Numerical Simulations and Studies on Round Jet Impingement, Twin Jet Impingement and Helicopter Rotor Blades Ground Effect

Han Gao
Washington University in St. Louis

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

Part of the Engineering Commons

Recommended Citation
https://openscholarship.wustl.edu/eng_etds/322

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 Simulations and Studies on Round Jet Impingement, Twin Jet Impingement and Helicopter Rotor Blades Ground Effect
by
Han Gao

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

May 2018
St. Louis, Missouri
# Table of Contents

Table of Contents ..................................................................................................................... ii  
List of Figures ........................................................................................................................... iii  
List of Tables .............................................................................................................................. viii  
Nomenclature ............................................................................................................................ ix  
Acknowledgments ..................................................................................................................... xiv  
ABSTRACT OF THE THESIS .................................................................................................... xvi  

Chapter 1: Numerical Study of Round Jet Impingement in Proximity of Ground and a Water Surface .............................................................................................................................. 1  
1.1 Introduction .......................................................................................................................... 1  
1.2.1 Physical Model ............................................................................................................... 3  
1.2.2 Computational Method .................................................................................................. 5  
1.2.3 Mesh Independence and Time Increment Independence ............................................. 8  
1.3 Comparison of Lift Loss on Baffle Plate due to Air Jet Impingement on Solid Ground and Water Surface .................................................................................................................. 10  
1.3.1 Comparison of Lift Loss at the Same Jet Velocity but at Different Heights .............. 11  
1.3.2 Comparison of Lift Change for Impingement on Solid Ground vs. Water Surface Cases at Same Height ................................................................................................................. 23  
1.4 Conclusions ....................................................................................................................... 33  

Chapter 2: CFD Simulations of Two Rectangular Water Jets Impinging on a Water Pool ........ 35  
2.1 Introduction .......................................................................................................................... 35  
2.2 Turbulence Transport Equations used in Various Turbulence Models ......................... 36  
2.3 Test Case and Results ......................................................................................................... 37  
2.3.1 The Twin Jet Experiment ........................................................................................... 38  
2.3.2 Validation of Various RANS Turbulence Models .................................................... 40  
2.3.3 Combining Process of Twin Jets (a 3-D Phenomenon) ............................................. 47  
2.4 Conclusions ....................................................................................................................... 54  

Chapter 3: Numerical Study of Helicopter ............................................................................ 55  
3.1 Introduction ........................................................................................................................ 55
3.2 Geometry, Mesh, Computational Method and Mesh Independence Study .................. 58
3.21 Geometry and Mesh .......................................................................................... 58
3.22 Computational Method ...................................................................................... 59
3.23 Mesh Independence ........................................................................................... 62
3.3 Validation and Results Analysis ......................................................................... 62
3.31 Load of Rotor Blades ....................................................................................... 62
3.32 Wake of Rotor Blades ....................................................................................... 67
3.4 Conclusions ......................................................................................................... 69

Chapter 4: Conclusion ............................................................................................. 70

References .............................................................................................................. 71

Appendix .................................................................................................................. 74

Curriculum Vita ....................................................................................................... 76
List of Figures

Figure 1 Model Geometry and Mesh in the computational domain (Jet impinges a solid wall).... 5
Figure 2 Model geometry and mesh in the computational domain (Jet impinges a water surface). .......................................................... 5
Figure 3 Mesh independence study .............................................................................................................. 9
Figure 4 Time increment independence study ............................................................................................... 10
Figure 5 Variation of non-dimensional lift loss with non-dimensional height at constant jet velocity for solid ground case and the water surface case. ......................................................... 11
Figure 6 Pressure distribution on the baffle plate and pressure contours for jet velocity of 25m/s and non-dimensional height of 0.15 for solid ground case .......................................................... 12
Figure 7 Pressure distribution on the baffle plate and pressure contours for jet velocity of 25m/s and non-dimensional height of 0.30 for solid ground case .......................................................... 12
Figure 8 Pressure distribution on the baffle plate and pressure contours for jet velocity of 25m/s and non-dimensional height of 0.45 for solid ground case .......................................................... 13
Figure 9 Pressure distribution on the baffle plate and pressure contours for jet velocity of 25m/s and non-dimensional height of 0.60 for solid ground case .......................................................... 13
Figure 10 Pressure distribution on the baffle plate and pressure contours for jet velocity of 25m/s and non-dimensional height of 0.75 for solid ground case .......................................................... 13
Figure 11 Streamlines and velocity Contours between the baffle plate and the solid ground for jet velocity of 25m/s and non-dimensional height of 0.15 in the solid ground case .............................................. 14
Figure 12 Streamlines and velocity Contours between the baffle plate and the solid ground for jet velocity of 25m/s and non-dimensional height of 0.30 in the solid ground case .............................................. 14
Figure 13 Streamlines and velocity Contours between the baffle plate and the solid ground for jet velocity of 25m/s and non-dimensional height of 0.45 ................................................................. 15
Figure 14 Streamlines and velocity Contours between the baffle plate and the solid ground for jet velocity of 25m/s and non-dimensional height of 0.60 in the solid ground case .............................................. 15
Figure 15 Streamlines and velocity Contours between the baffle plate and the solid ground for jet velocity of 25m/s and non-dimensional height of 0.75 in the solid ground case .............................................. 15
Figure 16 Air volume fraction contours, streamlines and velocity contours for jet velocity of 25m/s and non-dimensional height of 0.15 in water surface case .......................................................... 17
Figure 17 Air volume fraction contours, streamlines and velocity contours for jet velocity of 25m/s and non-dimensional height of 0.30 in water surface case .......................................................... 18
Figure 18 Air volume fraction contours, streamlines and velocity contours for jet velocity of 25m/s and non-dimensional height of 0.45 in water surface case .......................................................... 19
Figure 19 Air volume fraction contours, streamlines and velocity contours for jet velocity of 25m/s and non-dimensional height of 0.60 in water surface case .......................................................... 19
Figure 20 Air volume fraction contours, streamlines and velocity contours for jet velocity of 25m/s and non-dimensional height of 0.75 in water surface case .......................................................... 20
Figure 21 Pressure distribution on the baffle plate for jet velocity of 25 m/s and non-dimensional height of 0.15 for water surface case. .............................................................. 21
Figure 22 Pressure distribution on the baffle plate for jet velocity of 25 m/s and non-dimensional height of 0.30 for water surface case. .............................................................. 21
Figure 23 Pressure distribution on the baffle plate for jet velocity of 25 m/s and non-dimensional height of 0.45 for water surface case. .............................................................. 22
Figure 24 Pressure distribution on the baffle plate for jet velocity of 25 m/s and non-dimensional height of 0.60 for water surface case .............................................................. 22
Figure 25 Pressure distribution on the baffle plate for jet velocity of 25 m/s and non-dimensional height of 0.75 for water surface case .............................................................. 23
Figure 26 Non-dimensional lift loss at different jet velocities and constant non-dimensional height of 0.15 for water surface cases and solid ground cases .................................................. 24
Figure 27 Pressure contours, velocity contour and streamlines for non-dimensional height of 0.15, and jet velocity of 30 m/s for solid ground case .................................................. 25
Figure 28 Pressure contours, velocity contour and streamlines for non-dimensional height of 0.15, and jet velocity of 35 m/s for solid ground case. .................................................. 26
Figure 29 Pressure contours, velocity contour and streamlines for non-dimensional height of 0.15, and jet velocity of 40 m/s for solid ground case. .................................................. 27
Figure 30 Pressure contours, velocity contour and streamlines for non-dimensional height of 0.15, and jet velocity of 45 m/s for solid ground case. .................................................. 28
Figure 31 Pressure contours, velocity contour and streamlines for non-dimensional height of 0.15, and jet velocity of 50 m/s for solid ground case. .................................................. 29
Figure 32 Pressure distribution on the baffle plate at different jet velocities with non-dimensional height of 0.15 for solid ground cases .................................................. 29
Figure 33 Air volume fraction contours, streamlines and pressure contours for 0.15 non-dimensional height and jet velocity of 30 m/s for water surface case. .................................. 30
Figure 34 Air volume fraction contours, streamlines and pressure contours for 0.15 non-dimensional height and jet velocity of 35 m/s for water surface case. .................................. 31
Figure 35 Air volume fraction contours, streamlines and pressure contours for 0.15 non-dimensional height and jet velocity of 40 m/s for water surface case. .................................. 31
Figure 36 Air volume fraction contours, streamlines and pressure contours for 0.15 non-dimensional height and jet velocity of 45 m/s for water surface case. .................................. 32
Figure 37 Air volume fraction contours, streamlines and pressure contours for 0.15 non-dimensional height and jet velocity of 50 m/s for water surface case. .................................. 33
Figure 38 Pressure distributions on the baffle plate at different jet velocities for non-dimensional height of 0.15 for water surface cases .................................................. 33
Figure 39 Twin Jet Water Facility. ............................................................................. 38
Figure 40 Three regions in two jet interactions (a = 5.8 mm, s = 17.8 mm). ................. 39
Figure 41 Computational domain corresponding to twin-jet experiment and structured mesh... 39
Figure 42 Demonstration of orthogonality and quality of mesh ........................................40
Figure 43 Location of computed velocity profile in the pipe........................................40
Figure 44 Computed velocity profiles in the pipe using various turbulence models........41
Figure 45 Location of Symmetry Plane 1 ........................................................................42
Figure 46 Velocity contours, pressure contours and streamlines in symmetry plane 1 from k-ε model ........................................................................................................................43
Figure 47 Velocity contours, pressure contours and streamlines in symmetry plane 1 from SA model ........................................................................................................................43
Figure 48 Velocity contours, pressure contours and streamlines in symmetry plane 1 from SST k-ω model ................................................................................................................44
Figure 49 Streamlines in flow field obtained with k-ε model (left) and LES (right). ..........45
Figure 50 Locations of four lines used for computations of velocity profiles .................45
Figure 51 Velocity profiles at z/a=2 (left) and z/a=3.4 (right) ..................................... 45
Figure 52 Velocity Profile at z/a=7.2 (left) and z/a=12.3 (right) ................................ 46
Figure 53 Location of center line .......................................................................................46
Figure 54 Stream wise velocity along the center line .....................................................47
Figure 55 Turbulence viscosity along center line ...........................................................47
Figure 56 Location of Symmetry Plane 2 ..........................................................................48
Figure 57 Stream-wise velocity contours in symmetry plane 2 .....................................49
Figure 58 Stream-wise velocity iso-surfaces in the converging region of the two jets ...51
Figure 59 Planes parallel to symmetry plane1 ................................................................ 52
Figure 60 Planes parallel to symmetry plane1 off by 0.01m ........................................ 52
Figure 61 Planes parallel to symmetry plane1 off by 0.02m ........................................ 53
Figure 62 Planes parallel to Symmetry Plane1 off by 0.03m ....................................... 53
Figure 63 Planes parallel to symmetry plane1 off by 0.04m ........................................ 53
Figure 64 Model Geometry ............................................................................................58
Figure 65 Model Mesh ...................................................................................................59
Figure 66 Stationary and Moving Reference Frames ....................................................60
Figure 67 Position of the slices ......................................................................................63
Figure 68 Pressure distribution (r/R=0.50) .................................................................63
Figure 69 Pressure distribution (r/R=0.68) .................................................................63
Figure 70 Pressure distribution (r/R=0.80) .................................................................64
Figure 71 Pressure distribution (r/R=0.89) .................................................................64
Figure 72 Pressure distribution (r/R=0.96) .................................................................64
Figure 73 Ground effect a thrust increase at constant angular velocity ......................65
Figure 74 Pressure distribution (r/R=0.50) .................................................................66
Figure 75 Pressure distribution (r/R=0.68) .................................................................66
Figure 76 Pressure distribution (r/R=0.80) .................................................................67
Figure 77 Pressure distribution (r/R=0.89) ................................................................ 67
Figure 78 Pressure distribution (r/R=0.96) ................................................................. 67
Figure 79 symmetry plane .............................................................................................. 68
Figure 80 velocity magnitude contour and streamline from rotor blades (h=0.05) ........ 68
Figure 81 velocity magnitude contour and streamline from rotor blades (h=0.5) .......... 68
Figure 82 velocity magnitude contour and streamline from rotor blades (h=1.0) .......... 68
Figure 83 velocity magnitude contour and streamline from rotor blades (h=2.0) ........ 69
List of Tables

Table 1: Boundary Conditions .......................................................................................................................... 4
Table 2: Comparison between the steady and transient simulation for the solid ground case ...... 9
Table 3: CP and MP Locations .......................................................................................................................... 41
Table 4: Boundary Conditions .......................................................................................................................... 58
Table 5: Assessment of grid independence of the computed solution ......................................................... 62
Nomenclature

\( X \) = the axial coordinate
\( R \) = the radial coordinate
\( R \) = the universal gas constant
\( v_x \) = the axial velocity
\( v_r \) = the radial velocity
\( v_z \) = the swirl velocity
\( S_m \) = the mass added to the continuous phase from the second phase
\( p \) = the static pressure
\( p_{op} \) = the operating pressure
\( p_0 \) = the total pressure
\( F_x \) = the model-dependent source term in the axial direction
\( F_r \) = the model-dependent source term in the radial direction
\( \mu \) = the molecular viscosity
\( M_w \) = the molecular weight
\( M \) = Mach Number
\( \gamma \) = the ratio of specific heats
\( k \) = the turbulence kinetic energy
\( \varepsilon \) = the turbulence dissipation rate
\( G_k \) = the generation of turbulence kinetic energy due to mean velocity gradients
\( G_b \) = the generation of turbulence kinetic energy due to buoyance
\( Y_m = \) the contribution of the fluctuating dilatation to the overall dissipation rate

\( C_{1*} = \) constant (1.44)

\( C_{2*} = \) constant (1.92)

\( \sigma_k = \) the turbulent Prandtl number for \( k \) (1.0)

\( \sigma_\varepsilon = \) the turbulent Prandtl number for \( \varepsilon \) (1.3)

\( C_\mu = \) constant (0.09)

\( S_k = \) the source term for \( k \)

\( S_\varepsilon = \) the source term for \( \varepsilon \)

\( \mu_t = \) the turbulent (or eddy) viscosity

\( n + 1 = \) index for current time step

\( n = \) index for previous time step

\( \alpha_q^{n+1} = \) cell value of volume fraction at time step \( n+1 \)

\( \alpha_q^n = \) cell value of volume fraction at time step \( n \)

\( \alpha_{q,f}^{n+1} = \) face value of the \( q \)th volume fraction at time step \( n+1 \)

\( U_f^{n+1} = \) volume flux through the face at time step \( n+1 \)

\( V = \) cell volume

\( m_{pq} = \) the mass transfer from phase \( p \) to phase \( q \)

\( m_{qp} = \) the mass transfer from phase \( q \) to phase \( p \)

\( S_{\alpha q} = \) the source term

\( \sigma = \) surface tension coefficient

\( p_1, p_2 = \) the pressures in the two fluids on either side of the interface
\( R_1, R_2 \) = the two radiuses in orthogonal directions
\( \kappa \) = the curvature
\( F_{\text{vol}} \) = the force at the inter surface of two face
\( t \) = time
\( \rho \) = density
\( \tilde{\nu} \) = modified turbulent viscosity
\( x \) = coordinate
\( u \) = velocity
\( G_{\nu} \) = generation of turbulent viscosity
\( Y_{\nu} \) = destruction of turbulent viscosity
\( \sigma_{\nu} \) = constant
\( C_{b_2} \) = constant
\( S_{\nu} \) = user defined source term
\( k \) = turbulent kinetic energy
\( \varepsilon \) = turbulent dissipation rate
\( G_k \) = generation of turbulent kinetic energy due to mean velocity gradient
\( G_b \) = generation of turbulence kinetic energy due to buoyancy
\( Y_m \) = contribution of the fluctuating dilatation to the overall dissipation rate
\( C_{1\varepsilon} \) = constant
\( C_{2\varepsilon} \) = constant
\[ \sigma_k = \text{turbulent Prandtl number in } k\text{-equation} \]
\[ \sigma_\varepsilon = \text{turbulent Prandtl number in } \varepsilon\text{-equation} \]
\[ C_\mu = \text{constant} \]
\[ S_k = \text{source term for } k \]
\[ S_\varepsilon = \text{source term for } \varepsilon \]
\[ \omega = \text{specific dissipation rate} \]
\[ G_\omega = \text{generation of ratio of } \omega \]
\[ \Gamma_k = \text{the effective diffusivity of } k \]
\[ \Gamma_\omega = \text{the effective diffusivity of } \omega \]
\[ Y_k = \text{the dissipation of } k \text{ due to turbulence} \]
\[ Y_\omega = \text{the dissipation of } \omega \text{ due to turbulence} \]
\[ S_\omega = \text{source term for } \omega \]
\[ \mu_t = \text{turbulent (or eddy) viscosity} \]
\[ H = \text{distance from rotor blade to ground} \]
\[ r = \text{radial coordinate} \]
\[ R = \text{radius of rotor} \]
\[ h = \frac{H}{R}, \text{non-dimensional height} \]
\[ AR = \text{aspect ratio} \]
\[ \alpha = \text{angle of attack} \]
\[ \overline{v_t} = \text{relative translating velocity} \]
\[ \Omega = \text{angular velocity of rotor blade} \]
\[ \overline{\omega} = \text{angular velocity of moving reference frames} \]
\( \vec{r} = \) position vector

\( \vec{r}_0 = \) a position vector in example

\( \vec{v}_r = \) relative velocity

\( \vec{v} = \) absolute velocity

\( \vec{u}_r = \) velocity of the moving frame relative to the inertial reference frame

\( \rho = \) density

\( p = \) pressure

\( \ddot{\alpha} = \) angular acceleration in moving reference frame

\( \ddot{a} = \) translation acceleration in moving reference frame

\( E_r = \) relative internal energy

\( H_r = \) relative total enthalpy

\( \tau_r = \) viscous stress

\( T = \) temperature

\( C_p = \) pressure coefficient

\( C_T = \) thrust coefficient

\( c = \) chord length
Acknowledgments

I would like to take this opportunity to express my appreciation to those who helped me with various aspects during my research in computation fluid dynamics lab at Washington University in St. Louis.

First of all, I want to thank Professor Ramesh Agarwal for his guidance and patience throughout my research. His talent, both industrial and academic rich CFD experience inspired and encouraged me to explore the academic world in fluid dynamics.

I would like to thank Dr. Quanzhong Liu, Dr. Qiulin Qu and Dr. Xu Han for all the effort they put in to help me from the very beginning to the very end. The brilliant minds of Dr. Qu and Dr. Liu combined with their experience in fluid dynamics and aerodynamics have contributed immensely to this thesis. I would also like to thank Dr. Xu Han for exchanging ideas during this research; his knowledge in computer science and fluid dynamics helped me enormously.

I would also like to thank my another committee member, Professor Peters, for taking the time to teach me about helicopter theory, provide materials for my research, read this thesis and attend the defense.

Han Gao

Washington University in St. Louis

May 2018
Dedication

I would like to dedicate this thesis to my father (Yaoming Su), my mother (Xiaohong Han) and my sister (Hui Su) for their unconditional support.

I will never succeed without their guidance, influence and encouragement.
ABSTRACT OF THE THESIS

Numerical Simulations and Studies on Round Jet Impingement, Twin Jet Impingement and Helicopter Rotor Blades Ground Effect

by

Han Gao

Master of Science in Mechanical Engineering
Washington University in St. Louis, 2018

Research Advisor: Professor Ramesh K. Agarwal

In this thesis, the projects can be divided into three parts. The first part focuses on the numerical simulation and analysis of single round air jet impingement on a water surface. Currently the studies on round jet impingement on water surface are very few. The goal of this research is to study the difference in aerodynamics and flow physics of an air jet impinging on a water surface and a ground surface at different subsonic Mach numbers and heights above the ground. The conventional air jet impingement on a ground surface is compared to the air jet impingement on a water surface in the range of Mach number from 0 to 0.3. The incompressible Reynolds-Averaged Navier-Stokes equations with k-ε turbulence model are solved using the commercial CFD solver ANSYS FLUENT. For jet exit close to the flat ground surface and water surface, some important differences in flow phenomenon are observed due to that ground being solid and the water being a fluid. The second part the thesis focuses on twin-jet impingement on a water surface in a water tank. The complex flow phenomenon due to the interaction of two jets and fountain formation are successfully captured by the simulation including the relatively low-pressure region, flow curvature and backflow vortex. The twin jet after impingement on each other will combine as one single jet. The third part of the thesis focuses on the numerical simulation of a 3-D helicopter rotor blade in ground effect in transonic regime. This study consists of two parts. The first part involves the calculation of blade pressure distribution, lift on blade sections and rotor thrust performance. Along the span wise direction near the tip, local
Mach number becomes close to 1 and shock is generated which is captured numerically. The numerical results are validated against the experimental data for the hovering rotor blade. When rotor blade is close to the ground, the change in blade pressure distribution is studied. The second part mainly involves the properties of the rotor wake when rotor blade is close to the ground. It includes the determination of trajectory and strength of the wake.
Chapter 1: Numerical Study of Round Air Jet Impingement in Proximity of Ground and a Water Surface

The goal of this chapter is to study the difference in aerodynamics and flow physics of an air jet impinging on a solid ground and a water surface at low subsonic Mach numbers with different heights above the ground. Single air jet impingement on solid ground and water surface with baffle plate around the jet exit is simulated by employing the incompressible Reynolds-Averaged Navier-Stokes equations with k-ε turbulence model, and Volume of Fluid (VOF) tracking method for air/water interface. The simulation results show that at the same (small) impingement height, there is increase in lift force on the baffle plate due to impingement on the water surface which is contrary to the lift loss experienced in the presence of solid ground. Furthermore, in case of impingement on water surface, the simulation results at the same velocity but at different heights above the water surface show that the lift increase at lower height tends to become smaller as the height above the water surface increases finally showing a loss in lift as the jet exit moves from small distance to very large distance.

1.1 Introduction

Aircrafts like Joint Strike Fighter (JSF) F-35 equipped with thrust vector control engine (TVC Engine) can take off and land vertically by changing the direction of the jet from the TVC Engine. However, the whole process of taking off and landing occurs only on a rigid surface such as deck of an aircraft carrier or the runway of an airport, etc. Some fixed-wing aircrafts in the future may be required to have the ability to take off from and land on water surface
vertically. For short-take-off and vertical-landing (STOVL) aircrafts which can vertically land or hover above the water surface, there is hardly any research on understanding the change in lift on the aircraft undersurface due to an air jet from the engine impinging on the water surface.

Lift loss on a surface from which a round air jet exits and impinges on the ground (the ground effect on lift loss) has been widely studied in the past fifty years. Wyatt [1] of RAE Bedford (today DRA Bedford) was among one the first who used a baffle plate to represent the undersurface of an aircraft to simplify the model and in 1962 conducted a series of experiments on studying the lift loss on the baffle plate due to a single air jet with baffle plate around its exit impinging on the ground at different heights above the ground; he found that closer was the ground surface from the jet, higher was the lift loss. For studying the aerodynamic property of a fighter close to the ground, Kuhn and Eshleman [2] studied the ground effect and introduced the definition of non-dimensional lift loss which is the ratio of total suction force on the baffle plate to the jet propulsion force. To determine the performance of STOVL aircraft hovering above the ground, Frank and Douglas [3] studied the suction when STOVL aircraft nozzle is close to ground. Trying to explain the difference among all the characteristic curves from different researchers for the same model, Ing [4] conducted an experimental and computational analysis on ground-effect lift loss due to different geometries of the baffle plate edge. Agarwal and Bower [5] also obtained high Reynolds number numerical solutions of the Reynolds-averaged Navier-Stokes equations in conjunction with a two-equation (k-ε) turbulence model to obtain the STOVL ground effect flow fields.

Most researchers studying the air jet impingement on a water surface in past decades focused on the shape of the water surface and flow field of the liquid phase. Turkdogan [6] conducted experiments on studying the air jet impingement on water to determine the shape of water
surface cavity and explored the relationship between the velocity of the air jet and the depth of the cavity. To improve the efficiency in metallurgical and mining industry, Nguyen and Evans [7] used Computational Fluid Dynamics to study the water flow field due to the air jet impingement on the water surface. When studying the process of mixing in liquid metal, Mikaed and Ersson [8] formulated a mathematical model to calculate the shapes of the cavities due to air jet impingement on the water surface and provided contours of the liquid phase velocity. Recently, Qu and Agarwal [9] numerically investigated the water-landing characteristics of a regional airplane with high wing and high tail by solving the Reynolds-Averaged Navier-Stokes equations with realizable k-ε turbulence model using the finite volume method, and they used the volume-of-fluid method to capture the water-air interface.

Although numerous numerical simulation studies have been conducted to study the flow field of jets from a STOVL aircraft in ground effect due to an aircraft carrier deck or a runway, there are hardly any studies on an air jet in ground effect due to the proximity of a water surface. There is lack of understanding how the water surface will change the lift when jet is closer to the water surface. This paper, for the first time in the literature, presents the detailed comparison between the flow fields and resulting lift changes due to a single round air jet impingement on a solid ground and a water surface. The mechanism for the difference in the two flow fields is analyzed and explained. A simple model is developed to explain the difference in lift change due to jet impingement on a solid ground and a water surface.

1.2.1 Physical Model

Figure 1 shows the model geometry and mesh inside the computational domain in which the baffle plate, the air jet, the solid ground and side wall are shown schematically. The jet direction is downward from the jet exit. The computational domain and the jet are axisymmetric. Figure 2
shows an identical geometry but the jet impinges on water surface. In Figure 2, the water phase is shown in a specific marked area. The hydrostatic pressure field is specified by using a User Defined Function in Fluent. The boundary conditions which correspond to different parts are given in Table 1 below.

<table>
<thead>
<tr>
<th>Boundary Condition Type</th>
<th>Boundary</th>
<th>Jet exit</th>
<th>Baffle Plate</th>
<th>Axis</th>
<th>Atmosphere</th>
<th>Solid Ground</th>
<th>Side Wall</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Jet exit</td>
<td>Velocity Inlet</td>
<td>No Sip Wall</td>
<td>Axis</td>
<td>Pressure Outlet</td>
<td>No Slip Wall</td>
<td>No Slip Wall</td>
</tr>
</tbody>
</table>

The height H is defined as the distance between the lower surface of the baffle plate and the solid ground surface or the water surface. The length of the simplified fuselage undersurface is equal to the difference between the radius of the baffle plate R and the radius of pipe r which is $R - r$ in Figure 1 and Figure 2. The non-dimensional height is defined as the ratio $H/(R - r)$. The lift change on the baffle plate $\Delta L$ is defined as the difference between the lift when jet exit is close to the ground or water surface and the lift when jet is at a very large (infinite) distance from the ground/water surface. The non-dimensional change in lift is defined as the ratio of the lift change (here lift loss is treated as positive based on the past convention) $\Delta L$ to $\rho V^2 S$, where S is the area of cross-section of the pipe. Thus, the non-dimensional lift loss is denoted as $\Delta L / \rho V^2 S$. One of the goals of the chapter is to compare the non-dimensional lift change on the baffle plate due to solid ground vs. water surface at the same height. The radius of the jet is 25mm, the radius of the baffle plate is 250mm, and the pipe length is 500mm (which ensures the flow to be fully developed at the jet exit); these are kept the same in both cases. The calculation starts upstream of the jet exit in the pipe in order to eliminate the influence of the inlet boundary condition on the flow field outside the pipe exit.
Figure 1 Model Geometry and Mesh in the computational domain (Jet impinges a solid wall).

Figure 2 Model geometry and mesh in the computational domain (Jet impinges a water surface).

1.2.2 Computational Method

The governing equations used for the solution of the flow field are listed below:
For 2D axisymmetric geometries, the continuity equation is given by

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x} (\rho v_x) + \frac{\partial}{\partial r} (\rho v_r) + \frac{\rho v_r}{r} = S_m$$  \hspace{1cm} (1)

For 2D axisymmetric geometries, the axial and radial momentum conservation equations are given by

$$\frac{\partial}{\partial t} (\rho v_x) + \frac{1}{r} \frac{\partial}{\partial x} (r \rho v_x v_x) + \frac{1}{r} \frac{\partial}{\partial r} (r \rho v_r v_x)$$

$$= - \frac{\partial p}{\partial x} + \frac{1}{r} \frac{\partial}{\partial x} \left[ \mu \left( 2 \frac{\partial v_x}{\partial x} - \frac{2}{3} (\nabla \cdot \vec{v}) \right) \right] + \frac{1}{r} \frac{\partial}{\partial r} \left[ \mu \left( \frac{\partial v_x}{\partial r} + \frac{\partial v_r}{\partial x} \right) \right] + F_x$$  \hspace{1cm} (2)

and

$$\frac{\partial}{\partial t} (\rho v_r) + \frac{1}{r} \frac{\partial}{\partial x} (r \rho v_x v_r) + \frac{1}{r} \frac{\partial}{\partial r} (r \rho v_r v_r)$$

$$= F_r - \frac{\partial p}{\partial r} + \frac{1}{r} \frac{\partial}{\partial x} \left[ \mu \left( \frac{\partial v_r}{\partial x} + \frac{\partial v_x}{\partial r} \right) \right] + \frac{1}{r} \frac{\partial}{\partial r} \left[ \mu \left( 2 \frac{\partial v_r}{\partial r} - \frac{2}{3} (\nabla \cdot \vec{v}) \right) \right] - 2 \frac{v_r}{r^2} + \frac{2 \mu}{3 r} (\nabla \cdot \vec{v}) + \frac{\rho v_z^2}{r}$$  \hspace{1cm} (3)

where

$$\nabla \cdot \vec{v} = \frac{\partial v_x}{\partial x} + \frac{\partial v_r}{\partial r} + \frac{v_r}{r}$$  \hspace{1cm} (4)

The energy conservation equation takes the form

$$\frac{p_0}{p} = \left( 1 + \frac{\gamma - 1}{2} M^2 \right)^{\frac{\gamma}{\gamma - 1}}$$  \hspace{1cm} (5)

The equation of state is given by
\[ \rho = \frac{p_{op} + p}{R \frac{T}{M_w}} \]  

(6)

The two-equation k-\( \varepsilon \) turbulence models [10] is given by the equations:

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_m + S_k
\]

(7)

\[
\frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_i} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} \left( G_k + C_3 \varepsilon G_b \right) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon
\]

(8)

\[ \mu_t = \rho C_{\mu} \frac{k^2}{\varepsilon} \]

(9)

In addition, the Volume of Fluid (VOF) method [11] is employed to capture the interface between air and water. In the VOF method, the fluid volume fraction \( \alpha \) is defined in each mesh cell; if the \( q \)th fluid’s volume fraction in the cell is denoted as \( \alpha_q \), the following three conditions are defined:

\[ \alpha_q = 0: \text{the cell is empty of the } q^{\text{th}} \text{ fluid} \]  

(10)

\[ \alpha_q = 1: \text{the cell is full of the } q^{\text{th}} \text{ fluid} \]  

(11)

For \( 0 < \alpha_q < 1 \), the cell contains the interface between the \( q^{\text{th}} \) fluid and other fluids.  

(12)

The continuum surface force (CSF) model proposed by Brackbill et al. [12] is used to calculate the addition of surface tension in the VOF calculations as a source term in the momentum equation. Let \( n \) be the surface normal. The equations of the CSF model are as follows:

\[ p_2 - p_1 = \sigma \left( \frac{1}{R_1} + \frac{1}{R_2} \right) \]

(13)

\[ n = \nabla \alpha_q \]

(14)

\[ \kappa = \nabla \cdot \hat{n} \]

(15)

where

\[ \hat{n} = \frac{n}{|n|} \]

(16)
There are only two phases \((i\) and \(j)\); the force on the interface is given by:

\[
F_{vol} = \sigma_{ij} \frac{\rho \kappa_i \nabla \alpha_i}{2 (\rho_i + \rho_j)}
\]

In this chapter, the double precision solver in ANSYS FLUENT 16.0 is employed in all calculations. The gradient discretization is chosen as cell based least square. The momentum equations are discretized using a second order upwind scheme. The turbulent kinetic energy equation is discretized by a first order upwind scheme for stability reasons. The transient terms are discretized as first order implicit. The pressure interpolation scheme is the PRESTO scheme which can deal with rapid changes in pressure due to jet impingement on solid ground and water surface. The Pressure-Velocity coupling is used to calculate the transient flow cases. The reconstruction scheme is adopted to calculate the interface. The numerical scheme is selected as SIMPLE.

1.2.3 Mesh Independence and Time Increment Independence of Solution

First, the mesh independence of the solution for both the solid ground case and the water surface case are checked by computing the non-dimensional lift loss at non-dimensional height 0.15 with jet velocity of 25 meter per second using various mesh sizes. The smallest number of cells for the solid ground case is 200,000. As can be seen from Figure 3, when the number of cells reaches 550,000, the result does not vary for number of cells larger than 550 thousands. However, the flow field details in solid ground case cannot be fully captured when using even 550,000 cells; therefore 700,000 cells were finally used to obtain completely mesh independent solution capturing all features of the flow field. The computational study in water surface case begins
with 350,000 cells and finally 900,000 cells are used to obtain completely mesh independent solutions.

![Figure 3 Mesh independence study.](image)

In the solid ground cases, no unsteady behavior in the flow field was observed for larger times with the use of transient simulation solver and the flow achieved the steady state in an expected manner. The steady simulation solver was used to compare and verify the results from the transient solver. Table 2 clearly shows that in the solid ground case with 25m/s jet velocity and 0.15 non-dimensional height above the ground, it is better to use steady simulation solver to save time and cost.

**Table 2: Comparison between the steady and transient simulation for the solid ground case**

<table>
<thead>
<tr>
<th></th>
<th>Non-dimensional Lift Loss</th>
<th>Error in Continuity Equation</th>
<th>Number of Iteration Step to Reach Steady State</th>
<th>CPU Utilization (for same parallel core numbers = 3)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steady Simulation Solver</td>
<td>0.9123</td>
<td>&lt;1e-6</td>
<td>8000~10000</td>
<td>≈50%</td>
</tr>
<tr>
<td>Transient Simulation Solver</td>
<td>0.9138</td>
<td>&lt;1e-6</td>
<td>&gt;60000</td>
<td>≈84%</td>
</tr>
</tbody>
</table>
However in the water surface cases, it was found that only with the use of transient simulation solver, several transient flow phenomena could be captured and that the steady solver was not suitable. Therefore time increment independence study was conducted for the transient solver. The convergence monitor was set on the baffle plate to observe the independence of the total force on it with time increment. It was found that the majority of the water surface cases could be considered to achieve steady state when the force exerted on the baffle plate did not change within 0.5 % over 2000 time steps. Figure 4 shows the time increment independence in water surface case (0.15 non-dimensional height and jet velocity of 25m/s). For time increment which was larger than 1e-4 second, each time increment could not reach the convergent result within 50 iterations. When time increment was gradually decreased to less than 1e-7 second, the average of 2000 time increments didn’t vary with the length of time increment.

![Figure 4 Time increment independence study.](image)

### 1.3 Comparison of Lift Loss on Baffle Plate due to Air Jet Impingement on Solid Ground and Water Surface

In this section, the lift loss on the baffle plate is compared for different jet velocities and jet heights above the solid ground and the water surface.
1.3.1 Comparison of Lift Loss at the Same Jet Velocity but at Different Heights

The nearly incompressible jet velocity (25 m/s, Mach number = 0.0735) is kept unchanged at different non-dimensional heights $H/(R-r) (= 0.15, 0.30, 0.45, 0.60, 0.75)$.

Figure 5 shows that, at small non-dimensional height, the solid ground effect and the water surface effect behave in an opposite manner; the solid ground case shows a lift loss (suction) on the baffle plate while the water surface shows an increase in the lift. However, at larger non-dimensional height, the solid ground case and the water surface show the same effect of lift loss (suction) on the baffle plate. When the non-dimensional height goes to infinity (becomes very large), both effects due to solid ground and water surface approach each other asymptotically.

![Graph showing variation of non-dimensional lift loss with non-dimensional height](image)

Figure 5 Variation of non-dimensional lift loss with non-dimensional height at constant jet velocity for solid ground case and the water surface case.

Figure 6, Figure 7, Figure 8, Figure 9 and Figure 10 show the pressure distributions and pressure contours at same jet velocity but at different non-dimensional heights in solid ground cases. In each figure, the highest-pressure region is located in the impingement area at the center of the jet on the ground surface (stagnation point of the flow field). And between the lower surface of the baffle plate and the ground surface, the pressure contours show a low-pressure region which
results in the situation that the pressure distribution on the lower surface of the baffle plate is always lower than that on its upper surface. Thus, the total normal force exerted on the baffle plate is always downward in the direction of gravity. However as the distance of the baffle plate increases from the ground surface, the difference of pressure distribution between the upper and lower surfaces of the baffle plate continues to decrease as expected.

![Figure 6](image.png)  
**Figure 6** Pressure distribution on the baffle plate and pressure contours for jet velocity of 25m/s and non-dimensional height of 0.15 for solid ground case.

![Figure 7](image.png)  
**Figure 7** Pressure distribution on the baffle plate and pressure contours for jet velocity of 25m/s and non-dimensional height of 0.30 for solid ground case.
Figure 8 Pressure distribution on the baffle plate and pressure contours for jet velocity of 25m/s and non-dimensional height of 0.45 for solid ground case.

Figure 9 Pressure distribution on the baffle plate and pressure contours for jet velocity of 25m/s and non-dimensional height of 0.60 for solid ground case.

Figure 10 Pressure distribution on the baffle plate and pressure contours for jet velocity of 25m/s and non-dimensional height of 0.75 for solid ground case.
Figure 11, Figure 12, Figure 13, Figure 14 and Figure 15 show the streamlines and velocity contours at same jet velocity but at different non-dimensional heights in solid ground cases. After impinging on the ground surface, the jet spreads over the ground surface in all directions. Because of the existence of the friction between the moving flow and the ground surface, a boundary layer is generated. After the flow moves a certain distance from the stagnation point towards far field, the boundary layer separation occurs due to adverse pressure gradient. Then, a steady vortex is generated between the ground surface and the lower surface of the baffle plate. This steady vortex is the main cause of the suction on the baffle plate due to ground effect.

Figure 11 Streamlines and velocity Contours between the baffle plate and the solid ground for jet velocity of 25m/s and non-dimensional height of 0.15 in the solid ground case

Figure 12 Streamlines and velocity Contours between the baffle plate and the solid ground for jet velocity of 25m/s and non-dimensional height of 0.30 in the solid ground case
Figure 13 Streamlines and velocity Contours between the baffle plate and the solid ground for jet velocity of 25m/s and non-dimensional height of 0.45.

Figure 14 Streamlines and velocity Contours between the baffle plate and the solid ground for jet velocity of 25m/s and non-dimensional height of 0.60 in the solid ground case.

Figure 15 Streamlines and velocity Contours between the baffle plate and the solid ground for jet velocity of 25m/s and non-dimensional height of 0.75 in the solid ground case.
Next focusing on the water surface effect, the same jet velocity of 25m/s is adopted. At this jet velocity, the same series of non-dimensional heights as in the solid ground case above are calculated to explore the change in lift with different non-dimensional heights.

Figure 16, Figure 17, Figure 18, Figure 19 and Figure 20 show the contours of the volume fraction of air, contours of velocity and streamlines at different non-dimensional heights in the water surface impingement cases. The most distinctive difference between the water surface case and the solid ground case is that water surface is deformed by the air jet impingement resulting in a cavity-like shape as shown in these figures. And this geometric difference is the main cause of other flow field differences between solid ground and water surface impingement. Figure 16, Figure 17, Figure 18, Figure 19 and Figure 20 streamlines show that after the impingement between air jet and water surface, the air flow cannot separate instantaneously from water. Instead, it moves upwards around the curved sides of the cavity due to the mass conservation. The air trajectory results in a second impingement caused by the upward air flow on the lower surface of baffle plate. But this only happens when the baffle plate is close to the water surface as shown in Figure 16 and Figure 17. The steady vortex between the water surface and the lower surface of baffle plate can be seen as in the solid ground cases. But there is another steady vortex between the main downward jet flow and the upward jet when the baffle plate is close to water surface which can only be seen in Figure 16, Figure 17, Figure 18. In Figure 18, the vortex between downward jet and upward jet and the vortex between the water surface and the lower surface of baffle plate are about to merge. In Figure 19 and Figure 20, there is only one vortex.

Figure 16, Figure 17, Figure 18, Figure 19 and Figure 20 show the water volume fraction contours at same jet velocity of 25m/s but at different non-dimensional heights from the water surface. They clearly show that the deformation of water surface caused by air jet impingement
is becoming smaller when the baffle plate (or jet exit) is away from the water surface. It also means that the water surface is flatter which makes the water surface more like the solid ground surface which is absolutely flat. It can be extrapolated that when the height goes to infinity (becomes very large), the difference between the water surface cases and the solid ground cases becomes negligible.

![Diagram](image)

**Figure 16** Air volume fraction contours, streamlines and velocity contours for jet velocity of 25m/s and non-dimensional height of 0.15 in water surface case.
Figure 17 Air volume fraction contours, streamlines and velocity contours for jet velocity of 25m/s and non-dimensional height of 0.30 in water surface case.
Figure 18 Air volume fraction contours, streamlines and velocity contours for jet velocity of 25m/s and non-dimensional height of 0.45 in water surface case.

Figure 19 Air volume fraction contours, streamlines and velocity contours for jet velocity of 25m/s and non-dimensional height of 0.60 in water surface case.
At small dimensionless heights, the direct consequence of the second impingement on the lower surface of the baffle plate is that a relatively higher pressure region is created on the lower surface of baffle plate. The steady vortex between the water surface and the lower surface of baffle plate still contributes to the relatively lower pressure area on the lower surface of the baffle plate as in the solid ground cases. Figure 21, Figure 22, Figure 23, Figure 24 and Figure 25 show the static pressure distribution of the lower surface of the baffle plate at same jet velocity of 25m/s at different non-dimensional heights. Figure 21 and Figure 22 clearly show that the high-pressure region coincides with the second impingement location at small non-dimensional
heights. To explore the water surface effect on the baffle plate, pressure distribution on the both surfaces of the baffle plate is investigated. Integrating the pressure distribution, the effect of water surface at each non-dimensional height is determined to see whether it is a lift loss (suction) or a lift increase (anti-suction).

Figure 21 Pressure distribution on the baffle plate for jet velocity of 25m/s and non-dimensional height of 0.15 for water surface case.

Figure 22 Pressure distribution on the baffle plate for jet velocity of 25m/s and non-dimensional height of 0.30 for water surface case.
Figure 23 Pressure distribution on the baffle plate for jet velocity of 25m/s and non-dimensional height of 0.45 for water surface case.

Figure 24 Pressure distribution on the baffle plate for jet velocity of 25m/s and non-dimensional height of 0.60 for water surface case.
Figure 25 Pressure distribution on the baffle plate for jet velocity of 25m/s and non-dimensional height of 0.75 for water surface case

In conclusion, the flow field for the water surface cases is very different from the ground surface cases. In the solid ground case, the lift decreases monotonically as the height increases as shown in Figure 5. In the water surface case, Figure 5 shows that when the baffle plate is close to the water surface, the lift decreases quickly and the suction effect emerges. Also, as the baffle plate becomes farther from the water surface, larger distance is required for the upward flow to reach the lower surface of the baffle plate and the second impingement is being weakened and even disappears and the steady vortex becomes relatively more dominant. The difference between the solid ground cases and the water surface cases disappears as the non-dimensional height goes to infinity (becomes very large) as expected.

1.3.2 Comparison of Lift Change for Impingement on Solid Ground vs. Water Surface Cases at Same Height

The non-dimensional height 0.15 is kept unchanged for different jet velocities of 30 m/s, 35 m/s, 40 m/s, 45 m/s and 50m/s. The reason for choosing a small non-dimensional height is that at this height, the difference between the solid ground and water surface cases is large. Since the non-
dimensional height is small and fixed, the results show that for all cases with different jet velocities, there is lift loss in the solid ground cases and there is lift increase in water surface cases.

The result of lift loss at different jet velocities is shown in Figure 26. The lift loss in water surface cases is about -30%. The minus sign indicates that physically, there is a lift increase in water surface cases when the baffle plate is very close to the water surface. Comparing the results with the solid ground cases, the water surface cases contribute to lift increase, whereas the solid ground cases contribute to lift loss or the suction on the baffle plate.

![Graph of Non-Dimensional Lift Loss vs Jet Velocity](image)

**Figure 26** Non-dimensional lift loss at different jet velocities and constant non-dimensional height of 0.15 for water surface cases and solid ground cases

Figure 27, Figure 28, Figure 29, Figure 30 and Figure 31 show the pressure contours and velocity contours with streamlines at same non-dimensional height of 0.15 but different jet velocities in solid ground cases. The flow structure does not change much with jet velocity. The boundary layer separation is still seen in each figure. The steady vortex is captured in each figure, and the low-pressure region coincides with the center of the vortex.
Figure 27 Pressure contours, velocity contour and streamlines for non-dimensional height of 0.15, and jet velocity of 30 m/s for solid ground case.
Figure 28 Pressure contours, velocity contour and streamlines for non-dimensional height of 0.15, and jet velocity of 35m/s for solid ground case.
Figure 29 Pressure contours, velocity contour and streamlines for non-dimensional height of 0.15, and jet velocity of 40m/s for solid ground case.
Figure 30 Pressure contours, velocity contour and streamlines for non-dimensional height of 0.15, and jet velocity of 45m/s for solid ground case.
Figure 31 Pressure contours, velocity contour and streamlines for non-dimensional height of 0.15, and jet velocity of 50m/s for solid ground case.

The pressure distributions on the baffle plate in solid ground cases shown above are given in Figure 32. They clearly show that the lowest pressure in each curve decreases with increasing jet velocity which means that the suction increases as the jet velocity increases.

Figure 32 Pressure distribution on the baffle plate at different jet velocities with non-dimensional height of 0.15 for solid ground cases

Figure 33, Figure 34, Figure 35, Figure 36 and Figure 37 show the air volume contours, pressure contours and streamlines at the same non-dimensional height of 0.15 with different velocities for the water surface cases. The second impingement is still found in each figure which coincides with high-pressure on the lower surface of the baffle plate. The steady vortex is also captured in each figure.
Figure 33 Air volume fraction contours, streamlines and pressure contours for 0.15 non-dimensional height and jet velocity of 30m/s for water surface case.
Figure 34 Air volume fraction contours, streamlines and pressure contours for 0.15 non-dimensional height and jet velocity of 35m/s for water surface case.

Figure 35 Air volume fraction contours, streamlines and pressure contours for 0.15 non-dimensional height and jet velocity of 40m/s for water surface case.
Figure 36 Air volume fraction contours, streamlines and pressure contours for 0.15 non-dimensional height and jet velocity of 45m/s for water surface case.
Figure 37 Air volume fraction contours, streamlines and pressure contours for 0.15 non-dimensional height and jet velocity of 50m/s for water surface case.

The pressure distributions of the baffle plate in water surface cases described above are shown in Figure 38. They clearly show that the high-pressure in each curve increases with increasing jet velocity which means that the effect of second impingement becomes greater when the jet velocity increases.

Figure 38 Pressure distributions on the baffle plate at different jet velocities for non-dimensional height of 0.15 for water surface cases

1.4 Conclusions

The mechanism of lift loss on baffle plate due to jet impingement on a solid ground and a water surface is quite different; the air jet impingement on the water surface creates a cavity and the upward flow around the cavity occurs which can impinge back on the baffle plate depending on
the jet height above the water surface. As a result, the lift on the baffle plate increases as the jet height above the water surface increases while in case of solid ground the lift continues to decrease as the jet height above the ground increases. In summary, at small non-dimensional height, the water surface effect is to increase the lift on the baffle plate while the solid ground creates a loss in lift.

There is only one main steady vortex in the flow field between the solid ground and the baffle plate at all ground heights, which is responsible for lower pressure region and suction on the baffle plate. However, in case of water surface, in addition to a steady vortex which is responsible for lower pressure region, there is also a second impingement of jet flow on the lower surface of the baffle plate which is responsible for generating a high pressure region near the baffle plate.

For both the solid ground cases and the water surface cases, as the non-dimensional height increases the water surface effect becomes closer to solid ground effect. When the non-dimensional height becomes very large, the difference between the two effects becomes negligible.
Chapter 2: CFD Simulations of Two Rectangular Water Jets Impinging on a Water Pool

CFD simulations of two water jets emanating from rectangular pipes of large aspect ratio and impinging on a water pool are performed using the commercial CFD software ANSYS Fluent. Reynolds-Averaged Navier-Stokes (RANS) equations with a number of turbulence models, namely the SA, k-ε and SST k-ω are solved. The computations are compared with the experimental data to assess the accuracy of various turbulence models. It is shown that majority of the details of the flow field during the initial phase of jets interaction and intermediate phase of jets merging are best captured by k-ε model but the final phase of combining of two jets into a single jet are best captured by the SST k-ω model due to its shear stress transport correction. The process of evolution of interaction of two jets from the initial phase to final phase is described in detail.

2.1 Introduction

Even today, the accurate prediction of turbulent flows using RANS equations with eddy-viscosity models remains a challenging but very important task for industrial class of flows. Although Direct Numerical Simulation (DNS) can provide very accurate prediction of turbulent flows, it currently requires computational meshes with extremely large number of cells to solve the flows at high Reynolds numbers, which can easily exceed the capability of most advanced computers in the world. Large-Eddy Simulation (LES) is a relatively less compute intensive way
to predict turbulent flows. However, the use of LES to capture flow field details in near wall also requires large meshes to resolve the flow features at high Reynolds numbers.

Currently, the most widely used turbulence models in industry can be divided into two categories. The first is the one-equation models like Spalart-Allmaras (SA) model [13] and Nut-92 model [14]. The second is the two-equation model like the Chien k-ε model [15], the Menter Shear Stress Transport (SST k-ω) model [16] and the Wilcox k-ω model [17] among others. None of the models has universal applicability; they all have limitations and are successful only in a limited number of applications. Which model is going to work best for a very complex flow field is very difficult to predict a priori. The subject of this chapter is one such application where it is very difficult to choose a model and predict its success a priori. The goal of this paper is to consider three most widely used turbulence models – SA, k-ε and SST k-ω model and evaluate their performance for the computation of twin-jet impingement on a water surface. The flow field of twin jet case is quite complex but fortunately excellent experimental data is available for validation of turbulence models.

2.2 Turbulence Transport Equations used in Various Turbulence Models

The transport variable used in the one-equation Spalart-Allmaras [13] $\tilde{v}$ is identical to the turbulent kinematic viscosity except in the near-wall region. The transport equation for the modified turbulent kinematic viscosity $\tilde{v}$ is given by:

$$\frac{\partial (\rho \tilde{v})}{\partial t} + \frac{\partial (\rho \tilde{v} u_i)}{\partial x_i} = G_v + \frac{1}{\sigma_v} \left[ \frac{\partial}{\partial x_j} \left\{ (\mu + \rho \tilde{v}) \frac{\partial \tilde{v}}{\partial x_j} \right\} + C_{b_2} \rho (\frac{\partial \tilde{v}}{\partial x_j})^2 \right] - Y_v + S_v \quad (18)$$

The turbulence kinetic energy, $k$, and its rate of dissipation, $\epsilon$, are obtained from the following transport equations in k-ε model [15]:
\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left[ (\mu + \frac{\mu_t}{\sigma_k}) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_m + S_k \tag{19}
\]

And

\[
\frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_i} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left[ (\mu + \frac{\mu_t}{\sigma_\varepsilon}) \frac{\partial \varepsilon}{\partial x_j} \right] + C_1 \frac{\varepsilon}{k} (G_k + C_3 \varepsilon G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon \tag{20}
\]

The turbulence kinetic energy, \( k \), and the specific dissipation rate, \( \omega \), are obtained from the following transport equations in \( k-\omega \) model [17].

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k - Y_k + S_k \tag{21}
\]

And

\[
\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho \omega u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + S_\omega \tag{22}
\]

The SST \( k-\omega \) [16] isotropic model has a similar transport equation as \( k-\omega \) model above but has limiter on turbulence viscosity \( \mu_t \) for shear stress transport.

### 2.3 Test Case and Results

The commercial CFD software ANSYS Fluent 17.0 is used in flow simulations. A mesh independence study is conducted for each of the computed cases using different models. The maximum wall \( y^+ \) for all cases is less than 1.
2.3.1 The Twin Jet Experiment

Figure 39 shows the experimental apparatus used by Wang et al. [18] in their Laser-Doppler-Velocimetry (LDV) measurements of the turbulent mixing of two rectangular water jets impinging on a stationary water pool. The jet flows emanate from the exit of two circular pipes and then suddenly enter a large tank of water.

![Figure 39 Twin Jet Water Facility.](image)

The experiment was later repeated by Wang et al. [19] by using the Particle Image Velocimetry (PIV). Wang et al. found that there was no difference in the measurements from the two experiments using LDV and PIV [18] [19]; therefore, these experiments provide an excellent data for CFD validation study.

The flow fields of the turbulent mixing of two rectangular water jets impinging on a stationary pool of water are very rich in providing a variety of flow characteristics. In the experiment, the flow region can be divided into three different regions based on the behavior of the jets as shown in Figure 40. These are jets interaction region, jets merging region and the jets combining region. Among these three regions, the jets’ interaction region is most complex due to two features, one
is that there is a relatively low-pressure zone between the two jets, and the jets are deflected toward each other due to entrainment. Another is that there is a backflow in the jets’ interaction region whose velocity magnitude is very small compared to the magnitude of jets’ exit velocity. All these flow characteristics and details are of considerable importance in turbulence modeling of mixing flows containing jets and shear layers.

A 3D rectangular computational domain shown in Figure 41 is employed. The structured mesh is generated by ICEM 17.0 as shown in Figure 41. The mesh is clustered in the jet mixing regions and near the walls. The mesh is orthogonal and is of very high quality as demonstrated in Figure 42.
2.3.2 Validation of Various RANS Turbulence Models

The RANS turbulence models chosen are the k-ε model, SA model and SST k-ω model. These three RANS models are the most frequently used models in industrial applications. First, the velocity profile in the rectangular pipe is computed to validate its correctness. The experimental Reynolds number based on the pipe hydraulic diameter is ~9100 with jet exit velocity of 0.75m/s. The location of computed pipe velocity profile is highlighted by a line as shown in Figure 43.

The velocity profiles in the pipe are shown in Figure 44. Except for the k-ε model, there is not much difference when using other turbulence models. The velocity profiles show that all
turbulence models can capture the fully developed turbulent flow when reaching that location in the pipe which is consistent with the experimental results.

Figure 44 Computed velocity profiles in the pipe using various turbulence models.

When flow enters the stationary pool of water, the flow field becomes very complex. There are two very important points in the flow field. The first one is the jets’s Merging Point (MP), which is defined as the location where the velocity along the symmetry axis becomes zero. The second one is the Combining Point (CP), which is defined as the location where the velocity along the symmetry axis becomes maximum. The results for the locations of each of these points using various turbulence models are compared with the experiment as shown in Table 3.

<table>
<thead>
<tr>
<th>Turbulence Model</th>
<th>MP (non-dimensional distance)</th>
<th>CP (non-dimensional distance)</th>
</tr>
</thead>
<tbody>
<tr>
<td>k-ε</td>
<td>1.93</td>
<td>20</td>
</tr>
<tr>
<td>SST k-ω</td>
<td>Failed to Capture</td>
<td>14.3</td>
</tr>
<tr>
<td>SA</td>
<td>Failed to Capture</td>
<td>17.6</td>
</tr>
<tr>
<td>Experiment</td>
<td>1.72-3.45</td>
<td>19.7</td>
</tr>
</tbody>
</table>

Based on the experiment results [11], there is mutual entrainment between the two jets which corresponds to the sub-atmospheric pressure region. Because of the imbalance of pressure, the jets are curved towards each other. Inside this relatively low-pressure region, strong recirculation near the jet exits exists due to reverse flow. Only one turbulence model k-ε can capture this
behavior; that is why it can predict the Merging Point. The symmetry plane 1 is shown in Figure 45.

The velocity contours, pressure contours and streamlines calculated by each model in the symmetry plane 1 are shown in Figure 46, Figure 47 and Figure 48. All turbulence models capture that the jets are deflected towards each other, which is consistent with the experiment. All turbulence models capture the relatively low-pressure region between the two jets. However, only the $k-\varepsilon$ model can capture the backflow and recirculation quite well.
Figure 46 Velocity contours, pressure contours and streamlines in symmetry plane 1 from $k$-$\varepsilon$ model.

Figure 47 Velocity contours, pressure contours and streamlines in symmetry plane 1 from SA model.
It is well known that $k$-$\varepsilon$ model has problems in computing some flows; however it has been derived with less assumptions and empiricism compared to other two models. The $k$ transport equation, Eq. (19) is derived from incompressible turbulent Reynolds stress transport equation and $\varepsilon$ transport equation, Eq. (20) is derived by considering the dissipation of turbulent kinetic energy. The $k$-$\varepsilon$ has been found to be more accurate for mixing flows such as free shear layers, jets and wakes compared to SA and SST $k$-$\omega$ models. LES simulation was also performed to compare its results from those from $k$-$\varepsilon$ model. Figure 49 shows that the flow field in recirculation region captured by $k$-$\varepsilon$ model is very similar to LES result.
Figure 49 Streamlines in flow field obtained with k-ε model (left) and LES (right).

The velocity profiles at different non-dimensional heights lines are also chosen for validation. The locations for these lines are shown in Figure 50. Figure 51 and Figure 52 show the velocity profile for lines $z/a = 2.0, 3.4, 7.2$ and $12.3$ predicted by the three turbulence models; the results agree with each other reasonably well.

Figure 50 Locations of four lines used for computations of velocity profiles.

Figure 51 Velocity profiles at $z/a=2$ (left) and $z/a=3.4$ (right)
However, when focusing on the development of the twin jet flow, especially after twin jet combine as one single jet, k-ε model is less accurate due to its lack of shear stress transport correction. SST k-ω model shows better accuracy in far field of this twin jet flow field where unbounded flow is dominant instead of bounded flow.

Figure 53 shows the center line’s location. The stream wise velocity (z-velocity) along this center line provides an excellent validation for comparing the accuracy of various turbulence models. Figure 54 gives the centerline velocity decay; it clearly shows that SST k-ω model has a better prediction in the far field. When two jets finish combining as one single jet, the turbulent viscosity is the main reason for slowing down the jet flow. Figure 55 shows the turbulent viscosity distribution along the centerline. Because SST k-ω model has larger turbulent viscosity than the other two models, it can predict the flow without over-predicting.
In conclusion, all turbulent models capture the flow field due to twin jet interaction with acceptable accuracy. For flow details especially in the jets’ interaction region, $k-\varepsilon$ model is recommended, but for the far field region, SST $k-\omega$ model is better than others.

### 2.3.3 Combining Process of Twin Jets (a 3-D Phenomenon)

Most of the experiment data for this twin jet case is available only in the symmetry plane 1 shown in Figure 45. Due to high aspect ratio (87.6/5.8) of the jets’ exit cross-section, the flow in the symmetry plane 1 can be studied as a 2D flow with good accuracy using CFD. Carasik et al. [20] performed the 2-D numerical simulations and showed a good comparison with experimental data in the symmetry plane 1. They also performed a 3-D numerical simulation [21] but still focused mainly on the symmetry plane 1. Actually, for high aspect ratio rectangular jets also,
studying the flow only in symmetry plane 1 is not enough. In this chapter, we consider the symmetry plane 2 defined in Figure 56.

Figure 56 Location of Symmetry Plane 2

The velocity contours in the symmetry plane 2 are shown in Figure 57. They clearly show that near the jets’ exit, there is a relatively high-speed region, which disappears when the flow is far away from the jets’ exits. This figure shows that near the jets exit, the highest velocity in the symmetry plane 2 occurs not in the center, but on two sides of the center. This is unlike the traditional single rectangular jet where the highest velocity always occurs at the center.
Figure 57 Stream-wise velocity contours in symmetry plane 2.

A series of 3-D velocity iso-surfaces are given in Figure 58. From Figure 58, one can clearly see that the jets on two sides of the symmetry plane have higher velocity. This demonstrates that the jets merging starts not from the center but from the two endpoints along the longer side of the rectangular cross-section.
Figure 58 Stream-wise velocity iso-surfaces in the converging region of the two jets

The reason for this 2D behavior in center of rectangular jets exits is the high aspect ratio. If the aspect ratio goes to infinity (becomes very large), it becomes exactly a 2-D problem. With finite aspect ratio, along the longer side of the rectangular slit, 2-D effect becomes more dominant as expected and 3D effect is more dominant on shorter side of the slit. Therefore, the jet flow close to the center becomes more resistant then the jet flow close to the endpoints. That is why the flow near the endpoints has higher velocity. However, when twin jet flows are merging as one single jet, this phenomenon disappears. In Figure 59, several planes parallel to the symmetry plane 1 are shown. They are spaced by 0.01m increment off the symmetry plane 1.
The streamlines colored by velocity in each plane of Figure 59 are shown in Figure 60, Figure 61, Figure 62 and Figure 63. They clearly show that the suction effect due to two jets is stronger when being far away from the symmetry plane 1. They also demonstrate that the twin jet flow will first converges at the endpoints of the longer side. The center flow is the last to converge.
Figure 61 Planes parallel to symmetry plane 1 off by 0.02m

Figure 62 Planes parallel to Symmetry Plane 1 off by 0.03m

Figure 63 Planes parallel to symmetry plane 1 off by 0.04m
2.4 Conclusions

SA, k-ε and SST k-ω turbulence models are applied to compute the flow field of two rectangular jets of high aspect ratio impinging on a water pool. The computations show that k-ε model can be effectively used to capture the details of the twin jet flow field more accurately in region near jet exit slits and SST k-ω model is better in far field region. All the three regions of the twin jets flow field, namely the jets interaction region, the jets merging region and the jets convergence region combining into one jet are reasonably simulated by these three models. The CFD results show that the processing of combining of two jets in twin jet case starts from the ends of long side of slits due to 3-D relieving effect.
Chapter 3: Numerical Study of a Hovering Helicopter Rotor in Ground Effect

3.1 Introduction

This paper focuses on the numerical simulation of a 3-D hovering helicopter rotor blade in ground effect in transonic regime. The study consists of two parts. The first part involves the study of blade pressure distribution, lift on blade sections and rotor thrust performance. Along the span wise direction near the blade tip, local Mach number becomes close to 1 and shock is captured numerically. The numerical results are validated against the experimental data for the hovering rotor blade. When rotor blade is close to the ground, the change in blade pressure distribution is studied. The second part involves mainly the properties of the wake when rotor blade is close to the ground. It includes the trajectory and strength of the wake.

Accurate prediction of flow field and aerodynamics of helicopter blade without and with ground effect is of great interest in the development of efficient rotor blades for modern helicopters flying at higher speeds. When a helicopter is in hover, its flow field is determined by the shape of rotor blades, fuselage and angular velocity of hub and distance above the ground. Therefore, it is very challenging to simulate and calculate flow field and loads on rotating rotor blades in hover without and with ground effect, in particular if it results in transonic flow with shocks near the blade tip. Over the years, many theoretical, numerical and experimental studies have been conducted to understand the flow field and aerodynamics of a hovering helicopter rotor blade. However there are very few studies on a hovering rotor blade in ground effect. The primary focus of this paper is on the calculation of the flow field of a hovering rotor blade in ground effect.
Theoretical methods have mainly employed a free wake trajectory analysis model combined with aerodynamic model to calculate the loads of a rotor blade. Since the analysis time required for theoretical method is usually very short, theoretical methods are still being used widely in the design of rotor blades. In recent years, Peters and his students [22] [23] [24] [25] [26] have used both linear and nonlinear potential flow methods instead to greatly improve the theoretical methods for helicopter rotor blade load predictions in unbounded flow and in ground effect. Experimental studies have not only provided many benchmark tests to aid the development of theoretical methods and numerical methods, but also reveal some valuable details of helicopter flow field in hover. Caradonna and C.Tung [27] have generated massive experiment data for a helicopter rotor at different angles of attack for a wide range of tip Mach numbers. Their experimental study involved simultaneous blade pressure measurements and tip vortex surveys. With development of non-intrusive measurement techniques, velocity field can be directly measured by using particle image velocimetry (PIV) to analyze the vertex structure and its interaction with the blades [28] [29] [29].

In recent years, CFD simulations of flow fields of helicopter rotor blades have come a long way with enormous increase in computing power [30]. Garcia and Barakos [31] conducted a numerical study of the flow fields of two well-studied blades, namely the S-76 main rotor blade and the XV-15 tiltrotor blade. Their CFD results not only matched experimental data but also captured the effects of the tip Mach number, tip shape, blade aeroelasticity and flow transition on the performance of the blade as well as the wake structure and the rotor acoustics. It is worth noting that they used very modest structured mesh with periodical boundary condition to perform a steady-state calculation in a non-inertial reference frame. Kalra and Baeder [32] studied the flow field of a rotor consisting of an untwisted rectangular blade with a NACA2415 airfoil
section by using structured, multiblock mesh with a vortex tracking grid (VTG). They investigated four simulation cases by employing four different turbulent flow prediction models namely the RANS-based Spalart-Allmaras (S-A) model, laminar flow assumption, DES and DES with anisotropic grids. Their results showed that DES with anisotropic grids correction captured the wake of rotor best when compared with experimental data. Sugiura and Tanabe [33] studied the downwash caused by the helicopter rotor in ground effect both numerically and experimentally. Their numerical results indicated that the characteristic wake structure was connected to the taxiing speed. When taxiing speed was over a certain value, the helicopter’s wake developed horseshoe vortices like a fixed-wing. Also, the movement of wake was explored in their numerical work. Their experiments explored the merging, descending and endurance of the wake and supported their CFD results.

Simulation of 3-D rotor helicopter blade rotation usually requires massive number of cells and therefore a long time for calculations. The calculation of Kalra and Baeder [32] required 6.25 days to complete 19 revolutions run in parallel on a 32 AMD Interlagos Poteron 2.3 GHz processors, up to 240 processors were used in each calculation.

In this chapter, the aerodynamics and flow physics of a standard helicopter blade with airfoil section NACA0012 in ground effect for \( \alpha =12 \) deg is studied by using the compressible Reynolds-averaged Navier-Stokes equations to account for the effects of turbulence in the numerical simulations. First, the rotor pressure distribution, thrust and shock behavior are studied at different non-dimensional heights for ground effect. Second, the wake of rotor blade is studied in ground effect.
3.2 Geometry, Mesh, Computational Method and Mesh Independence Study

3.21 Geometry and Mesh

A two-blade helicopter with NACA 0012 blade cross-section section with $R = 2.286m$, $AR = 6$ is studied numerically using ANSYS Fluent software. The AOA of $\alpha = 12$ deg is selected to validate the numerical results as well as to study ground effect in hover. Figure 64 shows the model geometry of computational domain. The boundary conditions which correspond to different parts of the computational domain are given in Table 4.

<table>
<thead>
<tr>
<th>Boundary Condition Type</th>
<th>Rotor Blades Surfaces</th>
<th>Lower Disk Surface</th>
<th>Upper Disk Surface</th>
<th>Side Surface of Column</th>
</tr>
</thead>
<tbody>
<tr>
<td>Boundary</td>
<td>No Slip Wall</td>
<td>No Slip Wall</td>
<td>Pressure Outlet</td>
<td>No Slip Wall</td>
</tr>
</tbody>
</table>

The size of geometry in this paper is identical to that employed in the experiment of Caradonna and Tung [27] in order to validate the numerical simulations.

![Figure 64 Model Geometry](image)

Figure 65 shows the structured mesh in the entire computational domain along with a zoomed-in view of the mesh near the tow rotor blades. The structured grid generation is similar to the grid generated by Knoth and Breitsamter [34] for a helicopter engine. The whole computation domain is subdivided into fourteen sub-domains. Entire mesh is a block-structured mesh. In order to increase to the numerical accuracy and decrease the numerical truncation error, all meshes are
blocked structured and over 97% of the elements are hexahedrons. The meshes are created by using ANSYS ICEM CFD and comprise of 84 blocks in the whole computation domain.

O-grids are applied in the entire cylindrical domain in Figure 65 for high quality orthogonal mesh. Y-grids are applied at the leading edges and trailing edges of the rotor blades to capture the curvature with high quality orthogonality. The height of first layer chosen for the elements is $10^{-6}$ meter which results in a dimensionless wall distance of $y^+$ less than 1. Additionally, in combination with an expansion ration of 1.1, the boundary layer including the viscous sublayer can be accurately calculated without applying a wall function.

### 3.22 Computational Method

In order to reduce the computational cost, following the formulation of Garcia and Barakos [31], the governing equations are cast and solved as a steady-state problem in a non-inertial reference frame. Consider a coordinate system that is translating with a linear velocity $\vec{v}_t$ (in this paper $\vec{v}_t = 0$) and rotating with angular velocity $\vec{\omega}$ (in this paper $\vec{\omega} = \Omega$) relative to a stationary (inertial) reference frame as shown in Figure 66. The origin of the moving system is located at a position vector $\vec{r}_0$. 

---

59
The axis of rotation is denoted by a unit direction vector \( \hat{a} \) such that \( \vec{\omega} = \Omega \hat{a} \). The computational domain for the CFD problem is defined with respect to the moving frame, such that an arbitrary point in the CFD domain is located by a position vector \( \vec{r} \) from the origin of the moving frame. The fluid velocities can be transformed from the stationary frame to the moving frame by using the following relation:

\[
\vec{v}_r = \vec{v} - \vec{u}_r
\]  

(23)

where

\[
\vec{u}_r = \vec{v}_t + \vec{\omega} \times \vec{r}
\]  

(24)

In the above equations, \( \vec{v}_r \) is the relative velocity, \( \vec{v} \) is the absolute velocity, \( \vec{u}_r \) is the velocity of the moving frame relative to the inertial reference frame, \( \vec{v}_t \) is the translation frame velocity, and \( \vec{\omega} \) is the angular velocity.

For the relative velocity formulation, the governing equations of fluid flow in a moving reference frame can be written as follows:

Conservation of mass:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot \rho \vec{v}_r = 0
\]  

(25)

Conservation of momentum:
\[
\frac{\partial}{\partial t}(\rho \ddot{v}_r) + \nabla \cdot (\rho \ddot{v}_r) + \rho(2 \ddot{\omega} \times \ddot{v}_r + \ddot{\omega} \times \dddot{\omega} \times \ddot{r} + \dddot{\alpha} \times \ddot{r} + \dddot{a})
\]

\[
= -\nabla p + \nabla \cdot \bar{\tau}_r
\]  

where

\[
\ddot{\alpha} = \frac{d \ddot{\omega}}{dt}
\]

and

\[
\ddot{a} = \frac{d \ddot{v}_r}{dt}
\]

Conservation of energy:

\[
\frac{\partial}{\partial t}(\rho E_r) + \nabla \cdot (\rho \ddot{v}_r H_r) = \nabla \cdot (k \nabla T + \bar{\tau}_r \cdot \ddot{v}_r)
\]

Here, \(E_r\) is the relative internal energy, \(H_r\) is the relative total enthalpy and \(\bar{\tau}_r\) is the viscous stress. \(E_r\) and \(H_r\) are defined as:

\[
E_r = h - \frac{p}{\rho} + \frac{1}{2}(u_r^2 - u_r^2)
\]

\[
H_r = E_r + \frac{p}{\rho}
\]

The turbulence model is chosen as The Spalart-Allmaras model [13] which is a one-equation model that 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.

In this work, the double precision solver in ANSYS FLUENT is employed in all calculations. The solver type is a steady density-based solver to avoid oscillation in solutions which sometime
occur with the use of a pressure-based solver. The formulation is implicit. The flux type is chosen as Roe Flux-Difference Splitting Scheme [35]. Spatial discretization for gradient is chosen as cell based least square and second order upwind is chosen for both flow and modified turbulent viscosity.

3.23 Mesh Independence
The mesh independence of the solution is ascertained by computing the flow field of at AOA of 12 deg., tip Mach number is 0.794 and non-dimensional height is 0.5. The results are summarized in Table 5, demonstrating that the fine mesh results can be considered as mesh independent.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Cell number</th>
<th>y+</th>
<th>CT</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>4002365</td>
<td>3</td>
<td>0.00980</td>
</tr>
<tr>
<td>Medium</td>
<td>5003000</td>
<td>2</td>
<td>0.009738</td>
</tr>
<tr>
<td>Fine</td>
<td>7212200</td>
<td>≤1</td>
<td>0.009736</td>
</tr>
</tbody>
</table>

3.3 Validation and Results Analysis
In this section, first CFD validation results for a test case are shown to demonstrate the accuracy of the numerical method which is then followed by detailed discussion of other numerical results for ground effect flow fields.

3.31 Load of Rotor Blades
The positions of section slices are shown in Figure 67, and the pressure distributions on the intersecting lines of these slices and blades are validated in Figure 68, Figure 69, Figure 70, Figure 71 and Figure 72. These slices are located at r/R = 0.5, 0.68, 0.80, 0.89, 0.96. The experimental data at these intersecting lines are available in the paper [27].
Figure 68, Figure 69, Figure 70, Figure 71 and Figure 72 show the CFD validation of pressure distributions on these intersecting lines; it can be concluded that the CFD results match the experimental data [27] very well.
Also Figure 68, Figure 69, Figure 70, Figure 71 and Figure 72 indicate that along the span wise direction near the tip, local Mach number becomes close to 1 and shock is captured numerically. Additionally, shock only occurs at lower surface of rotor blades.
It has been known that the proximity of the ground surface to the hovering rotor blades constrains the rotor wake and reduces the induced velocity at the rotor, which means a reduction in the power required for a given thrust [36]. In this paper, when the angular velocity is fixed, the thrust increases as the rotor blades approach the ground. Figure 73 shows the variation of thrust coefficient ratio with non-dimensional heights above the ground. Our CFD result is in very close agreement with the theoretical and experimental results of Zbrozek [37].

Figure 73 Ground effect a thrust increase at constant angular velocity

Figure 74, Figure 75, Figure 76, Figure 77 and Figure 78 show the difference in the pressure distribution on the blade sections between the ground effect (h=0.05) and unbounded case (h=infinity). Due to its large aspect ratio, the middle part of span along the rotor blade can be treated as 2-D tunnel as Qu et al. did in their study of 2-D airfoil ground effect [38]. In the middle of the span, the air flow between the ground surface and the lower surface of the blade is slowed due to a convergent nozzle and the air flow above the upper surface is accelerated due to mass conservation. Therefore, pressure coupled with velocity drops down on upper surface and increases on the lower surface. Thus, in the middle part of the span where shock does not appear, ground effect contributes to lift increase.

Along span near to tip, where shock appears, there is not only similarity but also difference between the results of this paper and those of Gao et al. [39] on transonic shock in 2-D ground
effect shock. Firstly, the similarity is that as the rotor approaches the ground, the shock on the upper surface moves towards the leading edge causing the pressure to increase on the upper surface. However, in this paper shock on the blade is far from middle part of the span which means 3-D relieving effect plays an important role. The difference between this paper and that of Gao et al. [39] is that the shock cannot move so much as in purely 2-D cases. In Gao’s [39] work, the shock moves around 0.25c between unbounded case and ground effect case. But here, the maximum distance shock moves is only less than 0.15c. That’s why in Gao’s 2-D cases the lift decreases as the airfoil approaches the ground [39] and in 3-D rotor blade cases considered in this work, the thrust increases as the rotor blade approaches the ground.

![Figure 74 Pressure distribution (r/R=0.50)](image1)

![Figure 75 Pressure distribution (r/R=0.68)](image2)
3.32 Wake of Rotor Blades

Figure 79 shows the location of symmetry plane in the computational domain.
Figure 80, Figure 81, Figure 82 and Figure 83 show velocity contours in the symmetry plane and the streamline colored by vorticity magnitude.

Figure 80 velocity magnitude contour and streamline from rotor blades (h=0.05)

Figure 81 velocity magnitude contour and streamline from rotor blades (h=0.5)

Figure 82 velocity magnitude contour and streamline from rotor blades (h=1.0)
Figure 80, Figure 81, Figure 82 and Figure 83 indicate that the trajectory of wake can interact with rigid ground surface causing decrease in the strength of the wake decreasing.

### 3.4 Conclusions

In this chapter, the flow field of 3-D rotor blade is simulated at different non-dimensional heights above the ground. The following conclusion are made:

1) By using very modest computing resource, current CFD technology is able to calculate the flow of rotor blade in hover without and with ground effect in excellent agreement with experiments.

2) When rotor blade with large aspect ratio is hovering close to the ground, flow field can be analyzed by the 2-D airfoil theory at the middle of span and considering the 3-D relieving effect near to the tip.

3) When rotor blade is close to ground, the trajectory of wake in hover is interfered by the ground causing the strength of the wake to decrease.
Chapter 4: Conclusion

In this thesis, the flow fields of a single air jet impingement on a solid ground and water surface, a twin water jet impingement on a water pool and a hovering helicopter rotor blade in ground effect are studied by using modern CFD technology. Detailed flow fields are obtained and flow physics is analyzed. Excellent results are obtained from simulations which agree reasonably well with the experimental data.
References


Appendix
UDF for Boundary Condition in Chapter 1

#include "udf.h"
DEFINE_PROFILE(pressure_profile,t,i)
{
    real y[ND_ND];
    real x;
    face_t f;
    begin_f_loop(f,t)
    {
        F_CENTROID(y,f,t);
        x=y[0];
        if(x>=0)
            {F_PROFILE(f,t,i)=0;}
        else
            {F_PROFILE(f,t,i)=998.2*9.8*(-x);}
    }
    end_f_loop(f,t)
}  
#include "udf.h"
DEFINE_PROFILE(vof_profile,t,i)
{
    real y[ND_ND];
    real x;
    face_t f;
    begin_f_loop(f,t)
    {
        F_CENTROID(y,f,t);
        x=y[0];
        if(x>=0)
            {F_PROFILE(f,t,i)=0;}
        else
            {F_PROFILE(f,t,i)=1;}
    }
    end_f_loop(f,t)
}

#include "udf.h"
DEFINE_PROFILE(temperature_profile,t,i)
{
    real y[ND_ND];
    real x;
    face_t f;
    begin_f_loop(f,t)
    {
        F_CENTROID(y,f,t);
        x=y[0];
        if(x>=0)
            {F_PROFILE(f,t,i)=313;}
        else
            {F_PROFILE(f,t,i)=313;}
    }
    end_f_loop(f,t)
}
\{F_{PROFILE}(f,t,1) = 284.3;\}
}
end_f_loop(f,t)
Curriculum Vita

Han Gao

Degrees
M.S. Mechanical Engineering, May 2018
B.S. Shanghai University of Electric Power, Thermal dynamics, June 2016