23012 Clean Airfoil

4.1: Flow Conditions

In all simulation cases, Reynolds number is $1.8 \times 10^6$ and Mach number is 0.18. The inlet air is set as ideal gas. The viscosity is $\mu = 1.7894 \times 10^{-5}$ kg/m·s and the density is $\rho = 1.176674$ kg/m$^3$. The inflow velocity is 61.25 m/s.

4.2: Simulation Results and Validations

Table 1 shows the simulation results using SA model and Table 2 shows the simulation result using the WA model. It can be easily seen that values of $\Delta C_l = (C_l \text{ simulation} - C_l \text{ experiment}) / C_l \text{ experiment}$, are less than 5 percent for both SA model and WA model. The computed results are in good agreement with experimental data. The drag coefficient $C_d$ is in general difficult to predict accurately by any computational technique, as a result $\Delta C_d$ is relatively large compared to $\Delta C_l$. Overall WA model results are closer to the experimental data.

Table 1: Simulation results for clean surface airfoil case using SA model

<table>
<thead>
<tr>
<th>Angle of attack</th>
<th>0</th>
<th>2</th>
<th>4</th>
<th>6</th>
<th>8</th>
<th>10</th>
<th>12</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cl(experiment)</td>
<td>0.13</td>
<td>0.35</td>
<td>0.57</td>
<td>0.80</td>
<td>1.01</td>
<td>1.22</td>
<td>1.41</td>
</tr>
<tr>
<td>Cl(simulation)</td>
<td>0.13362</td>
<td>0.36404</td>
<td>0.59081</td>
<td>0.81329</td>
<td>1.0309</td>
<td>1.2615</td>
<td>1.4257</td>
</tr>
<tr>
<td>$\Delta C_l$</td>
<td>2.78%</td>
<td>4.01%</td>
<td>3.65%</td>
<td>1.66%</td>
<td>2.07%</td>
<td>3.40%</td>
<td>1.11%</td>
</tr>
<tr>
<td>Cd(experiment)</td>
<td>0.007</td>
<td>0.006</td>
<td>0.007</td>
<td>0.007</td>
<td>0.008</td>
<td>0.010</td>
<td>0.011</td>
</tr>
<tr>
<td>Cd(simulation)</td>
<td>0.009985</td>
<td>0.010308</td>
<td>0.011219</td>
<td>0.012952</td>
<td>0.015512</td>
<td>0.017933</td>
<td>0.023985</td>
</tr>
<tr>
<td>$\Delta C_d$</td>
<td>42.64%</td>
<td>71.80%</td>
<td>60.27%</td>
<td>85.02%</td>
<td>93.90%</td>
<td>79.33%</td>
<td>118.05%</td>
</tr>
</tbody>
</table>
Table 2: Simulation results for clean surface airfoil using WA 2017m model

<table>
<thead>
<tr>
<th>Angle of attack</th>
<th>0</th>
<th>2</th>
<th>4</th>
<th>6</th>
<th>8</th>
<th>10</th>
<th>12</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cl(experiment)</td>
<td>0.13</td>
<td>0.35</td>
<td>0.57</td>
<td>0.80</td>
<td>1.01</td>
<td>1.22</td>
<td>1.41</td>
</tr>
<tr>
<td>Cl(simulation)</td>
<td>0.12965</td>
<td>0.34257</td>
<td>0.56076</td>
<td>0.79322</td>
<td>0.98544</td>
<td>1.2381</td>
<td>1.3862</td>
</tr>
<tr>
<td>ΔCl</td>
<td>2.69%</td>
<td>2.12%</td>
<td>1.62%</td>
<td>0.85%</td>
<td>2.43%</td>
<td>1.48%</td>
<td>1.69%</td>
</tr>
<tr>
<td>Cd(experiment)</td>
<td>0.007</td>
<td>0.006</td>
<td>0.007</td>
<td>0.007</td>
<td>0.008</td>
<td>0.010</td>
<td>0.011</td>
</tr>
<tr>
<td>Cd(simulation)</td>
<td>0.009929</td>
<td>0.010003</td>
<td>0.011371</td>
<td>0.013020</td>
<td>0.015206</td>
<td>0.017302</td>
<td>0.021573</td>
</tr>
<tr>
<td>ΔCd</td>
<td>41.84%</td>
<td>66.72%</td>
<td>62.44%</td>
<td>86.00%</td>
<td>90.08%</td>
<td>73.02%</td>
<td>96.12%</td>
</tr>
</tbody>
</table>

Figure 13 shows the comparison of Cl between simulations and experimental data at various angles of attack using both the SA and WA turbulence models. It can be seen that overall WA model gives slightly more accurate results than SA model. From 0 to 10 degree angle of attack, the line of Cl vs. α is linear and it deviates slightly from linearity at α = 12 degree. It shows that stall doesn’t occur. Figure 14 shows the graph of Cd versus angle of attack and compares the simulation results with experimental. The results obtained with WA model are slightly more accurate than the results obtained with SA model. However, as expected they differ a great deal from experimental data. It should be noted that there is also uncertainty in experimental data.

Figure 13: Comparison of computed lift coefficient with experiment data using SA and WA turbulence models
4.3: Pressure and Velocity Contours

4.3.1: Pressure Contours at Various Angles of Attack

Figure 15 shows the pressure contours around the clean NACA23012 airfoil at seven angles of attack using WA model. It should be noted that similar pressure contours are obtained using SA model, but they are not presented here for the sake of brevity. From these figures, it can be seen that there is a region of high pressure near the leading edge, and a region of low pressure on the upper surface of the airfoil as expected. When the angle of attack increases, the pressure on the lower surface of the airfoil becomes higher. Because the pressure on the lower surface is much higher than that on the upper surface, it results in the increased lift on the airfoil.

Consider the case $\alpha = 0^\circ$ and $\alpha = 8^\circ$ for comparison. When $\alpha = 0^\circ$, the highest pressure is distributed near the leading edge, because the fluid comes in the normal direction from left side. The highest pressure is nearly 2320Pa. The difference in pressure between the upper and lower
surface of the airfoil is not very large, therefore the lift force is small reflected in the value of Cl. When \( \alpha = 8^\circ \), the high-pressure distribution at the lower surface of the airfoil increases to nearly 2310Pa, while the pressure on the upper surface of the airfoil near the leading edge drops to -6660Pa compared to the ambient pressure. Because of the angle of attack, the position of the high-pressure distribution on the lower surface of the airfoil is where the inlet flow enters the domain. From the magnitude of pressure, it is obvious that the pressure distribution around the airfoil develops smoothly. Even at \( \alpha = 12^\circ \), the pressure distribution seems to be smooth and regular around the airfoil. It can also be seen in the graph in Figure 13 that Cl doesn’t reach the maximum of Cl when \( \alpha = 12^\circ \). Before \( \alpha = 12^\circ \), the Cl versus angle of attack curve is approximately linear verified through both the experimental data and the simulations.

(a) \( \alpha = 0^\circ \)  
(b) \( \alpha = 2^\circ \)  
(c) \( \alpha = 4^\circ \)  
(d) \( \alpha = 6^\circ \)  
(e) \( \alpha = 8^\circ \)  
(f) \( \alpha = 10^\circ \)
(g) $\alpha = 12^\circ$

Figure 15: Pressure contours around the airfoil at different angles of attack

4.3.2: Velocity Contours at Various Angles of Attack

Figure 16 shows the velocity contours around the clean NACA23012 airfoil at seven angles of attack using the WA 2017m turbulence model. It should be noted that similar velocity contours are obtained using the SA turbulence model, but they are not presented here for the sake of brevity. The distribution of the velocity doesn’t have huge difference between the upper surface and lower surface of the airfoil at 0 degree angle of attack as expected. The subtle difference comes from the lack of symmetry of NACA23012 airfoil about its chord. When the angle of attack increases, the difference in velocity distribution between the upper and lower surface becomes larger. According to Bernoulli equation, a lower velocity contributes to a higher pressure, and a higher velocity contributes to a lower pressure, therefore the velocity distribution contours correspond to the pressure distribution contours.

Similar to the analysis of pressure distribution in the previous section, consider three cases of $\alpha = 0^\circ$, $\alpha = 8^\circ$ and $\alpha = 12^\circ$ for comparison. When $\alpha = 0^\circ$, the difference in the maximum velocity between the upper and lower surface of the airfoil is not very large. The velocity on the upper surface is slightly higher and can reach 80.9m/s; the free stream velocity at inlet is 61.25m/s. With the increase in angle of attack, when $\alpha = 8^\circ$, the velocity on the upper surface can
reach 122m/s, and when $\alpha = 12^\circ$, the maximum velocity on the upper surface can reach 162m/s. The high velocity can contribute to very low pressure. At the lower surface, in the figures the color changes from green to blue. The velocity is 3.23m/s, and even nearly 0 m/s at some points. The large difference in the velocity between the upper and lower surface contributes to a large difference in pressure which contributes to the lift force. From the magnitude of the velocity, it can be seen that the difference becomes larger and larger as the angle of attack increases, which contributes to the increase in lift force.

(a) $\alpha = 0^\circ$  
(b) $\alpha = 2^\circ$

(c) $\alpha = 4^\circ$  
(d) $\alpha = 6^\circ$

(e) $\alpha = 8^\circ$  
(f) $\alpha = 10^\circ$
Figure 16: Velocity contours around the airfoil at various angles of attack

4.3.3: Velocity Vectors at Various Angles of Attack

The Figures 17 - 19 show the velocity vectors around the airfoils at various angles of attack. Velocity vectors provide excellent visualization of the flow around the airfoil depicting details of the wake structure. Here, only the velocity vectors at 0, 6, and 12 degree angle of attack are shown. These three angles of attack represent low, medium, and high angles of attack respectively.

Figure 17: Velocity vectors around NACA23012 airfoil at $\alpha = 0^\circ$
Figure 18: Velocity vectors around NACA23012 airfoil at $\alpha = 6^\circ$

Figure 19: Velocity vectors around NACA23012 airfoil at $\alpha = 12^\circ$
The figures of velocity vectors provide similar information as the velocity contours. However, these are more instructive. As the angle of attack increases, the exact inflow velocity position can be easily seen in the velocity vector figures. They also show the exact flow path from the leading edge on the airfoil towards the trailing edge. The stagnation points can be seen in Figures 17(c) - 19(c). When the angle of attack increases, the stagnation point moves down from the leftmost point of the airfoil towards the lower surface of the airfoil. The velocity of the flow near the surface can be seen in Figures 17(b) - 19(b). The information from these figures is consistent with the knowledge of aerodynamics. Figure 19(d) shows that there is no separation at 12 degree angle of attack; therefore there is no separated flow in case of clean surface airfoil from $\alpha = 0$ to 12 degrees.
Chapter 5: Flow past NACA23012 Airfoil with Ice Accretion near the Leading Edge

5.1: Flow Conditions

In all simulation cases, Reynolds number is $1.8 \times 10^6$ and Mach number is 0.18. The inlet air is set as an ideal gas. The viscosity is $\mu = 1.7894 \times 10^{-5}$ kg/m·s and the density is $\rho = 1.176674$ kg/m$^3$. The inflow velocity is 61.25 m/s.

5.2: Simulation Results and Validations

Table 3 shows the simulation results of flow past NACA23012 airfoil with ice accretion on the leading edge using SA turbulence model, and Table 4 shows the simulation results of flow past NACA23012 airfoil with ice accretion on the leading edge using WA 2017m turbulence model. It can be noted that the value of $\Delta C_l$, when the angle of attack increases to 8, 10, and 12 degrees is relatively high compared to that for the clean airfoil case in Table 1 and Table 2 in Chapter 4; they reach 5% and even 6%. It can be attributed to several possible reasons listed below.

1. The mesh quality of unstructured triangular mesh is not as good as that of the structured mesh used in case of clean airfoil simulations which may contribute to low accuracy of simulation results.

2. The model in simulation is a subscale airfoil with lower Reynolds number and Mach number, therefore the matched Reynolds number and Mach numbers for the subscale model may not reflect the aerodynamics of the original airfoil and flow conditions.
previously.

3. The EG1164 horn ice has a complex irregular geometry, therefore the process of generating geometry may have some errors which may contribute to errors in results.

4. The model used in the experiment was created by 3D casting by molding, therefore the model may also have error in fabrication.

5. When the angles of attack are very high, it is more difficult to compute and get convergent solution in Fluent or other CFD software.

Nevertheless, the difference in the simulations and experimental results is still within the acceptable range. This proves that the aerodynamic performance of a full-scale airfoil with ice accretion can be represented by a subscale airfoil with ice accretion. A large number of simulations as well as experiments are needed for further verification.

As shown in Table 3 and Table 4, experimental data for Cd is not available; therefore the results of simulations could not be compared with experiment. The values of ∆Cd for α = 0 to 8 degree are still very high as expected. However, surprisingly the values of Cd at α = 6 and 8 degree are much closer to the experimental data compared to the clean airfoil case shown in Table 1 Table 2; ∆Cd is within 10% and in cases at α = 0 to 4 degrees within 5%.

A very important objective of this research is to compare the accuracy and efficiency of recently developed WA turbulence model and compare its accuracy with SA turbulence model. It is found that overall WA model results are more accurate than SA model results. At low angles of attack, the results from the two models are very close. However, at high angles of attack of 6 and 8 degree, the difference between the results of SA and WA model can be seen more clearly, and WA model results are clearly better.

Figure 20 shows the comparison of Cl between simulations and experimental data at various
angles of attack using both the SA and WA 2017m turbulence models and Figure 21 shows the comparison of Cd. The simulation results are quite good compared to the experiment.

Table 3: Simulation results for iced airfoil using the SA model

<table>
<thead>
<tr>
<th>Angle of attack</th>
<th>0</th>
<th>2</th>
<th>4</th>
<th>6</th>
<th>8</th>
<th>10</th>
<th>12</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cl(experiment)</td>
<td>0.14</td>
<td>0.35</td>
<td>0.55</td>
<td>0.71</td>
<td>0.82</td>
<td>0.85</td>
<td>0.73</td>
</tr>
<tr>
<td>Cl(simulation)</td>
<td>0.13707</td>
<td>0.34371</td>
<td>0.53302</td>
<td>0.68749</td>
<td>0.77319</td>
<td>0.79885</td>
<td>0.69899</td>
</tr>
<tr>
<td>ΔCl</td>
<td>2.09%</td>
<td>1.80%</td>
<td>3.09%</td>
<td>3.17%</td>
<td>5.71%</td>
<td>6.02%</td>
<td>4.25%</td>
</tr>
<tr>
<td>Cd(experiment)</td>
<td>0.016</td>
<td>0.017</td>
<td>0.021</td>
<td>0.027</td>
<td>0.050</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cd(simulation)</td>
<td>0.011020</td>
<td>0.011875</td>
<td>0.015582</td>
<td>0.024752</td>
<td>0.046875</td>
<td>0.083749</td>
<td>0.138497</td>
</tr>
<tr>
<td>ΔCd</td>
<td>31.12%</td>
<td>30.15%</td>
<td>25.80%</td>
<td>8.33%</td>
<td>6.25%</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 4: Simulation results for iced airfoil using WA model

<table>
<thead>
<tr>
<th>Angle of attack</th>
<th>0</th>
<th>2</th>
<th>4</th>
<th>6</th>
<th>8</th>
<th>10</th>
<th>12</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cl(experiment)</td>
<td>0.14</td>
<td>0.35</td>
<td>0.55</td>
<td>0.71</td>
<td>0.83</td>
<td>0.85</td>
<td>0.73</td>
</tr>
<tr>
<td>Cl(simulation)</td>
<td>0.14001</td>
<td>0.35772</td>
<td>0.53413</td>
<td>0.69475</td>
<td>0.78662</td>
<td>0.81202</td>
<td>0.70002</td>
</tr>
<tr>
<td>ΔCl</td>
<td>0.00%</td>
<td>2.21%</td>
<td>2.89%</td>
<td>2.15%</td>
<td>5.23%</td>
<td>4.47%</td>
<td>4.11%</td>
</tr>
<tr>
<td>Cd(experiment)</td>
<td>0.016</td>
<td>0.017</td>
<td>0.021</td>
<td>0.027</td>
<td>0.050</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cd(simulation)</td>
<td>0.013158</td>
<td>0.013921</td>
<td>0.016474</td>
<td>0.025908</td>
<td>0.046040</td>
<td>0.087998</td>
<td>0.140592</td>
</tr>
<tr>
<td>ΔCd</td>
<td>17.76%</td>
<td>18.11%</td>
<td>21.55%</td>
<td>4.04%</td>
<td>7.92%</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Figure 20: Comparison of computed lift coefficient using SA and WA turbulence models with 3D casting experimental data
Figure 21: Comparison of computed drag coefficient using SA and WA turbulence models with 3D casting experimental data

5.3: Pressure and Velocity Contours

5.3.1: Pressure Contours at Various Angles of Attack

Figure 22 shows the pressure distribution around the NACA23012 airfoil with ice accretion at different angles of attack. The pressure contours of the iced airfoil cases have large difference with the pressure contours of the clean surface airfoil cases. The biggest difference is the pressure distribution around the EG1164 horn ice. Because of the horn ice, the pressure around the leading edge changes significantly. The pressure behind the horn tips becomes very low and separation bubbles are generated. When angle of attack increases to 10 or 12 degree, the influence is large. For the clean surface airfoil, the pressure on the whole lower surface is relatively high to generate significant lift. In case of airfoil with ice accretion on the leading edge, the pressure on the bottom surface decreases significantly. In particular at 12 degree angle of attack, the color near the bottom surface of the airfoil changes from red to yellow indicating that
the pressure changes from nearly +2300Pa to nearly +100Pa compared to the ambient pressure. Therefore, it can be inferred that when the AOA is more than 12 degrees, the pressure on the bottom surface will continue to decrease and may even become negative compared to the ambient pressure, which will contribute the large decrease in lift. This can be seen in the figure of Cl versus angle of attack. When the angle of attack is more than 8 degree, the Cl vs. α curve is not linear and bends downward showing decrease in lift. Compared to the clean surface airfoil, the angle of attack of the maximum lift coefficient becomes much smaller. All of these are the effects of the horn ice shape on the leading edge of the airfoil.

(a) $\alpha = 0^\circ$  
(b) $\alpha = 2^\circ$  
(c) $\alpha = 4^\circ$  
(d) $\alpha = 6^\circ$  
(e) $\alpha = 8^\circ$  
(f) $\alpha = 10^\circ$
(g) $\alpha = 12^\circ$

Figure 22: Pressure contours around the airfoil with ice accretion at different angles of attack

5.3.2: Velocity Contours at Various Angles of Attack

Figure 23 shows the velocity distribution around the NACA23012 airfoil with ice accretion at the leading edge at different angles of attack. The velocity contours have behavior corresponding to the pressure contours in accordance with the Bernoulli equation. From the velocity contours, it can be seen that there are large blue-color regions near the trailing edge when the angles of attack are high which are indicative of flow separation flow in iced airfoil cases. The vortices at the trailing edge start to occur and become larger and larger when the angle of attack increases. In particular, when AOA is 12 degree, because of the formation of turbulent vortices on the upper surface of the airfoil, the velocity on the upper surface becomes lower, and therefore the pressure on the upper surface becomes very low, especially near the trailing edge. The vortices contribute to the redistribution of pressure, thus compared to the clean surface airfoil cases; pressure distribution is very different on the iced airfoil cases. Near the horn ice, especially behind the ice tips, there are also blue-color regions where vortices occur. For the airfoil with ice accretion on leading edge, the stagnation points always occur on the tips of the ice shape, and the separation location remains on the tips of the horn ice over a large angle of attack range. There are separated flow regions behind the stagnation points. This also contributes
to the redistribution of pressure.

Figure 23: Velocity contours around the airfoil with ice accretion at different angles of attack
5.3.3: Velocity Vectors at Various Angles of Attack

Figure 24 shows the velocity vectors around the airfoil with ice accretion on the leading edge at various angles of attack. This figure shows the flow path of flow past the airfoil more clearly. Figures 24 (a) – 24(g) correspond to the velocity contours. These figures show that at lower angles of attack, the main effect of the horn ice is the change in the magnitude of the velocity on the upper and lower surface of the airfoil compared to the clean airfoil cases; however at high angles of attack, there are vortices that occur at the trailing edge which influence the aerodynamic performance of the airfoil significantly such as the lift force and drag force. Some zoomed-in figures are shown in the next two sections in order to illustrate the velocity distribution at the leading edge and trailing edge more clearly.

(a) $\alpha = 0^\circ$    (b) $\alpha = 2^\circ$

(c) $\alpha = 4^\circ$    (d) $\alpha = 6^\circ$
5.3.4: Zoomed-in View of Velocity Contours and Velocity Vectors at Leading Edge

Figures 25 - 27 show the zoomed-in velocity contours and velocity vectors around the leading edge of the iced airfoil at α = 0, 6, and 12 degree respectively. From the figures of velocity vectors, it can be seen that there are three vortex regions near the leading edge. The most obvious region is the one behind the upper ice tip. At first, there is just a small separation bubble at small angle of attack. With increase in the angles of attack, this bubble becomes larger and larger. When AOA is 0 degree, the inflow location is very close to the upper ice tip, therefore the bubble behind it is small, and the separated flow reattaches quickly to the airfoil surface downstream. When the angles of attack are 10 to 12 degree, the location of the inflow
moves down which is closer to the lower ice tip, thus the vortices behind the upper ice tip become larger. On the other hand, the separation bubble behind the lower ice tip is very small at first and with increase in angle of attack, the bubble becomes larger. The middle separation bubble is very small because the ice tip is very small, and when the inflow reaches the tip directly, there is no bubble behind this tip as shown in Figure 27. There are only two bubbles in Figure 27 and three bubbles in Figure 25 and Figure 26.

Furthermore, the stagnation point is usually located on the ice shape and the boundary layer cannot negotiate the large adverse pressure gradient encountered at the ice tips. Therefore, the separation location always remains at the ice tips at in the large angles of attack range; it is obvious from the figures of velocity vectors. The presence of the bubbles increases the drag force significantly.

![Figure 25: Zoomed-in velocity contours and velocity vectors around the horn ice at \(\alpha = 0^\circ\)](image)

![Figure 26: Zoomed-in velocity contours and velocity vectors around the horn ice at \(\alpha = 6^\circ\)](image)
Figure 27: Zoomed-in velocity contours and velocity vectors around the horn ice at $\alpha = 12^\circ$

5.3.5: Zoomed-in View of Velocity Contours and Velocity Vectors at Trailing Edge

Figures 28 - 32 show the zoomed-in view of velocity contours and velocity vectors around the trailing edge at 0, 6, 8, 10 and 12 degree respectively. It can be seen that when AOA is less than 8 degree, the flow at the trailing edge is not separated and doesn’t form vortices. However, it is clear that the velocity at AOA of 6 degrees is much less compared to the velocity near the trailing edge of the clean surface airfoil; it is due to the influence of the ice accretion on leading edge. Figure 30 shows that when AOA is 8 degree, there are very small vortices near the trailing edge. In Figure 31, the vortices become larger at 10 degree angle of attack, and in Figure 32, the turbulent wake occurs. There is flow separation flow near the trailing edge in iced airfoil cases at high angles of attack. There is no turbulent wake near the trailing edge of the clean surface airfoil at angles of attack of 10 and 12 degree. Therefore, this is also the influence of the horn ice accretion on leading edge. The vortices at the trailing edge cause a large redistribution of pressure that results in pitching moment changes and decreased lift.
Figure 28: Zoomed-in velocity contours and velocity vectors near the trailing edge at $\alpha = 0^\circ$

Figure 29: Zoomed-in velocity contours and velocity vectors near the trailing edge at $\alpha = 6^\circ$

Figure 30: Zoomed-in velocity contours and velocity vectors near the trailing edge at $\alpha = 8^\circ$

Figure 31: Zoomed-in velocity contour and velocity vectors around the trailing edge at $\alpha = 10^\circ$
Figure 32: Zoomed-in velocity contours and velocity vectors near the trailing edge at $\alpha = 12^\circ$
Chapter 6: Conclusions and Future Work

6.1: Conclusions

Computations are performed for low Mach number flow past NACA23012 clean airfoil at seven angles of attack from 0 to 12 degree using the Reynolds-Averaged Navier-Stokes (RANS) equations with Spalart-Allmaras (SA) and Wray-Agarwal (WA) 2017m turbulence models. The computations agree well with the experimental data for lift coefficient; however there is significant difference in the computations and experiment for drag coefficient as expected. A careful examination of results in Table and Table 2 shows that overall WA model results have better accuracy compared to SA model results, especially at high angles of attack.

Computations are also performed for low Mach number flow past NACA23012 airfoil with ice accretion on the leading edge at seven angles of attack from 0 to 12 degree using the Reynolds-Averaged Navier-Stokes (RANS) equations with Spalart-Allmaras (SA) and Wray-Agarwal (WA) 2017m turbulence models. The computations agree reasonably well with the 3D casting experimental data for lift coefficient; the difference is within 6 percent. For the drag coefficient, the minimum difference is at 6 degree angle of attack of nearly 4% using WA model. The comparison of results in Table 3 and Table 4 shows that overall WA model results have better accuracy than the SA model results.

The experimental data was obtained by testing a full-scale airfoil model both for clean and horn-ice shape airfoil. The simulation models are sub-scale airfoils with matched low Reynolds number and low Mach number. The computations show that the aerodynamic performance of a full-scale airfoil with ice can be represented by a subscale airfoil with ice with the matched Reynolds number and Mach number in the paper within the acceptable range of error.
Figure 33: Comparison of lift coefficient between clean surface airfoil and iced airfoil cases

Figure 34: Comparison of drag coefficient between clean surface and iced airfoil cases.

Figure 33 shows the comparison of Cl between clean surface airfoil and iced airfoil cases. Figure 34 shows the comparison of Cd between clean surface and iced airfoil cases. The Cl vs. \( \alpha \) and Cd vs. \( \alpha \) graphs of the experimental data and simulation results using two different
turbulence models show that the values of Cl in clean surface cases are much larger than the values of Cl in iced cases, while the values of Cd in clean surface cases are much smaller than the values of Cd in iced cases, especially at high angles of attack. The accretion of horn ice on the leading edge contributes to decrease in lift and increase in drag.

6.2: Future Work

In this thesis, only EG1164 horn ice shape was selected for simulation. There are many other ice shapes which should be investigated to assess their effect on the aerodynamic performance of airfoils/wings. In addition, the airfoil chosen in this research is NACA23012 airfoil. There are many other series of airfoils. The aerodynamic performance of different airfoils with different ice shapes should be investigated. The formation of ice in flight conditions should also be investigated.
References


Vita

Boyu Wang

Degrees

M.S. Mechanical Engineering, Washington University in St. Louis, May 2021
B.S. Mechanical Engineering, Northeastern University in China, June 2019

Publications