2014

Study on the Vortex Wake of an Airfoil Equipped with Flexible Trailing Edge Fringes

Zhengkai He
Wright State University

Follow this and additional works at: http://corescholar.libraries.wright.edu/etd_all
Part of the Mechanical Engineering Commons

Repository Citation

This Thesis is brought to you for free and open access by the Theses and Dissertations at CORE Scholar. It has been accepted for inclusion in Browse all Theses and Dissertations by an authorized administrator of CORE Scholar. For more information, please contact corescholar@www.libraries.wright.edu.
Study on the Vortex Wake of an Airfoil Equipped with Flexible Trailing Edge Fringes

A thesis submitted in partial fulfillment of the requirements for the degree of Master of Science in Engineering

by

Zhengkai He
B.Eng., 2007, Central South University
M.S.Eng., 2010, Dalian Jiaotong University

2014
Wright State University
I HEREBY RECOMMEND THAT THE THESIS PREPARED UNDER MY SUPERVISION BY Zhengkai He ENTITLED Study on the Vortex Wake of an Airfoil Equipped with Flexible Trailing Edge Fringes BE ACCEPTED IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE OF Master of Science in Engineering.

Zifeng Yang
Thesis Director

George P.G. Huang, Ph.D., Chair
Department of Mechanical and Materials Engineering
College of Engineering and Computer Science

Committee on Final Examination

Zifeng Yang, Ph.D.

Joseph Shang, Ph.D.

Ha-Rok Bae, PhD

Robert E. W. Fyffe, Ph.D.
Vice President of Research and Dean of the Graduate School
ABSTRACT
Zhengkai He. M.S.Egr, Department of Mechanical and Materials Engineering, Wright State University, 2014. Study on the Vortex Wake of an Airfoil Equipped with Flexible Trailing Edge Fringes.

With the inspiration of owl’s silent flight, a traditional airfoil S833 equipped with flexible fringes on the trailing edge is investigated through numerical simulation and experiments in a wind tunnel. The newly constructed airfoil is modeled and numerically investigated. An incompressible, 2D and viscous flow solver in the Computational Fluid Dynamics (CFD) software FLUENT is utilized to conduct the numerical simulation on the vortex flow feature in the wake of the airfoil. A User Defined Function code was applied to generate the defined motion of flexible fringes. The effects of the flapping frequency of the fringe and the deformation pattern of the fringe are investigated in the parametric study. On the other hand, the airfoil model with the flexible fringe is manufactured for the experimental study. A digital Particle Image Velocimetry (PIV) system is employed to investigate the flow structure in the wake and the deformation of the flexible fringes. The motion of the trailing edge fringes is extracted from the experimental measurements as the input for CFD simulation. It has been found that the addition of the flexible fringe has a significant effect on the flow characteristics in the vortex wake downstream of the airfoil as well as the aerodynamic performance of the airfoil.
# Table of Contents

1  **Introduction** .................................................................................................................. 1

2  **Methodology** .................................................................................................................. 7
   2.1 Practical Advantages of Employing CFD ................................................................. 7
   2.2 Fluent Software ............................................................................................................. 8
   2.3 Turbulence Model ....................................................................................................... 10
   2.4 One-equation and two-equation models ................................................................. 13
   2.5 The $\kappa$-$\varepsilon$ Model ....................................................................................... 14
   2.6 Initial and Boundary Conditions ............................................................................. 15
   2.7 Experimental apparatus ............................................................................................ 17
       2.7.1 Experimental rig and test model ..................................................................... 17
       2.7.2 Experimental setup ......................................................................................... 17

3  **Computational Grid** ....................................................................................................... 20
   3.1 Model and mesh .......................................................................................................... 20
   3.2 Code validation ............................................................................................................ 21
       3.2.1 Convergence criteria ....................................................................................... 22
       3.2.2 Different mesh method .................................................................................... 22
       3.2.3 Different size of domain .................................................................................. 25
       3.2.4 Different viscous model .................................................................................. 28
       3.2.5 The plunging motion of rigid airfoil ................................................................. 36

4  **Results and discussion** .................................................................................................. 40
   4.1 A comparison of the original model and the flexible fringe model ................. 40
   4.2 The different frequency of the vibration ............................................................... 45
       4.2.1 Comparison at $t=2T$ .................................................................................... 46
       4.2.2 Comparison at $t=3T$ .................................................................................... 50
       4.2.3 Comparison at $t=3s$ ..................................................................................... 53
4.3 The different wave motion of the fringe ........................................56
5 Conclusion ..........................................................................................63
6 Future work ..........................................................................................66
7 Reference ..............................................................................................68
## List of Figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Figure 1 – The structure of the owl’s wing</td>
<td>3</td>
</tr>
<tr>
<td>Figure 2 - The flow condition around the airfoil</td>
<td>4</td>
</tr>
<tr>
<td>Figure 3 - Basic Program Structure</td>
<td>9</td>
</tr>
<tr>
<td>Figure 4 - Schematic of the experimental setup</td>
<td>19</td>
</tr>
<tr>
<td>Figure 5 - The model of the airfoil</td>
<td>21</td>
</tr>
<tr>
<td>Figure 6 - The Computational Domain and boundary condition</td>
<td>21</td>
</tr>
<tr>
<td>Figure 7 - Four different mesh methods</td>
<td>23</td>
</tr>
<tr>
<td>Figure 8 - The $C_l$ of the different cases under different AOA</td>
<td>24</td>
</tr>
<tr>
<td>Figure 9 - Different size of the domain</td>
<td>27</td>
</tr>
<tr>
<td>Figure 10 - The $C_l$ of the different cases under different AOA</td>
<td>28</td>
</tr>
<tr>
<td>Figure 11 - The definition of the turbulence model</td>
<td>32</td>
</tr>
<tr>
<td>Figure 12 - $C_d$ vs AOA under different turbulence models and experiments</td>
<td>33</td>
</tr>
<tr>
<td>Figure 13 - $C_l$ vs AOA under different turbulence models and experiments</td>
<td>34</td>
</tr>
<tr>
<td>Figure 14 - $C_p$ of the airfoil S834 at AOA of 8º</td>
<td>35</td>
</tr>
<tr>
<td>Figure 15 - $C_p$ of the airfoil S834 at AOA of 12º</td>
<td>35</td>
</tr>
<tr>
<td>Figure 16 – The plunging motion of the airfoil</td>
<td>36</td>
</tr>
<tr>
<td>Figure 17 – The mesh around airfoil</td>
<td>37</td>
</tr>
<tr>
<td>Figure 18 –The comparison of the pressure coefficient distribution</td>
<td>38</td>
</tr>
</tbody>
</table>
Figure 19 – The comparison of the drag coefficient at third time period .......... 39
Figure 20 - The comparison of the vorticity distribution at different time period ...... 42
Figure 21 - The comparison of the pressure distribution at different time period ...... 44
Figure 22 - The comparison of the $C_p$ of the two models at t=3s ......................... 45
Figure 23 - The comparison of the vorticity distribution at t=2T ......................... 47
Figure 24 - The comparison of the pressure distribution at t=2T ......................... 48
Figure 25 - The comparison of the $C_p$ distribution at different cases at t=2T .......... 49
Figure 26 - The comparison of the vorticity distribution at t=3T ......................... 50
Figure 27 - The comparison of the pressure distribution at t=3T ......................... 51
Figure 28 - The comparison of the $C_p$ distribution at different cases at t=3T .......... 52
Figure 29 - The comparison of the vorticity distribution at t=3s ......................... 53
Figure 30 - The comparison of the pressure distribution at t=3s ......................... 54
Figure 31 - The comparison of the $C_p$ distribution at different cases at t=3s ........... 55
Figure 32 - Two different vibration mode of the flexible fringe ......................... 56
Figure 33 - The comparison of the vorticity distribution at t=3s ......................... 57
Figure 34 - The comparisons of the pressure distribution at t=3s ......................... 57
Figure 35 - The comparisons of the vorticity distribution at t=4s ......................... 58
Figure 36 - The comparisons of the pressure distribution at t=4s ......................... 59
Figure 37 - The comparisons of the $C_p$ distribution at different cases at t=3s .......... 60
Figure 38 - The comparison of turbulence kinetic energy in different cases ............. 61
Figure 39 - 3D model with rectangular fringe ............................................... 66
Figure 40 - 3D model with cylinder fringe .................................................. 67
List of Tables

<table>
<thead>
<tr>
<th>Table</th>
<th>Reference value of calculation</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Table 1</td>
<td>Reference value of calculation</td>
<td>33</td>
</tr>
<tr>
<td>Table 2</td>
<td>Reference value of calculation</td>
<td>40</td>
</tr>
<tr>
<td>Table 3</td>
<td>Reference value of calculation</td>
<td>46</td>
</tr>
</tbody>
</table>
Nomenclature

C  Chord length (m)

C_\text{p}  Pressure coefficient

C_\text{f}  Skin friction coefficient

k  Turbulence kinetic energy

\varepsilon  Turbulent dissipation

v  Inflow velocity

\omega  Specific dissipation rate; vorticity vector magnitude

\theta  The momentum thickness

P  Production tensor

P_\text{\infty}  Pressure in free stream

x  x-direction location

y  y-direction location

d  Distance from the field point to the nearest wall

u_\tau  Friction velocity

\tau_\omega  Wall shear stress
$\nu$  Kinematic viscosity

$\nu_t$  Turbulence kinematic viscosity

$R_e L$  Reynolds number based on length $L$

$R_e \theta$  Reynolds number based on momentum thickness

$Y_M$  Contribution of the fluctuating dilatation

$G_K$  Generation of turbulence kinetic energy due to mean velocity gradient

$G_b$  Generation of turbulence kinetic energy due to buoyancy

$y^+$  Non-dimensional wall normal distance

$u^+$  The dimensionless velocity

$\mu$  molecular dynamic viscosity

$\mu_t$  Turbulence eddy viscosity

$\rho$  Fluid density (kg/m$^3$)

$\sigma_\epsilon, \sigma_k$  Turbulent prandtl number for $k$ and $\epsilon$

$\Omega$  Specific dissipation rate, vorticity vector magnitude

$f_\mu$  Damping function

$\tau_{ij}$  Reynolds stress tensor
## Abbreviations

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>2D</td>
<td>Two-dimensional</td>
</tr>
<tr>
<td>3D</td>
<td>Three-dimensional</td>
</tr>
<tr>
<td>AOA</td>
<td>Angle of Attack</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamic</td>
</tr>
<tr>
<td>DNS</td>
<td>Direct Numerical Simulation</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulation</td>
</tr>
<tr>
<td>DES</td>
<td>Detached Eddy Simulation</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds-Averaged Navier-Stokes</td>
</tr>
<tr>
<td>SST</td>
<td>Shear Stress Transport</td>
</tr>
<tr>
<td>SA</td>
<td>The Spalart-Allmaras</td>
</tr>
<tr>
<td>SKE</td>
<td>Standard k-(\epsilon)</td>
</tr>
<tr>
<td>AOA</td>
<td>Angle of attack</td>
</tr>
<tr>
<td>LEV</td>
<td>Leading edge vortex</td>
</tr>
</tbody>
</table>
ACKNOWLEDGEMENTS

First of all, I want to thank Dr. Zifeng Yang for all of his advice, help, and time spent on my research work. Without these, I cannot finish my research and thesis. I am so happy to work under his direction. He taught me how to search information related to my research work, guided me to finish my research. He was always there and helped me whenever or whatever I needed his help on my learning and research. Under his encouragement and suggestion, I had a lot of confidence to finish my work. I know how to study efficiently, and how to be a good researcher under his influence. I can bring all these skills into my future work and life.

Second, I should thank Professor Fang Chen. He helped me a lot for my project studies and gave me a lot of new ideas. He also helped me to do the experiments. The results from the experiments can support my simulation results very well.

Last but not least, I would like to thank my family and friends from IFI. They gave me a lot of positive support for my study and life. Their support is the reason I have the power to meet the future challenges. Thanks to all of you.
1 Introduction

Flow induced noise is one of the major contributors in the noise generation of various industrial applications. For instance trailing edge noise is an important component of aircraft airframe noise, in particular during landing and approach. Moreover trailing edge noise is also a noise generation mechanism for wind turbine rotor blades and helicopter blades. Due to stronger regulations with respect to noise pollution, the implementation of wind turbines will tend to hamper. To ensure its further development, it is important to reduce this noise mechanism and therefore requires better modeling. Computing trailing edge noise is complex since it is inherently connected with turbulence. The trailing edge noise is caused by the interaction of turbulent structures with the trailing edge.

The effective flight capabilities of insects and birds have long fascinated biological scientists involved in the investigation of low Reynolds number aerodynamic regimes. The ability of insects and birds to hover, take off, and land etc. via flapping propulsion has inspired researcher and aviation engineers to consider equipping aircraft with flapping mechanisms rather than rotating mechanisms as a means of generating thrust and lift forces. Many examples can be found in nature of winged creatures exhibiting excellent aerodynamic characteristics. In many cases, the capabilities of these creatures exceed those demonstrated by man-made aircraft. The application of flapping wings in wind and water energy conversion plants has been actively considered in recent years. Inspired by the flapping wings, another character of the insects and birds also concerned by the researchers is the salient flying.

Night animal have the excellent ability to keep silent while they are flying. In order to hide from predator or get close to the prey without notice, insects and birds have their own way
to achieve it. Owl is the most famous night animal which can give people plenty of new ideas. Owls have evolved special features on their wings to fly silently. Three characteristics of these features have been identified by Graham\(^1\), namely the velvet-like surface, the leading edge serrations, and the trailing edge fringes. The aeroacoustic field is always determined by the flow field over any kind of configuration; thus, the interaction between the flow and the surface defines the noise emitted by wings\(^2\). It was claimed that small vortices which occur at the trailing edge of the wing due to the different velocities of the upper and lower streams are delayed by the fringes. The existence of the flexible air-transmissive fringe results in a smooth surface without sharp edges where noise might be generated\(^3\).

With the rapid expansion of airports and busy air traffic in airports, the dramatic increase of noise pollution influence more and more people living in airport neighborhoods. More recently, with the rapid development of wind energy, the noise emission from wind turbines has become an important issue. How to reduce the aerodynamic noise may be learned from owls\(^1\). Pioneer researches\(^2, 3, 4\) have been conducted to understand the mechanisms underlying silent flight of owls. Klan et al.\(^2\) conducted experimental analysis of the flow field over a wind model with the geometry mimicking the wing of a barn owl. It has been found that applying velvet to the suction side drastically rescues the size of this separation at moderate angles of attack and higher Reynolds numbers.
Figure 1 – The structure of the owl’s wing.

(a) indicates the leading edge serrations; (b) indicates the trailing edge fringes

Based on previous researches, the noise sources come from:

1. Turbulent boundary layer - trailing edge noise
2. Laminar boundary layer - vortex shedding noise
3. Separation-stall noise
4. Trailing edge bluntness - vortex shedding noise
5. Tip vortex shedding noise

For a 2D model, the first, second and fourth items have the greatest influence. For a full scaled wind turbine including 3D effects, it will be necessary to include the tip vortex shedding noise.

Figure 1 shows the structure of the owl’s wing. According to the analysis of the wing, the special character can help to reduce the noise. (a) shows the serrations at the leading edge. Comparing to the smooth leading edge, it can help to reduce the tip vortex shedding noise. (b) shows the fringes at the trailing edge. The fringes can help to improve the flow condition.
around the airfoil and also reduce the noise from trailing edge. This thesis’s idea comes from this character.

Figure 2 - The flow condition around the airfoil

Flexible trailing edge has been widely investigated as a passive/active flow control technique in the study of flapping wings. Lai and Liu\textsuperscript{5} performed the computation of a rigid airfoil in lunging motion with a flexible trailing beam. It showed some potential in increasing thrust generation by a simple addition of a short, flexible, trailing edge to the normal trailing of a NACA0012 airfoil. Hu and Wang used PIV to investigate the flow structure of an oscillating cylinder attached to a flexible fringe. It was concluded that the long, flexible fringe induces a significant vertical velocity in the flow field around the trailing edge, which could press the leading-edge vortices to approach to the surface of flexible fringe and accelerate the dissipation of the leading-edge vortices. Monnier et al.\textsuperscript{7} reported an investigation of the influence of the structural flexibility of sinusoidally pitching airfoils on the pattern of vortices shed into the wake. The results are complemented with analyses using a vortex array model to clarify the relationship between variation in the three wake-vortex pattern parameters and the
net mean thrust acting on the airfoil. Although these studies utilized flexible fringes, they did not take into account the porosity of a real fringe in the trailing edge of an owl’s wing.

All studies mentioned above focus on improving the aerodynamic performance, but not on reducing noise or suppressing the turbulent wake. A recently proposed concept for passive reduction of jet noise involves the suspension of a flexible fringe in the jet mixing region. It has been shown to be quite effective in reducing both screech tones and broadband shock noise\textsuperscript{8}. Whether this technique can be used to reduce noise in the external flow for the application of aircraft or wind turbine is still unclear.

An understanding of the mechanisms owls make use of could lead to a design of modern airfoil configurations with reduced noise emission. However, owls fly at relatively low Reynolds numbers (Re=20,000 – 150,000). In contrast, modern aircraft operated at Reynolds number of some thousand millions and modern wind turbine operate at Reynolds number of millions. Hence, the specialized structures must not be only copied, but have to be understood, and applicable solutions have to be transformed to higher Reynolds number. But there are also practical applications in the daily environment, such as fans, small wind turbines, and cooling systems or car-spoilers and pantographs of trains.

Low Reynolds number airfoil aerodynamics is important for both military and civilian applications. These applications include both military and civilian applications. These applications include propellers, sailplanes, ultra-light man carrying air craft, high altitude vehicles, wind turbines, unmanned aerial vehicles, and micro air vehicles. For all the applications listed above, the combination of small length scale and low flight velocities results in flight regimes with lowing-chord Reynolds number. It is well know that the boundary layer on low Reynolds number airfoil remain laminar at the onset of the pressure recovery unless
artificially tripped. The behaviors of the laminar boundary layers are unable to withstand any significant affect the aerodynamic performances of the airfoil. Since laminar boundary layers are unable to withstand any significant adverse pressure gradient, laminar flow separation is usually found on low Reynolds numbers airfoils. Post separation behavior of laminar boundary layers accounts for the deterioration in the aerodynamic performances of low Reynolds number airfoils. The deterioration is exhibited by an increase in drag and decrease in lift\textsuperscript{26}.

In order to study the underlying mechanism of the flexible trailing edge fringes in reduction of noise emission in the trailing edge flow, a traditional airfoil S833 equipped with flexible fringes on the trailing edge is investigated through numerical simulation and experiments in a wind tunnel. The newly constructed airfoil is modeled and numerically investigated. An incompressible, 2D and viscous flow solver in the Computational Fluid Dynamics (CFD) software FLUENT is utilized to conduct the numerical simulation on the vortex flow feature in the wake of the airfoil. A User Defined Function code was applied to generate the defined motion of flexible fringes. The effects of the length of the fringe, the deformation pattern and penetration rate of the airflow through the fringe are investigated in the parametric study. On the other hand, the airfoil model with the flexible fringe is manufactured for the experimental study. A digital Particle Image Velocimetry (PIV) system is employed to investigate the flow structure in the wake and the deformation of the flexible fringes. The motion of the fringes is extracted from the experimental measurements as the input for CFD simulation. It has been found that the addition of the flexible fringe has a significant effect on the flow characteristics in the vortex wake downstream of the airfoil as well as the aerodynamic performance of the airfoil.
2 Methodology

Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing, e.g. flight tests.

A key advantage of CFD is that it is a very compelling, non-intrusive, virtual modeling technique with powerful visualization capabilities. Moreover, using this, engineers can evaluate the performance of a wide range of HVAC/IAQ system configurations on the computer without having to go the physical site, thereby saving much time and money.

CFD has seen dramatic growth over the last several decades. This technology has widely been applied to various engineering applications such as automobile and aircraft design, weather science, civil engineering, and oceanography. Today, the HVAC/IAQ industry is one such field that has initiated utilizing CFD techniques widely and rigorously in its design26.

2.1 Practical Advantages of Employing CFD

The many reasons CFD is being widely used today are as follows:

1. CFD predicts performance before modifying or installing systems:
Without modifying and/or installing actual systems or a prototype, CFD can predict which design changes are most crucial to enhance performance.

2. CFD provides exact and defringeed information about HVAC design parameters:

The advances in HVAC/IAQ technology require broader and more defringeed information about the flow within an occupied zone, and CFD meets this goal better than any other method, (i.e., theoretical or experimental methods).

3. CFD Saves Cost and Time:

CFD costs much less than experiments because physical modifications are not necessary. (Note that the cost and time for physical changes/modifications increase almost exponentially as the size of the system increases).

4. CFD is Reliable:

The numerical schemes and methods upon which CFD is based are improving rapidly, so CFD results are increasingly reliable. CFD is a dependable tool for design and analyses.

2.2 Fluent Software

Fluent is the world's largest provider of commercial computational fluid dynamics (CFD) software and services. Fluent offers general-purpose CFD software for a wide range of industrial applications, along with highly automated, specifically focused packages.

Fluent is a state-of-the-art computer program for modeling fluid flow and heat transfer in complex geometries. FLUENT provides complete mesh flexibility, including the ability to solve your flow problems using unstructured meshes that can be generated about complex
geometries with relative ease. Supported mesh types include 2D triangular/quadrilateral, 3D tetrahedral/hexahedral/pyramid/wedge/polyhedral, and mixed (hybrid) meshes. FLUENT also allows you to refine or coarsen your grid based on the flow solution.

FLUENT is written in the C computer language and makes full use of the flexibility and power offered by the language. Consequently, true dynamic memory allocation, efficient data structures, and flexible solver control are all possible. In addition, FLUENT uses a client/server architecture, which allows it to run as separate simultaneous processes on client desktop workstations and powerful computer servers. This architecture allows for efficient execution, interactive control, and complete flexibility between different types of machines or operating systems.

---

Figure 3 - Basic Program Structure
2.3 Turbulence Model

In fluid dynamics, turbulence or turbulent flow is a flow regime characterized by chaotic property changes. This includes low momentum diffusion, high momentum convection, and rapid variation of pressure and velocity in space and time\textsuperscript{26}.

Flow in which the kinetic energy dies out due to the action of fluid molecular viscosity is called laminar flow. While there is no theorem relating the non-dimensional Reynolds number (Re) to turbulence, flows at Reynolds numbers larger than 5000 are typically (but not necessarily) turbulent, while those at low Reynolds numbers usually remain laminar. In Poiseuille flow, for example, turbulence can first be sustained if the Reynolds number is larger than a critical value of about 2040; moreover, the turbulence is generally interspersed with laminar flow until a larger Reynolds number of about 4000\textsuperscript{26}.

In turbulent flow, unsteady vortices appear on many scales and interact with each other. Drag due to boundary layer skin friction increases. The structure and location of boundary layer separation often changes, sometimes resulting in a reduction of overall drag. Although laminar-turbulent transition is not governed by Reynolds number, the same transition occurs if the size of the object is gradually increased, or the viscosity of the fluid is decreased, or if the density of the fluid is increased. Nobel Laureate Richard Feynman described turbulence as "the most important unsolved problem of classical physics."\textsuperscript{26}

Turbulence modeling is one of the three key elements in Computational Fluid Dynamics (CFD). Very precise mathematical theories have evolved for the other two key elements: grid generation and algorithm development. By its nature in creating a mathematical model that approximates the physical behavior of turbulent flows- far less precision has been achieved in
turbulence modeling. This is not really a surprising event since our objective has been to approximate an extremely complicated phenomenon.

The most popular three numerical computational methods are:

1. Direct numerical simulation (DNS)

A direct numerical simulation (DNS) is a simulation in computational fluid dynamics in which the Navier–Stokes equations are numerically solved without any turbulence model. This means that the whole range of spatial and temporal scales of the turbulence must be resolved. All the spatial scales of the turbulence must be resolved in the computational mesh, from the smallest dissipative scales (Kolmogorov microscales), up to the integral scale L, associated with the motions containing most of the kinetic energy.

2. Large eddy simulation (LES)

Large eddy simulation (LES) is a mathematical model for turbulence used in computational fluid dynamics. It was initially proposed in 1963 by Joseph Smagorinsky to simulate atmospheric air currents, and many of the issues unique to LES were first explored by Deardorff (1970). LES grew rapidly beginning with its invention in the 1960s and is currently applied in a wide variety of engineering applications, including combustion, acoustics, and simulations of the atmospheric boundary layer. LES operates on the Navier–Stokes equations to reduce the range of length scales of the solution, reducing the computational cost.

The principal operation in large eddy simulation is low-pass filtering. This operation is applied to the Navier–Stokes equations to eliminate small scales of the solution. This reduces the computational cost of the simulation. The governing equations are thus transformed, and
the solution is a filtered velocity field. Which of the "small" length and time scales to eliminate are selected according to turbulence theory and available computational resources.

Large eddy simulation resolves large scales of the flow field solution allowing better fidelity than alternative approaches such as Reynolds-averaged Navier–Stokes (RANS) methods. It also models the smallest (and most expensive) scales of the solution, rather than resolving them as direct numerical simulation (DNS) does. This makes the computational cost for practical engineering systems with complex geometry or flow configurations, such as turbulent jets, pumps, vehicles, and landing gear, attainable using supercomputers. In contrast, direct numerical simulation, which resolves every scale of the solution, is prohibitively expensive for nearly all systems with complex geometry or flow configurations\textsuperscript{22}.

\begin{equation}
\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_j \partial x_j} - \frac{\partial r_{ij}}{\partial x_j} \tag{1-1}
\end{equation}

3. Reynolds-averaged Navier–Stokes equations

The Reynolds-averaged Navier–Stokes equations (or RANS equations) are time-averaged equations of motion for fluid flow. The idea behind the equations is Reynolds decomposition, whereby an instantaneous quantity is decomposed into its time-averaged and fluctuating quantities, an idea first proposed by Osborne Reynolds. The RANS equations are primarily used to describe turbulent flows. These equations can be used with approximations based on knowledge of the properties of flow turbulence to give approximate time-averaged solutions to the Navier–Stokes equations.

\begin{equation}
\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] \tag{1-2}
\end{equation}
2.4 One-equation and two-equation models

As computers have advanced exponentially since the 1960’s, turbulence models based upon the equation for turbulence kinetic energy have become the cornerstone of modern turbulence modeling research. There are two types of eddy viscosity models: one-equation models and two-equation models, the latter being utilized more often. These rarely-used models which both retain Boussinesq approximation based on the turbulence energy equation, are incomplete because they relate the turbulence length scale to some typical flow dimension. Whereas one-equation models are based on an equation for the eddy viscosity, two-equation models automatically provide the turbulence length scale, or its equivalent, and are thus complete with the addition of parameters\textsuperscript{15}.

To complete the closure of the turbulence kinetic energy equation, based on twenty years of experience with the mixing-length, Prandtl (1945) had sufficient confidence that he could generalize established prescriptions for the one equation model as follow:

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = - \tau_{ij} \frac{\partial U_i}{\partial x_j} - C_D \frac{k^{3/2}}{l} + \frac{\partial}{\partial x_j} \left[ (\nu + \nu_T) \frac{\partial k}{\partial x_j} \right]$$

(1-3)

Two-equation models of turbulence have served as the foundation for much of the turbulence model research during past two decades. For example, almost all of the computations done for the 1980-1981 AFOSR-HTTM-Stanford Conference on Complex Turbulent Flows were done using two-equation turbulence models. These models provide not only for the computation of $\kappa$, but also for the turbulence length scale or equivalent\textsuperscript{26}. Consequently, two-equation models are complete, meaning they can be used to predict the properties of a given turbulent flow with no prior knowledge of the turbulence structure \textsuperscript{16,17}. 


2.5 The κ-ε Model

By far, the most popular two-equation model is the κ-ε model. The earliest development efforts based on this model were those of Chou (1945), Davidov (1961), and Harlow and Nakayama (1968). The principal paper however, is that by Jones and Launder (1972) which, in the turbulence modeling community, has nearly reached the status of the Boussinesq and Reynolds papers. That is, the model is so well-known that it is referred to as the Standard κ-ε Model and reference to the Jones-Launder paper is often omitted.\(^{26}\)

The k-ε model has been fringeored specifically for planar shear layers and recirculating flows. This model is the most widely-used and validated turbulence model with applications ranging from industrial to environmental flows, which explains its popularity. It is typically useful for free-shear layer flows with relatively small pressure gradients, as well as in confined flows where the Reynolds shear stresses are most important. It can also be asserted as the simplest turbulence model for which only initial and/or boundary conditions needs to be supplied.\(^{19-21}\)

However, it is more costly in terms of memory than the mixing length model since it requires two extra PDEs. This model would be an inadequate choice for problems such as inlets and compressors as accuracy has been experimentally shown to be reduced for flows containing large adverse pressure gradients. The k-ε model also functions poorly in a variety of important cases such as unconfined flows, curved boundary layers, rotating flows, and flows in non-circular ducts.

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k \mu)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_{\varepsilon} + S_k \tag{1-4}\]

\(\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon \mu)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_1 \rho \frac{\varepsilon}{k} \frac{\partial k}{\partial x_j} \frac{\partial k}{\partial x_j} - C_2 \frac{\varepsilon}{k} \frac{\partial k}{\partial x_j} \frac{\partial k}{\partial x_j} \tag{1-5}\]
\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\sigma_{\varepsilon}} \right) \frac{\partial \varepsilon}{\partial x_i} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_p) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_{\varepsilon} \quad (1-5)
\]

In this paper, in order to calculate the initial value of \( \kappa \) and \( \varepsilon \), the turbulence intensity:

\[
I = \frac{u^l}{u} = 0.16Re^{-0.125} \quad (1-6)
\]

Since the velocity of the incoming flow is 6m/s. Reynolds number is \( Re=4 \times 10^4 \).

\[
k = \frac{3}{2} (UI)^2 \quad (1-7)
\]

\[
\varepsilon = C_{\mu}^{0.75} k^{1.5} L \quad (1-8)
\]

So the initial value of \( \kappa \) is 0.138 and \( \varepsilon \) is 0.02.

**2.6 Initial and Boundary Conditions**

In computational fluid dynamics (CFD), the definition of the initial condition and boundary conditions are the key elements. Almost every CFD problem is defined under the limits of initial and boundary conditions. The initial condition defines the initial value of all the parameters at \( t=t_0 \) such as:

\[
\begin{align*}
    u &= u(x, y, z, t_0) = u_0(x, y, z) \\
    v &= v(x, y, z, t_0) = v_0(x, y, z) \\
    w &= w(x, y, z, t_0) = w_0(x, y, z) \\
    p &= p(x, y, z, t_0) = p_0(x, y, z) \\
    \rho &= \rho(x, y, z, t_0) = \rho_0(x, y, z) \\
    T &= T(x, y, z, t_0) = T_0(x, y, z)
\end{align*}
\]  (1-9)

For the implementation of boundary conditions when constructing a staggered grid, an extra node is added across the physical boundary in order that the nodes just outside the inlet of the system are used to assign the inlet conditions and that the physical boundaries can
coincide with the scalar control volume boundaries. These allow us to introduce the boundary conditions and achieve discretization equations for nodes near the boundary with small modifications. The most common boundary conditions used in computational fluid dynamics are: intake conditions, symmetry conditions, physical boundary conditions, cyclic conditions, pressure conditions, and exit conditions.

The CFD process consists of three primary steps:

Pre-processing: This is the first step of the CFD simulation process which describes the geometry in the best possible manner. One needs to identify the fluid domain of interest, which is then further divided into smaller segments known as mesh generation steps. There are different popular pre-processing software available in the market including Gridgen, CFD-GEOM, ANSYS Meshing, ANSYS ICEM CFD, and TGrid.

Solver: Once the physics problem has been identified, fluid material properties, flow physics models, and boundary conditions are set to solve using a computer. There are popular commercial software available for this including ANSYS FLUENT, ANSYS CFX, Star CCM, CFD++, and OpenFOAM. All these software have their unique capabilities. Using this software, it is possible to solve the governing equations related to flow physics problems.

Post-processing: The next step after getting the results is to analyze the results with different methods like contour plots, vector plots, streamlines, and data curves for appropriate graphical representations and reports. Some of the popular post-processing software includes ANSYS CFD-Post, EnSight, FieldView, ParaView, and Tecplot 36022.
2.7 Experimental apparatus

2.7.1 Experimental rig and test model

The experiments were conducted in a low-speed, open-circuit wind tunnel that has a maximum velocity of 40 m/s and is located in the Department of Mechanical and Materials Engineering in Wright State University. The tunnel has an optically transparent two by two foot test section (610mm×610 mm) in a cross section. The tunnel has a ten-to-one contraction section upstream of the test section with honeycombs and screen structures installed ahead of the contraction section to provide uniform, low-turbulence incoming flow into the test section. The turbulence intensity in the center of the inlet of the test section was found to be about one percent of the incoming flow and was measured by using a hotwire manometer. A S833 airfoil model with a chord length of 100 mm and span length of 300 mm is manufactured. There is a movable part in the trailing edge of the airfoil. The flexible fringe is inserted into the gap between the movable part and the fixed part of the trailing edge, and fixed by these two parts.

2.7.2 Experimental setup

A high-resolution digital Particle Image Velocimetry (PIV) system is used to achieve detailed flow field measurements to quantify the characteristics of the flow around flapping wings. Fig. 4 shows the schematic of the PIV system used in the present study. For the PIV measurements, the flow was seeded with ~ 1 μm water-based droplets by using a fog generator. Illumination was provided by a double-pulsed Nd: YAG laser (New Wave Gemini 120) adjusted on the second harmonic and emitting two pulses of 120 mJ at a wavelength of 532 nm. The laser beam was shaped to a sheet by a set of mirrors with spherical and cylindrical
lenses. The thickness of the laser sheet in the measurement region was about 2 mm. The time interval between two laser pulses is set to 100μs. A high-resolution 14-bit CCD camera (Pixelfly, CookeCorp) was used for PIV image acquisition with the axis of the camera perpendicular to the laser sheet. The CCD camera and the double-pulsed Nd: YAG lasers were connected to a workstation (host computer) via a Digital Delay Generator (Berkeley Nucleonic, Model 565), which controlled the timing of the laser illumination and the image acquisition.

In the present study, instantaneous PIV velocity vectors were obtained by a frame to frame cross-correlation technique involving successive frames of patterns of particle images in an interrogation window of 32×32 pixels. An effective overlap of 50% of the interrogation windows was employed in the PIV image processing. After the instantaneous velocity vectors \((u_i, v_i)\) were determined, the vorticity \((\omega_z)\) can be derived. The distributions of the ensemble-averaged flow quantities such as the mean velocity, turbulence intensity, Reynolds Stress, and turbulence kinetic energy were obtained from a cinema sequence of about 500 frames of the instantaneous PIV measurements.

The measurement uncertainty level for the velocity vectors is estimated to be within 2% and 5% for the turbulent velocity fluctuations, Reynolds stress, and turbulent kinetic energy calculations.
Figure 4 - Schematic of the experimental setup
3 Computational Grid

3.1 Model and mesh

The chord length of the airfoil is 100 mm and the length of the fringe is about 10% of the chord length. In order to capture the vortex distribution and evolution around the airfoil and fringe, the structured grids are used for the boundary layers. The mesh size for the boundary layers around the airfoil and the fringe is 0.2 mm. The fine mesh and 20 layers of the boundary layer can keep the $y+$ value less than 10, which can capture the behavior of the vertical flow feature well. The total number of the cells is $1.1 \times 10^5$. The computational domain is proved to be large enough to satisfy the boundary condition and observe the flow feature in the wake. Also, the mesh should be fine enough for a larger part around the model in order to keep the mesh resolution quality uniform with time. A convergent test was performed to verify the mesh. A C-type grid with the chord wise dimension more than 20 times that of the chord length is created. The domain is split into three regions. The region attached to the model is a mesh of small structure grids which ensure good quality for boundary layer flow. The region far away from the model is a mesh of large structure grids since the structure grids have fewer nodes than unstructured grids. The region between these two parts is a transition mesh of unstructured grids.
3.2 Code validation

In order to verify the code, the code is used to simulate the different airfoil model in different situations. The results also be compared with other experimental data and simulation results. During the conversion test, different mesh methods, boundary conditions and calculation methods are considered.
3.2.1 Convergence criteria

The convergence criteria must be defined before the convergence test. Setting up reasonable convergence criteria is not always straightforward. Although most codes estimate convergence by observing the residual, different codes have their own scaling schemes. Therefore, defining convergence criteria requires highly related experience utilizing a specific code and solving a particular problem. However, even rich experience is not good enough because an exact residual which can guarantee a converged solution cannot be determined by experience alone, but requires further validation. Hence, the independent study of convergence criteria should be applied during the study.

The role of convergence criteria in independent study is as follows: always start with the code’s default residual or default convergence criteria. This default value is verified by the code developer and can usually be used to acquire the converged value. After that, reduce the residual and run the simulation again. Then compare the results between the two simulations. If the results agree with each other, the independent solution is reached; otherwise, reduce the residual again, and do further comparison until the independent solution is reached\textsuperscript{9-13}.

3.2.2 Different mesh method

This section is about conversion test on the mesh method. Four different mesh methods will be tested.
Figure 7 - Four different mesh methods

Figure 7 shows four different ways to mesh the model:

(a). The coarse structure mesh are used to grid the whole domain area. The total number of the mesh is 10,000 cells. The calculation time is 20 minutes under steady state. The Y+ value is 30.

(b). 20 layers of the boundary layer are used to mesh the area close to the airfoil, and the coarse structure mesh are used to grid the rest of the domain. The total number of the mesh is 18,000 cells. The calculation time is 30 minutes under steady state. The Y+ value is 10.

(c). The 20 layers of boundary layer are used to mesh the area close to the airfoil and the coarse structure mesh are used to grid the area far from the airfoil. The unstructured mesh grid
the area between the boundary layer and the coarse mesh in the far area. The total number of the mesh is 189,000 cells. The calculation time is 40 minutes under steady state. The Y+ value is 13.

(d). The structure mesh are used in the whole domain. 20 layers of boundary layer are used to mesh the area close to the airfoil and the coarse structure mesh are used to grid the area far from the airfoil. The unstructured mesh grid the area between the boundary layer and the coarse mesh in the far area. The total number of the mesh is 170,000 cells. The calculation time is 35 minutes under steady state. The Y+ value is 7.

Using a different way to grid the model will significantly affect the simulation results. Comparing the simulation to experimental results:

![Cl vs AOA](image)

Figure 8 - The $C_l$ of the different cases under different AOA
The figure 8 shows that at the low angle of attack the $C_d$ value is almost the same in all four cases. But at the high angle of attack, cases A and B show significant difference with the experimental values. The coarse mesh will cause a low $Y+$ value and this will cause inaccurate results. Both cases C and D get accurate results compared to the experimental data. Both cases C and D have finer mesh and a lower $Y+$ value which will give more accurate results. However, case D has a shorter calculation time. Since case D produces more accurate results in a shorter time frame, it is best to use this method in future work to calculate a lot of different cases with complex conditions.

3.2.3 Different size of domain

According to the experiment results, the incoming flow’s effect will influence the area 7-10 cord length behind the airfoil, 4-6 cord length at top and bottom of the airfoil. In order to identify the suitable size of the domain, four different sizes of the domain were considered. And the simulation results are compared with the experimental data.
Figure 9 - Different size of the domain

(A). The total length of the domain is 60 * chord length. The total number of the mesh is 460,000 cells. The calculation time is 40 minutes under steady state. The Y+ value is 7.

(B). The total length of the domain is 196 * chord length. The total number of the mesh is 240,000 cells. The calculation time is 1 hour and 20 minutes under steady state. The Y+ value is 7.

(C). The total length of the domain is 10 * chord length. The total number of the mesh is 118,000 cells. The calculation time is 30 minutes under steady state. The Y+ value is 13.
The figure 10 shows that all three models have accurate results at a low value of angle of attack. Even at a high value of angle of attack, none of the cases show significant differences compared to the experimental data because they all have low $Y+$ values. Cases A and B still show accurate results at a high value of angle of attack but case B takes a lot more time to calculate the results compared to case A. In future work, the case needs to be calculated over a long time period so the domain size of case A should be used as a model.

3.2.4 Different viscous model

In the Fluent, there are several different viscous models can be chosen to do the calculation.

- Inviscid: Specifies inviscid flow.
- Laminar: Specifies laminar flow.
- **Spalart-Allmaras**: Specifies turbulent flow to be calculated using the Spalart-Allmaras model.

The Spalart-Allmaras model is a relatively simple one-equation model that solves a modeled transport equation for the kinematic eddy (turbulent) viscosity. This embodies a relatively new class of one-equation models in which it is not necessary to calculate a length scale related to the local shear layer thickness. 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. It is also gaining popularity for turbo machinery applications.

- **K-epsilon**: Specifies turbulent flow to be calculated using one of three \( \kappa - \varepsilon \) models.

All three \( \kappa - \varepsilon \) models (standard, RNG, and realizable) have similar forms, with transport equations for \( \kappa \) and \( \varepsilon \). The major differences in the models are as follows:

1. The method of calculating turbulent viscosity
2. The turbulent Prandtl numbers governing the turbulent diffusion of \( \kappa \) and \( \varepsilon \)
3. The generation and destruction terms in the \( \varepsilon \) equation

The transport equations, methods of calculating turbulent viscosity, and model constants are presented separately for each model. The features that are essentially common to all models follow, including turbulent production, generation due to buoyancy, accounting for the effects of compressibility, and modeling heat and mass transfer.

- **K-omega** specifies turbulent flow to be calculated using one of two \( \kappa - \omega \) models.
Both κ-ω models (standard and shear-stress transport (SST)) have similar forms, with transport equations for κ and ω. The major ways in which the SST model differs from the standard model are as follows:

Gradual change from the standard κ-ω model in the inner region of the boundary layer to a high-Reynolds-number version of the κ-ω model in the outer part of the boundary layer

Modified turbulent viscosity formulation to account for the transport effects of the principal turbulent shear stress.

The transport equations, methods of calculating turbulent viscosity, and methods of calculating model constants and other terms are presented separately for each model.

- Reynolds Stress: Specifies turbulent flow to be calculated using the RSM.

The Reynolds stress model (RSM) is the most elaborate turbulence model that FLUENT provides. Abandoning the isotropic eddy-viscosity hypothesis, the RSM closes the Reynolds-averaged Navier-Stokes equations by solving transport equations for the Reynolds stresses, together with an equation for the dissipation rate. This means that five additional transport equations are required in 2D flows and seven additional transport equations must be solved in 3D.

- Detached Eddy Simulation: specifies turbulent flow to be calculated using the DES.

Fluent offers three different models for the detached eddy simulation: the Spalart-Allmaras model, the realizable κ-ω model, and the SST κ-ω model. In the DES approach, the unsteady RANS models are employed in the near-wall regions, while the filtered versions of the same models are used in the regions away from the near-wall. The
LES region is normally associated with the core turbulent region where large turbulence scales play a dominant role. In this region, the DES models recover the respective subgrid models. In the near-wall region, the respective RANS models are recovered.

The application of DES, however, may still require significant CPU resources and therefore, as a general guideline, it is recommended that the conventional turbulence models employing the Reynolds-averaged approach be used for practical calculations. The DES models, often referred to as the hybrid LES/RANS models combine RANS modeling with LES for applications such as high-Re external aerodynamics simulations. In Fluent, the DES model is based on the one-equation Spalart-Allmaras model, the realizable $\kappa$-ω model, and the SST $\kappa$-ω model. The computational costs, when using the DES models is less than LES computational costs, but greater than RANS.

- Large Eddy Simulation (3D only): Specifies turbulent flow to be calculated using the LES model.

Turbulent flows are characterized by eddies with a wide range of length and time scales. The largest eddies are typically comparable in size to the characteristic length of the mean flow. The smallest scales are responsible for the dissipation of turbulence kinetic energy.
In order to choose the correct viscous model, airfoil S833 is used to do the convergence test. Under different angles of attack and the same velocity of incoming flow, the test will keep calculating until the results converge. The drag coefficient and lift coefficient at different angles of attack are recorded and the pressure coefficient is plotted. Comparing all the results with the experimental data obtained from other researches, the comparison shows in figure 12, 13.
<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Area</td>
<td>1m²</td>
</tr>
<tr>
<td>Density</td>
<td>1.225kg/m³</td>
</tr>
<tr>
<td>Velocity</td>
<td>6m/s</td>
</tr>
<tr>
<td>Viscosity</td>
<td>1.7894E-5kg/m-s</td>
</tr>
</tbody>
</table>

**Figure 12 - \( C_d \) vs AOA under different turbulence models and experiments**
Figure 13 - $C_l$ vs AOA under different turbulence models and experiments

Figure 12, 13 show that comparing to the experimental data, most of the models can acquire fairly accurate results, especially at low angles of attack. At high angles of attack some models do not get accurate results because of the separation. Comparing all the results above, the $k$-$\varepsilon$ (realizable $\kappa$-$\varepsilon$ models) is the best model to use for future calculations.

In order to test the realizable $\kappa$-$\varepsilon$ models, this viscous model is used continually to calculate different airfoil models such as S834 and NACA2415. Comparing the simulation results with the experimental data can confirms that $\kappa$-$\varepsilon$ models can give accurate results.
Figure 14 - $C_p$ of the airfoil S834 at AOA of 8°

Figure 15 - $C_p$ of the airfoil S834 at AOA of 12°
Comparing the simulation data with the experimental data by Dan M. Somers (2001), the results can match each other fairly well under different boundary conditions. Although there are some differences between two results at the leading edge area, since the simulation results are under ideal condition, so the difference cannot be avoided. The minority differences can be ignored.

3.2.5 The plunging motion of rigid airfoil

All the confirmations above are under steady state. Since future work which related the moving fringes, so the case under transient state should also be considered.

![Figure 16 – The plunging motion of the airfoil](image)

In this case, the plunging motion of the rigid airfoil shows in figure 16. The mesh around the airfoil show in figure 17.
Figure 17 – The mesh around airfoil

The simulation in this case is under transient state, the airfoil has the variational velocity during the plunging motion. $h_0=0.8\text{m}$
Comparing the distribution of the pressure coefficient at different positions during the fourth time period, the results can match each other fairly well. At this time point, the rigid airfoil is moving down stroke, generating a vortex located on the upper surface left hand side. Due the airfoil is moving down, a vortex is produced near the trailing edge which is moving up.
The figure 19 shows the comparison of the drag coefficient at the third time period. The results can match with the previous research very well. During the plunging motion, the distribution of the drag coefficient likes sine curve.

Comparing the results with previous research which under the steady state, the difference is larger since it’s under more complex conditions. The different mesh method, domain size and time step will cause difference between each other. Since the results still can match each other fairly well, which can verify the accuracy of the code.
4 Results and discussion

In this chapter, several parameters of the flexible fringe will be considered. The results will show which parameter has the most influence on improving the flow condition behind the airfoil.

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Area</td>
<td>1m²</td>
</tr>
<tr>
<td>Density</td>
<td>1.225kg/m³</td>
</tr>
<tr>
<td>Velocity</td>
<td>6m/s</td>
</tr>
<tr>
<td>Viscosity</td>
<td>1.7894E-5kg/m-s</td>
</tr>
<tr>
<td>Angle of Attack</td>
<td>9°</td>
</tr>
</tbody>
</table>

4.1 A comparison of the original model and the flexible fringe model

In order to reduce the intensity of the vortex behind the airfoil, a flexible fringe is added at the end of the airfoil, thus the existence of the flexible fringe will improving the flow condition behind the airfoil.
(a) Without fringe

(b) With fringe

T=1.9s

(c) Without fringe

(d) With fringe

T=3s
Figure 20 - The comparison of the vorticity distribution at different time period

(a) Airfoil without fringe at $T=1.9s$, (b) Airfoil with fringe at $T=1.9s$ (c) Airfoil without fringe at $T=3s$ (d) Airfoil with fringe at $T=3s$, (e) Airfoil without fringe at $T=5s$, (f) Airfoil with fringe at $T=5s$

Figures 20 above show the distribution of the vorticity at different time period. After adding a flexible fringe to the end of the airfoil, the vorticity condition behind the airfoil is significantly improved. Comparing the figure (a) and (b), the separation starts close to the leading edge in the case (a) without fringe. Due to the trailing edge fringe, the total length of the model is extended, so when the vortex is already shed in the case (c) without fringe, the vortex just generated in the case (d). At 5th seconds shown in figure 21 (e) and (f), vortex shedding still occurs for both airfoil. Comparing to the case (f) without fringe, the intensity of the vorticity is weaker in the case (e) with fringe. The vorticity behind the airfoil is one of the sources of noise; therefore the reduction of the vorticity also can help to reduce the noise.
The pressure distribution also changed after the flexible fringe is added.

(a) Without fringe

(b) With fringe

T=1.9s

(c) Without fringe

(d) With fringe

T=3s
Figure 21 - The comparison of the pressure distribution at different time period
(a) Airfoil without fringe at T=1.9s, (b) Airfoil with fringe at T=1.9s (c) Airfoil without fringe at T=3s (d) Airfoil with fringe at T=3s, (e) Airfoil without fringe at T=5s, (f) Airfoil with fringe at T=5s.

Figure 21 above shows the pressure distribution on the surface area. The trend is similar to the vorticity distribution. The lower surface does not show a significant change, but the upper surface does change a lot. The case without the fringe has large negative pressure area and will dissipate behind the airfoil. The case with the fringe only generates small area of the negative pressure area. After the leading edge vortex is generated, it will keep attaching on the upper surface. So the case with the fringe can keep the lift coefficient better than the original model.
Figures 22 show the comparison of pressure coefficient along the airfoil. An existence of the fringe, the airflow condition changes around the airfoil. The separation at the leading edge in the case without fringe causes the pressure drop at the upper surface. The case with the fringe can maintain the pressure even when the leading edge vortex is generated. At the trailing edge, with the existence of the fringe, the total length of the model, at the position of $x/c = 1.0$, the case with fringe does not have significant shedding behind the airfoil. The fringe at the end of the airfoil can further improves the flow condition around the airfoil. It can prevents the separation and also can maintains the lift coefficient.

4.2 The different frequency of the vibration

In this section, the effects of different frequencies of the vibration are considered.

For figure 23 – 33:

(a) $F=2$Hz
(b) F=1Hz

(c) F=0.5Hz

(d) F=0.25Hz

Table 3 Reference value of calculation

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Area</td>
<td>$1\text{m}^2$</td>
</tr>
<tr>
<td>Density</td>
<td>$1.225\text{kg/m}^3$</td>
</tr>
<tr>
<td>Velocity</td>
<td>$6\text{m/s}$</td>
</tr>
<tr>
<td>Viscosity</td>
<td>$1.7894\times10^{-5}\text{kg/m-s}$</td>
</tr>
<tr>
<td>Angle of Attack</td>
<td>$9^\circ$</td>
</tr>
<tr>
<td>Vibration Amplitude</td>
<td>$0.003\text{m}$</td>
</tr>
</tbody>
</table>

4.2.1 Comparison at $t=2T$

(a) F=2Hz

(b) F=1Hz
Figure 23 - The comparison of the vorticity distribution at t=2T

Figure 23 shows the vorticity distribution at the second time period. Each cases shows significantly different on vorticity distribution. The figure (a) shows the case with the high frequency equal to 2Hz, the upper surface boundary layer is very thin and the wake is narrow. The figure (b) shows the same trend like the case (a) but the intensity is stronger. The figure (c) shows a totally different situation than case (a) and (b). A vortex is shedding from the airfoil and a new vortex is just started. The figure (d) shows different result with the others, the separation start close to the leading edge and has several vorticity behind the airfoil. When comparing the four different cases, cases (a) and (b) with high vibration frequency do not show the strong vorticity so the flow condition is better. The reason is because the fringe rapid wave can prevent the generation of the vorticity. In contrast, cases (c) and (d) have low vibration frequency, so the fringe cannot improve the flow condition efficiently.
Figure 24 - The comparison of the pressure distribution at t=2T

Figure 24 shows the pressure distribution in different cases at the second time period. The flow condition changes drastically behind the airfoil. Cases (a) and (b) have higher frequencies, therefore, at the second time period, there is no leading edge vortex generated on the upper surface. Consequently, in the pressure contour, no obvious negative pressure area is shown. Cases (c) and (d) show different conditions between each other. Since the vortex has already detached from the airfoil and a new vortex has begun to form, a negative pressure can be found behind the airfoil where the vortex exists. As a result of the different frequencies, the vorticity also shows different intensity and shapes in different cases.
Figure 25 - The comparison of the $C_p$ distribution at different cases at $t=2T$

Figure 25 shows the $C_p$ in different cases. Basically, each case shows the same trend except for the frequency that equals 0.5Hz. In figure 23, a leading edge vortex is generated on the upper surface at the position $x/c=0.5$, therefore in the figure 25, the pressure coefficient drops at same the position. Although all four different cases have same trend, when compared, their values are different. The case with the frequency equal to 0.25Hz has a lower pressure coefficient on the upper surface compared to the other cases. In all four cases, it appears that higher frequency maintain higher pressure coefficient. Therefore the higher frequency can not only reduce the vorticity, but can also maintain a higher lift force.
4.2.2 Comparison at t=3T

Figure 26 - The comparison of the vorticity distribution at t=3T

Figure 26 shows the vorticity distribution at the third time period. The trend is similar to the second time period. For case (a), the vortex becomes stronger but still does not shed from the airfoil. For case (b), a leading edge vortex starts at the leading edge, attaches on the upper surface of the airfoil, and a new leading edge vortex begins to form. A vortex already sheds from the airfoil and moves downstream. For case (c), the separation starts at the leading edge, and a strong vorticity sheds from the airfoil and moves downstream. For case (d), complete
separation occurs at the leading edge, and the strong vorticity sheds from the airfoil and moves downstream.

![Pressure Distribution Diagrams](attachment:image)

(a) F=2Hz  
(b) F=1Hz  
(c) F=0.5Hz  
(d) F=0.25Hz

Figure 27 - The comparison of the pressure distribution at t=3T

Figure 27 shows the pressure distribution in different cases at the third time period. After one time period, case (a), with the high vibration frequency does not show a huge different when compared to the second time period. There is no apparent negative pressure area around the airfoil. For case (b), because of the existence of the leading edge vortex, two negative pressure areas can be found at the upper surface of the airfoil. For case (c) and (d), it is easier to observe negative pressure areas behind the airfoil since the vorticity already shed from the
airfoil. When comparing the results from different cases, the flow condition behind the airfoil is observed to experience a significant change as the fringe’s vibration decreases. In the same time period, the case with higher frequency does not show strong vortex and separation.

![Graph showing Cp distribution](image)

**Figure 28** - The comparison of the Cp distribution at different cases at t=3T

Figure 28 shows the Cp in different cases. The cases have the same trend except for when the frequency is equal to 0.5Hz. In figure 27, a vorticity is just shedding from the trailing edge of the airfoil, so in the figure there is a large pressure drop along the airfoil at the position x/c=0.8-0.9. From this figure, the cases with higher frequencies can maintain higher pressure coefficients, so they have a higher lift force.
4.2.3 Comparison at t=3s

Figure 29 - The comparison of the vorticity distribution at t=3s

Figure 29 shows the vorticity distribution at the third seconds. Since different cases have different vibration frequencies at the same time, they are not in the same time period. All the cases produce vorticity behind the airfoil. For case (a), two leading edge vortexes can be seen in the figure. One has just formed and another still attach on the upper surface of the airfoil. A vortex can also be generated at the trailing edge. For case (b), the strong separation can be found at the upper surface of the airfoil. Also, a vortex is about to shed from the trailing edge. For cases
(c) and (d), with low vibration frequencies, strong vorticity can be observed behind the airfoil, and the separation on the upper surface begins close to the leading edge.

![Pressure distribution images](image)

(a) F=2Hz  
(b) F=1Hz  
(c) F=0.5Hz  
(d) F=0.25Hz

Figure 30 - The comparison of the pressure distribution at t=3s

Figure 30 shows the pressure distribution in different cases at third seconds. For cases (a) and (b), the existence of the leading edge vortex causes a negative pressure on the upper surface. Both cases have similar pressure distribution, however, case (b) with lower vibration frequency has a larger area of negative pressure. On the other hand, the cases (c) and (d)
already have the vorticity shedding from the airfoil and move to the downstream. The difference between these two cases at this time is that in case (c), the vorticity is still strong in the wake. In the case (d), the vorticity still can be seen, but the intensity is lower than the other. In comparing the different cases, a different vibration frequency has been found to have a significant influence on the flow condition behind the airfoil.

![Diagram](image)

**Figure 31 - The comparison of the $C_p$ distribution at different cases at t=3s**

Figure 31 shows the distribution of $C_p$ in different cases. At the same time step, the different cases show a different trend for $C_p$. At this time step, because of the separation, the pressure drops in different cases at different positions. In case (b), a strong leading edge vortex forms at position of x/c=0.5. Thus in this figure, a large pressure drop occurs at x/c=0.5 along the upper surface of the airfoil. As shown in figure 31, the vibration frequency decrease causing the point of separation moves upstream. Also the size of the leading edge vortex decreases due to the high frequency vibration. Comparing the four distributions of the pressure coefficient, case (a), with high vibration frequency, can maintain the higher lift force.
4.3 The different wave motion of the fringe

In real-life applications, the wave motion of the fringe depends on the velocity of the incoming flow and the geometry of the airfoil. In this investigation, two common vibration modes were isolated and modeled for in-depth study. These vibration modes are shown in figure 36:

![Figure 36](image)

(a) Second order vibration mode

(b) First order vibration mode

Figure 32 - Two different vibration mode of the flexible fringe

For figure 32-38

(a) Second order vibration mode

(b) First order vibration mode
Figure 33 - The comparison of the vorticity distribution at t=3s

Figure 33 shows the vorticity distribution at three seconds in two different cases. The position of the separation is almost the same in the two different cases. When the leading edge vortex is generated at the upper surface, the size of the vortex is almost the same. But it will become different then it arrives at the flexible trailing edge fringe. But in case (a) with second order vibration, the curved fringe breaks down the vortex and dissipates it when it moves downstream. In the case with first order vibration, the vortex still maintains at the upper surface of the airfoil.

Figure 34 - The comparisons of the pressure distribution at t=3s
Figure 34 shows the pressure distribution at 3rd seconds for both cases. The pressure shows a very large difference between the two cases. For the case with second order vibration mode, because the vortex is broken down by the fringe, the distribution of pressure does not show a significant change behind the airfoil. In the case with the second vibration mode, because of the leading edge vortex, there is negative pressure area exists at the upper surface close to the trailing edge.

![Image](image_url)

(a) Second order vibration mode  
(b) First order vibration mode

Figure 35 - The comparisons of the vorticity distribution at t=4s

Figure 35 shows the vorticity distribution at 4th seconds for the two different cases. The situation at this time is same as before. The new vortex is broken down around the fringe because of the second order vibration. Therefore the vorticity is smaller than the other case. In the case with first order vibration, the vortex is just about to detach from the vortex. Comparing the vorticity of the two cases, case (a) can efficiently dissipates the vortex.
Figure 36 shows the pressure distribution at 4 seconds for the two cases. In case (a) which has a second order vibration, the pressure distribution does not have significant change behind the airfoil. The value of the negative pressure is higher than case (b). In case (b), because of the strong vorticity, two negative pressure areas appear behind the airfoil. Comparing the two cases, the second vibration order of the fringe can help improve the flow condition and decrease the drag force.
Figure 37 - The comparisons of the $C_p$ distribution at different cases at $t=3s$

Figure 37 shows $C_p$ at the same time step in two different cases. Since the two cases have the same structure under same flow condition, the distribution of the pressure coefficient along the airfoil is almost the same. This means the effect of the vibration mode only influence the flow condition in the wake behind the airfoil. The difference only happens near the trailing edge. The motion of the second order vibration can prevent the vortex shedding from the fringe and reduce the vorticity. So comparing to the second order vibration mode, the pressure coefficient in the case of first order vibration drops a bit at the trailing edge. Since the difference between the two cases is small, both cases can maintain the same lift force.
The comparison of turbulence kinetic energy in different cases.

Figure 38 shows the distribution of the turbulence kinetic energy in different cases. The influence area and intensity of the turbulence kinetic energy shows huge different. The mean and maximum value of the intensity is smaller in case (a) which has second order vibration.
fringe. The different vibration mode can improve the flow condition and dissipate the turbulence. The high turbulence kinetic energy concentrate at the fringe in case (b) which has the first order vibration mode, and the value is higher than case (a). So the second order can also help to reduce the turbulence kinetic energy behind the airfoil.
5 Conclusion

The main goal of this thesis was to study the configuration of trailing edge fringes that capable of reducing the trailing edge noise emitting from an airfoil. The answer to this question and the conclusion is found to be positive in the sense that the potential of the hybrid methods was demonstrated. Nevertheless some issues concerning the simulations still remain. To reach this conclusion a variety of numerical experiments and validations were conducted. In this chapter, the conclusions based on the test cases are addressed.

After verifying the code with the former research, the results from the simulation were accurate enough comparing to the experimental data. So it can be considered as fairly accurate results. Comparing the different cases which include different parameters, the results can show the effect of the trailing edge fringe and the key characters which can cause the significant influence on the flow condition around the airfoil.

1. The existence of the flexible trailing edge fringe extends the total length of the model and improves the flow condition around the airfoil. It has a significant impact on the aerodynamic performance since it influences the separation, consequently the transition onset, and thus, the reattachment. Compared with the model without the fringe, the new model can prevent the separation at the leading edge. The separation flow can reattach to the airfoil instead of forming a large separation vortex that eventually detaches from the vortex and moves downstream. The noise can be reduced because of the reduction of the vortex

2. The length of the fringe will cause the different results on the flow condition. The results show that the position of the vortex structure detaching from a leading edge
vortex is different because of the different length of the fringe. The vortex will move further downstream and breaks down and dissipates. The longer fringe will also increase the lift coefficient of the airfoil. In this thesis, the length of the fringe in four different cases does not have significant difference, so the value varies from different result is similar. The initiation, growth and detachment of the LEV were largely unaffected. The importance of the length still needs to be investigated.

3. The frequency of the vibration of the flexible fringe has a significant effect on the flow condition around the airfoil. Because of the high frequency of vibration, the leading edge vortex reattach to the upper surface sooner than the other cases. The size of the vortex is also smaller. The rapid wave of the fringe can breaks down the vortex and dissipates it. So it can also prevent the vortex detaching from the vortex. When the vortex move downstream, the rapid flap of the fringe can help to reduce the vorticity and decrease the time period of the vorticity. The high frequency can also inducing the point of separation moving upstream. Combining all the results in different cases at different time period, the high frequency of the vibration is crucial on reducing the noise.

4. The different vibration model of the flexible fringe also affects the flow condition. The results show that position of the separation on the upper surface is almost the same in both cases. The size of leading edge vortex is same like each other. When the leading edge vortex reach the fringe, in the case which has the first order vibration mode, the vortex keeps the size and the vortex detaches from the vortex and moves downstream. In contrast, in the case of the second order vibration, the flexible shape of fringe will disturb the vortex. The fringe breaks it down and dissipates it when the
leading edge vortex arrives. Although the vortex is still generated in the case with second vibration mode, but the vorticity is lower than the other. Both cases can maintain the same lift coefficient but the second order vibration can significantly reduce the vorticity which may results in low noise emission.
6 Future work

This paper discussed several parameters of the flexible fringe. Comparing the results, the frequency of the vibration and the different vibration modes play an important role in reducing the noise. Meanwhile, many parameters need to be investigated in the future.

1. The amplitude of the vibration.
2. The permeability of the fringe.
3. The material of the fringe.
4. The other different vibration mode.

While the 2D model gives a good approximation of the vortex activity, a 3D model will provide more accurate results. Furthermore, in the 3D model, different fringe geometries may be tested to optimize the fringe design.

Figure 39 - 3D model with rectangular fringe
Currently, two types of 3D model have been investigated. The results show the same trend as the 2D model. In the future, more researches need to be done and the related experiment will help to support the results.
7 Reference


FLUENT Introduction.


