Washington University in St. Louis

Washington University Open Scholarship

McKelvey School of Engineering Theses & Dissertations

McKelvey School of Engineering

Spring 5-18-2017

Numerical Study of the Aerodynamics of DLR-F6 Wing-Body in Unbounded Flow Field and in Ground Effect

Ning Deng

Washington University in St. Louis

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

Part of the [Aerodynamics and Fluid Mechanics Commons](#)

Recommended Citation

Deng, Ning, "Numerical Study of the Aerodynamics of DLR-F6 Wing-Body in Unbounded Flow Field and in Ground Effect" (2017). *McKelvey School of Engineering Theses & Dissertations*. 252.
https://openscholarship.wustl.edu/eng_etds/252

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.

WASHINGTON UNIVERSITY IN ST. LOUIS

School of Engineering and Applied Science
Department of Mechanical Engineering and Material Science

Thesis Examination Committee:

Ramesh Agarwal, Chair

Qiulin Qu

Swami Karunamoorthy

Numerical Study of the Aerodynamics of DLR-F6 Wing-Body
in Unbounded Flow Field and in Ground Effect

by

Ning Deng

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 Engineering

May 2017

St. Louis, Missouri

© 2017, Ning Deng

Table of Contents

List of Figures	iv
List of Tables	vii
Acknowledgments.....	viii
ABSTRACT OF THE THESIS	x
Chapter 1: Introduction	14
1.1 Ground Effect.....	14
1.2 Ground Effect of a Wing.....	14
1.2.1 Chord Dominated Ground Effect of Single-Element Airfoil.....	15
1.2.2 Chord Dominated Ground Effect of Multi-Element Airfoil	17
1.2.3 Span Dominated Ground Effect of Single-Element Wing	18
1.3 Ground Effect of Slender Body.....	19
1.4 Motivation	19
1.4.1 To Enhance the Safety during the Takeoff and Landing Phases.....	19
1.4.2 To Study the Ground Effect of Wing-body.....	20
Chapter 2: Physical Model.....	22
2.1 Body Shape	22
2.2 Flow Conditions	23
Chapter 3: Numerical Method	24
3.1 Methodology for CFD Simulation	24
3.1.1 Turbulence Model.....	24
3.1.2 Boundary Condition for the Ground	27
3.1.3 Convergence Monitors and Criteria.....	28
3.2 Computational Domain and Boundary Conditions	28
3.3 Mesh Generation	29
Chapter 4: Validation of Computed Solutions for Wing-Body in Unbounded Flow	31
4.1 Flow Conditions Used in Validation.....	31
4.2 Results of the Validation.....	31
4.2.1 Aerodynamic Forces	31
4.2.2 Pressure Distribution.....	33

4.2.3	Separation Bubble	35
Chapter 5: Simulation Results		39
5.1	Influence of Ride Height	39
5.1.1	Influence of Ride Height on the Lift Coefficient	44
5.1.2	Influence of Ride Height on the Drag Coefficient	51
5.1.3	Influence of Ride Height on the Pitching Moment Coefficient	53
5.1.4	Influence of Ride Height on the Separation Bubble	58
5.2	Influence of Angle of Attack in Ground Effect.....	61
5.2.1	Influence of AOA on Lift Coefficient in Ground Effect.....	62
5.2.2	Influence of AOA on Drag Coefficient in Ground Effect.....	65
5.2.3	Influence of AOA on Pitching Moment Coefficient in Ground Effect.....	66
Chapter 6: Conclusions		68
Chapter 7: Future Work		69
References		70
Vita.....		73

List of Figures

Figure 2.1 DLR-F6 wing-body geometry.	22
Figure 3.1 Computational domain of DLR-F6.	29
Figure 3.2 Surface grid on DLR-F6.	30
Figure 4.1 Comparison of C_L for DLR-F6.	32
Figure 4.2 Comparison of C_D for DLR-F6.	32
Figure 4.3 Comparison of C_M for DLR-F6.	32
Figure 4.4 Wing section locations.	33
Figure 4.5 Comparison of computed and experimental pressure distributions at $M = 0.75$ and $\alpha = 0.49^\circ$	34
Figure 4.6 Oil-flow visualization of the separation bubble.	35
Figure 4.7 Geometrical parameters of the separation bubble.	36
Figure 4.8 Comparison of parameters of separation bubble.	37
Figure 5.1 Vorticity contours and streamlines on various cross sections behind DLR-F6 IGE ($\alpha = 1.23^\circ, h/c = 0.5$).	40
Figure 5.2 Vorticity contours and streamlines on various cross sections behind DLR-F6 OGE ($\alpha = 1.23^\circ, h/c = \infty$).	40
Figure 5.3 Pressure contours on lower and upper surfaces of DLR-F6 at $\alpha = 0.49^\circ$	41
Figure 5.4 Variation in C_L with h/c	42
Figure 5.5 Variation in C_D with h/c	42
Figure 5.6 Variation in C_M with h/c	42
Figure 5.7 Variation in ΔC_L with h/c	43
Figure 5.8 Variation in ΔC_D with h/c	43

Figure 5.9 Variation in C_M with h/c	43
Figure 5.10 Comparison of pressure distributions with h/c at eight wing sections at $\alpha = 0^\circ$	44
Figure 5.11 Pressure difference with respect to $h/c = \infty$ on eight wing sections at $\alpha = 0.49^\circ$.	45
Figure 5.12 Comparison of pressure distributions with h/c at eight wing sections at $\alpha = 0.49^\circ$	45
Figure 5.13 Pressure difference with respect to $h/c = \infty$ on eight wing sections at $\alpha = 0.49^\circ$.	46
Figure 5.14 Comparison of pressure distributions with h/c at eight wing sections at $\alpha = 0.49^\circ$	46
Figure 5.15 Pressure difference with respect to $h/c = \infty$ on eight wing sections at $\alpha = 0.49^\circ$.	47
Figure 5.16 Comparison of pressure contours OGE and IGE around a wing section in the plane $y/b = 0.239$	48
Figure 5.17 Comparison of velocity contours in OGE and IGE around a wing section in the plane $y/b = 0.239$	48
Figure 5.18 Variation in C_l distributions on the fuselage with h/c	49
Figure 5.19 Variation in ΔC_l distributions on the fuselage with h/c	50
Figure 5.20 Span-wise trajectory of the wingtip vortex for various h/c	53
Figure 5.21 Variation in C_m distributions on the fuselage with h/c	54
Figure 5.22 Variation in ΔC_m distributions on the fuselage with h/c	55
Figure 5.23 Pressure contours in the plane of overshoot plane at $\alpha = 0^\circ$	56
Figure 5.24 Pressure contours in the plane of overshoot plane at $\alpha = 0.49^\circ$	57
Figure 5.25 Pressure contours in the plane of overshoot plane at $\alpha = 1.23^\circ$	57
Figure 5.26 Pressure distribution on the surface of fuselage in the plane of overshoot	58
Figure 5.27 ΔC_p distribution on the surface of fuselage in the plane of overshoot	58

Figure 5.28 3-D streamlines close to the upper surface of the wing near the wing root at $\alpha = 0.49^\circ, h/c = 0.25$.	59
Figure 5.29 Surface streamlines on the wing and fuselage at the corner of wing root and trailing edge at $\alpha = 0^\circ$.	60
Figure 5.30 Surface streamlines on the wing and fuselage at the corner of wing root and trailing edge at $\alpha = 0.49^\circ$.	60
Figure 5.31 Surface streamlines on the wing and fuselage at the corner of wing root and trailing edge at $\alpha = 1.23^\circ$.	60
Figure 5.32 Variation in C_L with α .	61
Figure 5.33 Variation in C_D with α .	62
Figure 5.34 Variation in C_M with α .	62
Figure 5.35 Pressure distribution on eight wing sections at $h/c = 1$.	63
Figure 5.36 Pressure difference with respect to $\alpha = 0^\circ$ on eight wing sections at $h/c = 1$.	63
Figure 5.37 Variation in C_l distributions along stream-wise direction on the fuselage with AOA.	64
Figure 5.38 Variation in ΔC_l distributions along stream-wise direction on the fuselage with AOA.	65
Figure 5.39 Variation in C_m distributions along stream-wise direction on the fuselage with AOA.	66
Figure 5.40 Variation in ΔC_m distributions along stream-wise direction on the fuselage with AOA.	67

List of Tables

Table 1.1 Fatal accidents and onboard fatalities by phase of flight.....	20
Table 2.1 Geometric properties of DLR-F6 wing-body.	23
Table 5.1 Comparison of ΔC_L on the fuselage	50
Table 5.2 Comparison of ΔC_M on the fuselage	55

Acknowledgments

First of all, I would like to acknowledge my research advisor, Dr. Ramesh Agarwal, for his patient guidance and persistent encouragement. His rich experience and enthusiasm on academics impress me profoundly and motivate me to keep stepping forward on the road of research.

Secondly, those who helped me with this thesis are acknowledged for their accumulated effort without reciprocation. Thanks to Dr. Qiulin Qu for his detailed instruction on research and presentation preparation. Thanks to Dr. Tim Wray for the discussion about turbulence model. Thanks to Junhui Li for his help with complex grid generation. Thanks to Boshun Gao with his suggestions about numerical simulation and data analysis. Thanks to Xu Han, and Xiao Zhang for their advice. Thanks to all colleagues in the CFD lab for their kindness and company.

In addition, I would like to acknowledge the committee members, Dr. Ramesh Agarwal, Dr. Qiulin Qu and Dr. Swami Karunarmoorthy for participation in my oral defense and their advice on the thesis.

Ning Deng

Washington University in St. Louis

May 2017

Dedicated to my family for their emotional support.

ABSTRACT OF THE THESIS

Numerical Study of the Aerodynamics of DLR-F6 Wing-Body

in Unbounded Flow Field and in Ground Effect

by

Ning Deng

Master of Science in Mechanical Engineering

Washington University in St. Louis, 2017

Research Advisor: Professor Ramesh Agarwal

The main focus of this thesis is on the simulation of flow past a three-dimensional wing-body configuration (DLR-F6) in ground effect; a complex 3D wing-body configuration in ground effect has never been analyzed in the aerodynamics literature to date. For the purpose of validation of the simulation approach, computations are performed for the DLR-F6 wing-body in unbounded flow and are compared with the experimental data. The commercial CFD solver ANSYS FLUENT is employed for computations. Compressible Reynolds-Averaged Navier-Stokes (RANS) equations in conjunction with Spalart-Allmaras (SA) and $k-\omega$ Shear Stress Transport (SST) turbulence models are solved. The validated code is employed to calculate the flow field in ground effect; the effects of flight heights above the ground and angle of attack on the aerodynamic properties and flow field are analyzed.

Chapter 1: Introduction

This chapter describes the background of the research on ground effect. Brief definition and classification of ground effect are given. The relevant literature of ground effect of airfoil, wing and slender body is introduced. The motivation for the study of ground effect is stated.

1.1 Ground Effect

When an object moves closer to the ground, the airflow between the object and the ground is forced to become parallel to the ground. Under this condition, the flow physics and aerodynamics of the object become different than those in the flow without boundaries (unbounded flow). This phenomenon is known as the ground effect (GE). Typically, a clean airfoil or wing generates higher lift when in proximity to the ground at moderate angles of attack; it is known as the positive GE. By taking advantage of the positive GE, wing-in-ground-effect (WIG) crafts are designed to fly in proximity of the ground [1-5]. Compared to traditional transport airplane, a WIG craft has higher lift to drag ratio, needs lower propulsive power, can carry larger load, and has wider flight range due to positive GE. The commercial aircrafts on the other hand experience reduced lift when taking off or landing; it is known as the negative GE. In case of a racing car, an inverted highly cambered airfoil produces a downward lift near the ground. The closer is the airfoil to the ground, the greater is the down force. It is again a positive GE.

1.2 Ground Effect of a Wing

The ground effect for a wing has been thoroughly studied in the literature. Based on the flow physics, GE can be divided into two categories: the two-dimensional (2-D) chord-dominated GE and the three-dimensional (3-D) span-dominated GE [3, 6, 7]. For a 2-D airfoil at positive angle

of attack (AOA), ground proximity generally causes a high-pressure distribution on the lower surface of the airfoil leading to increase in lift, and higher lift to drag ratio; this phenomenon is called the 2-D chord-dominated ground effect. For a 3-D wing at positive AOA, ground proximity pushes the wingtip vortices outward along the span leading to decrease in downwash angle and induced drag; this phenomenon is called the 3-D span-dominated GE.

1.2.1 Chord Dominated Ground Effect of Single-Element Airfoil

The previous research on chord dominated GE has mainly focused on 2-D airfoils [8-14].

Coullietter and Plotkin [10] used the analytical and numerical methods to study the airfoils with zero thickness and non-zero thickness in GE. For a zero-thickness airfoil at a certain ground height, the lift decreases as the camber ratio increases; for a non-zero thickness airfoil, lift grows with the thickness ratio. Hsiun and Chen [15] studied the effect of camber and thickness on the aerodynamics of a 2D airfoil in GE by numerical method. They compared the aerodynamic results of NACA0006, NACA0009, NACA0012, NACA2412 and NACA4412 airfoils at different AOA and ride heights, and concluded that the aerodynamic forces are determined by the shape of the passage between the lower surface of the airfoil and the ground. Ahmed et al. [14] reported the wind tunnel experimental results for a NACA4412 airfoil in GE for $\alpha = 0^\circ \sim 10^\circ$. A strong suction effect was observed on the lower surface of the airfoil at $h/c = 0.05$ and $\alpha = 0^\circ$. At AOA of 4° and above, the high pressure coefficients were recorded on the lower surface at small ride heights, which contributed to a gain in lift. There is a loss in the suction on upper surface at small ride heights for all AOA, contributing to a reduction in the lift. For AOA up to 4° , the loss on the upper surface is higher than the gain on the lower surface, resulting in a lower lift close to the ground. For higher AOA of 8° and 10° , the pressure rise on the lower surface was considerable, resulting in a higher lift close to the ground. Qu et al. [16] investigated

the flow physics and aerodynamics of a NACA4412 airfoil in GE for a wide range of $\alpha = -4^\circ \sim 20^\circ$ by numerical simulations. For low to moderate AOA, when the ride height is reduced, the airflow is blocked in the convergent passage between the lower surface of the airfoil and the ground resulting in increase of pressure on the lower surface of the airfoil; at the same time, there is less upward deflection of the streamlines and the effective AOA decreases resulting in increase in pressure on the upper surface of the airfoil. For high AOA, when the ride height is reduced, the adverse pressure gradient along the chord-wise direction increases resulting in a larger region of separated flow. Additionally, for negative AOA generating negative lift, the airflow accelerates in the convergent-divergent passage between the lower surface of the airfoil and the ground due to the Venturi effect resulting in a large suction on the lower surface of the airfoil.

Ahmed and Sharma [17] studied a NACA 0015 in GE in a low turbulence wind tunnel. The AOA and the ride height both had a strong influence on the aerodynamic characteristics. A suction effect was observed on the lower surface of airfoil at certain ride heights at AOA up to 5° . At higher AOA, high pressures were recorded on the lower surface, which resulted in a higher lift. However, the pressure distribution on the upper surface of airfoil did not show significant variation with ride height. Zerihan and Zhang [18], and Mahon and Zhang [19] performed numerical simulation and wind tunnel experiment to study the negative GE of the Tyrrell-02 airfoil, which is a highly cambered inverted airfoil. When the ride height was reduced, the downforce first increased gradually to a peak value and then decreased.

As discussed above, for a single-element airfoil with low to moderate AOA, the pressure on the lower surface increases and the suction on the upper surface decreases with decreasing ride height. As the pressure gain on the lower surface of the airfoil becomes greater than the pressure loss on the upper surface of the airfoil, the lift increases with the decreasing ride height.

1.2.2 Chord Dominated Ground Effect of Multi-Element Airfoil

The aerodynamics and flow physics of the 2-D high-lift devices in GE has also been studied thoroughly. During take-off and landing, GE further accentuates the complexity of the flow around the high-lift devices. Recant [20] conducted experimental studies on a two-element airfoil (NACA23012 with slotted flap, $\delta_f = 40^\circ$) in GE. He found that when the ride height decreased, the lift at $\alpha = -6^\circ \sim 4^\circ$ changed without a pattern, however the lift at $\alpha = 6^\circ \sim 12^\circ$ continually decreased. Yang et al. [21] reported numerical results for a three-element airfoil LIT2 ($\delta_s = 25^\circ$ and $\delta_f = 20^\circ$) and a two-element airfoil (modified from LIT2) in GE. Their results indicated that the lifts of both the airfoils decreased gradually as the ride height was reduced, but the decrease was very small. For GAW-1 ($\alpha = 2^\circ, 5^\circ, 8^\circ$) and 30P30N ($\alpha = 6^\circ, 12^\circ, 18^\circ$) airfoils, the lift decreased all the way when the ride height decreased. Furthermore, lower the ride height, larger was the reduction in lift. Qu et al. [22] proposed a new evolution method to study the GE of two-element airfoils. It was found that the increase in camber strengthened the reduction of effective AOA in GE resulting in an increase in the lift loss of the upper surface; the camber increment limited the pressure increment margin on the lower surface in GE resulting in a decrease in the lift enhancement of the lower surface. Gratzner and Mahal [23] studied the aerodynamics of a STOL aircraft in GE using theoretical analysis and wind tunnel experiment. They found that the slope of lift curve decreased, and the pressure on the upper surface decreased with the decreasing ride height ($\delta_f = 50^\circ, \alpha = 8^\circ$). Qu et al. [24] numerically studied the ground effect of the 30P30N airfoil at $\alpha = 0^\circ \sim 24^\circ$. It was demonstrated that the large geometry camber results in reduction of effective camber and weak blockage effect; hence the lift in GE decreases.

1.2.3 Span Dominated Ground Effect of Single-Element Wing

For the span dominated ground effect, majority of the investigations have focused on clean wings about the aerodynamics and the trajectories of the wingtip vortices, while a multi-element wing has been seldom studied in literature. The earliest theory of span dominated GE was developed by Wieselsberger [25] in 1922. He used the lifting-line theory and the mirror method to calculate the induced drag in GE. Moon et al. [26] conducted numerical studies on the influence of the aspect ratio on the aerodynamics of a 3D wing of an Aero-levitation Electric Vehicle (AEV) in ground effect. It was found that the lift to drag ratio decreased as the span was reduced due to the formation of an arch vortex at the junction of the main and vertical wings. Chawla et al. [27] studied the effect of endplates on the aerodynamics of a wing with NACA4415 section and aspect ratio of 2.33 using a fixed ground. The AOA was varied from 0° to 25° . They found that the use of endplates significantly improved the lift at small ride heights. Han and Cho [28] studied the motion of wingtip vortices in ground effect using discrete vortex method. It was noted that the wingtip vortices moved outwards along the span direction. Additionally, the ground restrained the downward movement of the wingtip vortices. Lee [29] reported experimental results about the motion of wingtip vortices of a rectangular wing in ground effect. It was found that the motion of wingtip vortices could be divided into two stages: (1) the downward movement of vortices due to the induced velocity effect and (2) the slowing down of the motion of descending vortices due to the presence of the ground and their outward movement along the span-wise direction due to the mirror-image effect from the ground. Harvey and Perry [30] conducted the experiments to study the path of wingtip vortices in ground effect. They found that the wingtip vortices first descended to the ground and then rebounded downstream; it was because the wingtip vortices first induced the secondary vortices from the ground and then these secondary vortices induced the wingtip vortices to move upwards. Qu et

al. [31] numerically simulated the flow past a rectangular wing with NACA4412 airfoil section in ground effect. They concluded that the wingtip vortex moves outward along the span-wise direction due to the ground mirror effect, and rebounds in the vertical direction due to the induction from the secondary vortex generated from the ground boundary layer. The strength of the near-field wingtip vortex along the flow direction depends not only on the initial vortex strength and the shear layer developing from the trailing edge of the wing, but also on to the generation of the secondary vortex in the ground boundary layer and the interaction between the wingtip vortex and the secondary ground vortex.

1.3 Ground Effect of Slender Body

The ground effect of slender body has also been studied in literature. Wang and Tan [32] studied the lift and pitching moment of a slender spheroid moving near flat ground at $\alpha = 1^\circ, 1.5^\circ$ and 2° . They found when ride height decreases, the lift decreases and the nose-up pitching moment increases for the slender body. This conclusion can be used to approximately predict the aerodynamics of fuselage in ground effect, since a fuselage can be treat as a slender body.

1.4 Motivation

1.4.1 To Enhance the Safety during the Takeoff and Landing Phases

Generally, a flight includes the following phases: taxi, takeoff, climb, cruise, descent, approach and landing. For the phases of takeoff and initial climb, and final approach and landing, the exposure time is very small. According to a statistical summary of commercial jet airplane accidents from 1959-2008 [33], the fatal accidents and onboard fatalities in these two phases account for more than 50% of all accidents, shown in Table 1.1. Therefore, landing and takeoff are the most dangerous phases of a flight. The research on the aerodynamics of the transport

aircraft in ground effect is helpful to understand the flight in these two phases and thus improve the safety design of the aircraft.

Table 1.1 Fatal accidents and onboard fatalities by phase of flight.

Phases	Taxi	Takeoff and Initial Climb	Climb	Cruise	Descent	Initial Approach	Final Approach and Landing
Fatal Accidents	12%	20%	10%	8%	4%	10%	36%
Onboard Fatalities	0%	30%	13%	16%	4%	12%	25%
Exposure (Percentage of flight time estimated for a 1.5-hour flight)	N/A	2%	14%	57%	11%	12%	4%

1.4.2 To Study the Ground Effect of Wing-body

Although a significant body of literature exists about the ground effect of a single body shape such as a wing and a slender body, there are hardly any studies on the ground effect of a wing-body. An aircraft mainly consists of wings, fuselage, nacelle, and empennage, and aerodynamics of each part is affected by the ground effect. Since the nacelle and empennage have minor impact on the aerodynamics of an aircraft, the wing-body consisting of wing and fuselage plays the most significant role in the aerodynamics of an aircraft. Therefore high-lift devices, which are widely used in landing and takeoff of a transport aircraft, should be studied in ground effect. However, since multi-element wing has been rarely studied, the ground effect of a wing-body with clean wing should be investigated first. Then a 3-D high-lift configuration in ground effect should be studied as a case closer to the real situation. As stated previously, the ground has different impact on the aerodynamics of the wing and fuselage compared to the unbounded flow. In ground effect, the interaction between the wing and fuselage makes a significant difference compared to

the situation with a single body shape in ground effect. The dominant part between wing and fuselage should be investigated in ground effect. Hence, in this thesis, the wing-body is considered to determine which part, wing or fuselage, dominates in ground effect.

In order to study the flow past a transport aircraft in ground effect, the flow past a wing-body with clean wing in unbounded flow should be investigated first followed by a 3D high-lift configuration. These flow fields should be validated against the experimental data. The validated code should then be used to study the wing-body in ground effect and the 3D high lift wing-body configuration in ground effect. The German Aerospace Research Center F6 Geometry (DLR-F6) is a typical wing-body designed for cruise condition and DLR-F11 is a typical high-lift configuration similar to those employed on a commercial aircraft. As a first step towards achieving these goals, this thesis first focuses on the aerodynamics and the near-field wingtip vortex characteristics of DLR-F6 wing-body in unbounded flow and compares the calculations with available experimental data and then the validated code is used to study the DLR-F6 wing-body in ground effect. Computations are performed by solving the Reynolds-averaged Navier-Stokes (RANS) equations with the Spalart-Allmaras (SA) turbulence model. The flow physics resulting from the interaction between the wing-body and the ground is analyzed and discussed. The effect of ground on DLR-F6 at different flight heights and different angles of attack is simulated and its effects on the aerodynamic characteristics are investigated. In the future, the flow past DLR-F11 will be studied based on the results and conclusions obtained for the DLR-F6 in GE.

Chapter 2: Physical Model

In this chapter, the physical model employed for simulation is briefly described. Some properties of the body shape used in the calculation are listed. The flow conditions studied in the simulation are introduced.

2.1 Body Shape

Figure 2.1 shows the DLR-F6 wing-body, which is representative of modern wide-body transport aircraft of Airbus type. A list of DLR-F6 properties is given in Table 1. It was designed by DLR more than 20 years ago and was derived from the DLR-F4 configuration. F6 is a modification of F4 with the aim to have a more elliptic lift distribution and less boundary layer separation at the rear upper wing surface. The wing is defined with four airfoil sections, among which the first section is Ha5 airfoil and the 2nd to 4th sections are R4/4 airfoil. The Translation of airfoil in vertical direction in kink area is to make a smoother surface than for DLR-F4. Its flow field was studied in a wind tunnel in August 1990 as a cooperative program between ONERA and DLR [34].

Figure 2.1 DLR-F6 wing-body geometry.

Table 2.1 Geometric properties of DLR-F6 wing-body.

Property	Value
$S_{ref}/2$	$72,700.0 \text{ mm}^2$
c	141.2 mm
$b/2$	585.647 mm
AR	9.4356
M_{des}	0.75
X_{ref}	157.9 mm
Y_{ref}	0.0 mm
Z_{ref}	-33.92 mm

2.2 Flow Conditions

In the simulations, the flow field about DLR-F6 is computed at $\alpha = 0^\circ, 0.49^\circ$, and 1.23° . The freestream Mach number is $M = 0.175$. The free stream temperature is $T = 322.22 \text{ K}$. The Reynolds number is $Re_c = 7 \times 10^5$ based on mean aerodynamic chord c . The flight direction is along the negative direction of the x axis. The flight heights considered in the simulation over the flat ground are $h/c = 1, 0.5$ and 0.25 . For the unbounded flow, the flight height h is regarded much larger than c , therefore $h/c = \infty$.

Chapter 3: Numerical Method

In this chapter, the numerical method used for the calculation of aerodynamics properties and flow field is briefly presented. Some important aspects of Computation Fluid Dynamics (CFD) simulation methodology are presented. The computational domain and boundary conditions are provided. The mesh generation is also described briefly.

3.1 Methodology for CFD Simulation

The commercial CFD software ANSYS FLUENT is used as the solver to perform the computations. The steady compressible Reynolds Averaged Navier-Stokes (RANS) equations with pressure-based solver and the Spalart-Allmaras (SA) turbulence model are solved using the finite-volume method. A second-order upwind scheme is used for the convection terms and a central difference scheme is used for the diffusion terms. The SIMPLE coupled algorithm is used for the pressure-velocity coupling.

3.1.1 Turbulence Model

The SA turbulence model is chosen to solve the steady compressible RANS equations after comparison of the SA and $k-\omega$ SST models in the validation of a test case. In ANSYS FLUENT, the transport equation of the SA one-equation turbulence model for the modified turbulent viscosity $\tilde{\nu}$ is given by the following equation:

$$\frac{\partial}{\partial t}(\rho\tilde{\nu}) + \frac{\partial}{\partial x_j}(\rho\tilde{\nu}u_j) = G_\nu + \frac{1}{\sigma_{\tilde{\nu}}} \left\{ \frac{\partial}{\partial x_j} \left[(\mu + \rho\tilde{\nu}) \frac{\partial \tilde{\nu}}{\partial x_j} \right] + C_{b2}\rho \left(\frac{\partial \tilde{\nu}}{\partial x_j} \right)^2 \right\} - Y_\nu$$

where G_ν is the production of turbulent viscosity, and Y_ν is the destruction of turbulent viscosity that occurs in the near-wall region due to wall blocking and viscous damping. $\sigma_{\tilde{\nu}}$ and C_{b2} are the

constants and ν is the molecular kinematic viscosity. The turbulent eddy viscosity is computed from the equation:

$$\mu_t = \rho \tilde{\nu} f_{v1}$$

where the viscous damping function, f_{v1} , is given by

$$f_{v1} = \frac{\chi^3}{\chi^3 + c_{v1}^3}$$

and

$$\chi \equiv \frac{\tilde{\nu}}{\nu}$$

The production term, G_ν , is modeled as

$$G_\nu = C_{b1} \rho \tilde{S} \tilde{\nu}$$

where

$$\tilde{S} \equiv S + \frac{\tilde{\nu}}{\kappa^2 d^2} f_{v2}$$

and

$$f_{v2} = 1 - \frac{\chi}{1 + \chi f_{v1}}$$

C_{b1} and κ are constants, d is the distance from the field point to the nearest wall, and S is a scalar measure of the deformation tensor. S is based on the magnitude of the vorticity:

$$S \equiv \sqrt{2\Omega_{ij}\Omega_{ij}}$$

where Ω_{ij} is the mean rate-of-rotation tensor and is defined by

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

The destruction term is modeled as

$$Y_v = C_{w1} \rho f_w \left(\frac{\tilde{v}}{d} \right)^2$$

where

$$f_w = g \left[\frac{1 + c_{w3}^6}{g^6 + c_{w3}^6} \right]^{1/6}$$

$$g = r + c_{w2}(r^6 - r)$$

$$r \equiv \frac{\tilde{v}}{\bar{S} \kappa^2 d^2}$$

C_{w1} , C_{w2} and C_{w3} are constants.

The model constants C_{b1} , C_{b2} , $\sigma_{\tilde{v}}$, C_{v1} , C_{w1} , C_{w2} , C_{w3} and κ have the following values:

$$C_{b1} = 0.1355, C_{b2} = 0.622, \sigma_{\tilde{v}} = \frac{2}{3}, C_{v1} = 7.1$$

$$C_{w1} = \frac{C_{b1}}{\kappa^2} + \frac{1 + C_{b2}}{\sigma_{\tilde{v}}}, C_{w2} = 0.3, C_{w3} = 2.0, \kappa = 0.4187$$

In ANSYS FLUENT, the Spalart-Allmaras model has been extended with a y^+ -intensive wall treatment, which automatically blends all solution variables from their viscous sublayer formulation

$$\frac{u}{u_\tau} = \frac{\rho u_\tau y}{\mu}$$

to the corresponding logarithmic layer values depending on y^+ .

$$\frac{u}{u_\tau} = \frac{1}{\kappa} \ln E \left(\frac{\rho u_\tau y}{\mu} \right)$$

where u is the velocity parallel to the wall, u_τ is the friction velocity, y is the distance from the wall, $\kappa = 0.4178$ is the von Karman constant, and $E = 9.793$.

3.1.2 Boundary Condition for the Ground

For studying the aerodynamics of the ground effect by numerical simulation, three methods were used in the literature for the ground before a consensus emerged: the image method [10, 12, 23], the stationary-ground method [11, 18, 27], and the moving-ground method [8, 18, 14, 26]. The image method sets the ground plane as a symmetry; the stationary-ground method sets the ground plane as fixed with no-slip wall boundary condition; the moving-ground method sets the ground plane as a moving wall with a velocity same as the free stream. In order to determine the best method for predicting the ground effect accurately, Barber et al. [13] simulated the flow past a NACA 4412 airfoil near a flat ground by solving the RANS equations with RNG $k-\varepsilon$ turbulence model. They concluded that the moving-ground method provides the best answer when compared to the real situation, followed by the second-best method being the image method; the stationary-ground method was found to be entirely inaccurate in simulating the flow field. Hence, in this thesis, the moving ground method is used to simulate the presence of the ground.

3.1.3 Convergence Monitors and Criteria

Two convergence monitors are used to ensure whether the results have converged to reasonable accuracy. For the first convergence monitor, the solution is considered as converged when the absolute residuals of the six governing equations are reduced below 10^{-6} . For the second convergence monitor, the results are accepted as converged when the coefficients of lift, drag and pitching moment achieve a steady state constant value. In the simulations, requirements of the second convergence monitor are easily satisfied, while that of convergence monitor are sometimes difficult to satisfy. The coefficients of lift, drag and pitching moment reach steady state value, while the residual of continuity remains around 10^{-4} and does not change with the growing number of iterations. Under this condition, the solution is treated as converged.

3.2 Computational Domain and Boundary Conditions

To save the computational resources, the half-model of the DLR-F6 is employed in the simulations exploiting symmetry. A rectangular computational domain shown in Figure 3.1 is employed. The inlet boundary is about $140c$ away from the nose of the fuselage; the outlet boundary is about $200c$ away from the tail of the fuselage; the top boundary is about $70c$ away from the highest point of the fuselage; the distance between the lowest point on the fuselage and the bottom boundary is $h = c, 0.5c$ and $0.25c$ for flow in ground effect and $h = 70c$ for unbounded flow; the side boundary is about $140c$ away from the wing tip. The inlet, outlet, side and top boundaries are set as pressure-far-field. The bottom boundary is set as pressure-far-field for unbounded flow and as a moving wall with a velocity V_∞ in x direction for the flow in ground effect. The symmetry plane of DLR-F6 is set as symmetry boundary condition. Surfaces of DLR-F6 are set as no-slip wall boundary conditions.

Figure 3.1 Computational domain of DLR-F6.

3.3 Mesh Generation

Figure 3.2 shows the structured surface grid on the wing-body. A multi-block structured mesh in the flow domain is generated by using the software ANSYS ICEM CFD, which has great capability for structured mesh generation and is widely used in aerospace industry. Computations are performed on a sequence of three grids; a medium size grid is selected based on the grid independence study of the computed solution. The number of hexahedral cells is between 7.5 million and 9.3 million for different ride heights such that the wall distance of the first mesh layer away from the body is always $y^+ < 1$.

Figure 3.2 Surface grid on DLR-F6.

Chapter 4: Validation of Computed Solutions

for Wing-Body in Unbounded Flow

To validate the accuracy of the simulation, a test case for the flow conditions given in the Second AIAA CFD Drag Prediction Workshop (DPW2) is computed and compared with the experimental data [35] and the computations of other participants in the workshop [36]. In Ref. [36], the results from different CFD solvers and different turbulence models are compared and provide the assessment of their accuracy when compared to the experimental data.

4.1 Flow Conditions Used in Validation

In the validation, computations are performed at $M = 0.75$ and $Re_c = 3 \times 10^6$ for wider range of angles of attack including -3° , -2° and -0.304° . Since the flow field is unbounded, the bottom boundary of the computational domain is set as pressure-far-field and is $70c$ away from the lowest point of the fuselage. The SA and $k-\omega$ Shear-Stress Transport (SST) turbulence models are employed in the computations. In addition, the results using the pressure-based and density-based solvers in FLUENT are also compared when employing the SA model.

4.2 Results of the Validation

4.2.1 Aerodynamic Forces

Aerodynamic coefficients of lift, drag and nose-up pitching moment are computed and are shown in Figures 4.1, 4.2 and 4.3, respectively. These figures also include the experimental data and the results from the participants in DPW2. Present results obtained with the SA model and the $k-\omega$ SST model both predict reasonably accurate results when compared to the experimental data and results from the participants of DPW2. For the SA model, both the pressure-based solver and the density-based solver in FLUENT give very close results.

Figure 4.1 Comparison of C_L for DLR-F6

Figure 4.2 Comparison of C_D for DLR-F6

Figure 4.3 Comparison of C_M for DLR-F6

4.2.2 Pressure Distribution

Pressure distributions on eight wing sections are computed using FLUENT with the SA model and are compared with the experimental data in Figure 4.5. Comparisons are made at eight wing sections shown in Figure 4.4, from wing root to wing tip at $y/b = 0.150, 0.239, 0.331, 0.377, 0.411, 0.514, 0.638$ and 0.847 . The computed pressure distributions at an AOA of 0.49° are compared with the experimental data. The experimental data is not available for the pressure distribution on the lower surface at $y/b = 0.847$.

Figure 4.4 Wing section locations.

Figure 4.5 Comparison of computed and experimental pressure distributions at $M = 0.75$ and $\alpha = 0.49^\circ$.

Pressure distributions on eight wing sections are shown and compared with experimental data and the result from FLUENT Inc. At $\alpha = 0.49^\circ$, present CFD simulation is also compared with the result from the FLUENT Inc. participating in DPW2, which shows the simulation presented in this thesis reflects a reasonable result. All the simulation results simulate the pressure distribution on the lower surface perfectly well. The results from all three turbulence models with medium mesh predict the shock locations quite well, with a little difference in the shock strength compared to the experimental data. The pressure distributions on the upper surface right at and after the shock show a little difference between the present CFD simulations and the experiment data. On the upper surface of the wing, the $k-\omega$ SST model always predicts a lower pressure at the shock and the difference with the experimental data is more compared to the SA model. From the computed pressure distributions for DLR-F6 wing-body, it can be concluded that the SA model gives more accurate results compared to the $k-\omega$ SST model. In addition, the pressure-based solver in FLUENT gives the same accuracy as the density-based solver but

requires more CPU time. Although the flow conditions for this validation case are not the same as for the flow in ground effect considered in the next section, the accuracy of the results with higher Mach number and Reynolds number provides a strong evidence that accurate results can be obtained with a pressure based solver and SA model at a lower Mach number and a lower Reynolds number for the computation of the flow field of DLR-F6 in ground effect.

4.2.3 Separation Bubble

Separation appears at the wing root trailing edge of DLR-F6, the separation bubble in the experiment with oil-flow visualizations is shown in Figure 4.6. Geometric parameters of the separation bubble are studied and compared with the results from the participants in DPW2, under the condition of $\alpha = 0.49^\circ$. Unfortunately, the detailed experimental data is not available.

Figure 4.6 Oil-flow visualization of the separation bubble.

Figure 4.7 Geometrical parameters of the separation bubble.

Figure 4.8 Comparison of parameters of separation bubble

Separation bubble has nine geometric parameters measured in the coordinate system consisting of fuselage station (FS), buttock line (BL) and water line (WL), shown in Figure 4.7 [36]. FS indicates the chord-wise position; BL indicates the span-wise position; and WL indicates the vertical position. FS_{BUB} , BL_{BUB} and WL_{BUB} indicate FS at the leading edge, BL measured on the wing at the outboard edge and WL measured on the fuselage at the top edge of the wing root separation bubble, respectively; FS_{EYEW} , BL_{EYEW} and WL_{EYEW} are the position of the center of the separation bubble on the wing; FS_{EYEB} , BL_{EYEB} and WL_{EYEB} are the position of the center of

the separation bubble on the fuselage. Figure 4.8 shows the comparison between the validation results and simulation results of participants in DPW2. In the solutions using the pressure-based solver, the buttock line of the separation bubble cannot be found because of absence of the saddle point on the trailing edge of the wing. Other parameters are predicted in a reasonable range when comparing the present simulation results with those from the participants in DPW2.

Chapter 5: Simulation Results

In this section, the results of simulation of the flow field about DLR-F6 in ground effect are presented. The simulation results are compared with those in the unbounded flow field for $h/c \sim \infty$. The analysis of aerodynamics forces is provided in detail.

5.1 Influence of Ride Height

In the simulation of ground effect, the freestream Mach number is $M = 0.175$, and the Reynolds number is $Re_c = 7 \times 10^5$ based on mean aerodynamic chord c . The angles of attack of 0° , 0.49° and 1.23° are considered. The three flight heights h considered are $h/c = 1$, 0.5 and 0.25 . The flight direction is along the negative direction of the x axis. As an overview, Figure 5.1 and Figure 5.2 show the vorticity contours and streamlines at several cross-sections behind the DLR-F6 wing-body at $\alpha = 1.23^\circ$ with $h/c = \infty$ (Unbounded flow) and 0.5 . Figures show the comparison of the vortex formation behind the trailing edge of the fuselage and its evolution downstream in ground effect (IGE) and out of ground effect (OGE). Figure 5.3 shows the comparison of pressure contours on the surface of DLR-F6 wing-body in unbounded flow and in ground effect at $h/c = 0.5$ at $\alpha = 0.49^\circ$. These comparisons indicate that there exist differences between the unbounded flow field and the flow field in ground effect.

Figure 5.1 Vorticity contours and streamlines on various cross sections behind DLR-F6 IGE ($\alpha = 1.23^\circ$, $h/c = 0.5$).

Figure 5.2 Vorticity contours and streamlines on various cross sections behind DLR-F6 OGE ($\alpha = 1.23^\circ$, $h/c = \infty$).

(a) Pressure contours on the lower surface of DLR-F6.

(b) Pressure contours on the upper surface of DLR-F6.

Figure 5.3 Pressure contours on lower and upper surfaces of DLR-F6 at $\alpha = 0.49^\circ$.

Coefficients of lift (C_L), drag (C_D) and pitching moment (C_M) are the most important aerodynamic parameters, so they are analyzed in detail. Figures 5.4, 5.5 and 5.6 compare the aerodynamic coefficients when h/c decreases from ∞ to 0.25 at three different angles of attack. The total coefficients are divided into the contributions from the wing and fuselage, respectively. Figures 5.7, 5.8 and 5.9 show the corresponding changes in aerodynamic coefficients for the unbounded flow field. They show three main results when the ride height decreases: (1) the lift gain generated on the wing is much larger than the lift change on the fuselage; (2) the drag loss suffered by the wing is much larger than the drag change on the fuselage; and (3) the gain in nose-up pitching moment generated on the fuselage is much larger than the C_M change on the wing. Next, these three effects in ground effect are analyzed on the wing and the fuselage.

Figure 5.4 Variation in C_L with h/c .

Figure 5.5 Variation in C_D with h/c .

Figure 5.6 Variation in C_M with h/c .

Figure 5.7 Variation in ΔC_L with h/c .

Figure 5.8 Variation in ΔC_D with h/c .

Figure 5.9 Variation in C_M with h/c .

5.1.1 Influence of Ride Height on the Lift Coefficient

Pressure distributions on various wing cross sections are analyzed to determine the causes of the lift gain on the wing. The pressure distributions on the wing cross sections are shown in Figures 5.10, 5.12 and 5.14. To get a clearer view of the pressure change on the wing as h/c decreases from ∞ to 0.25, the differences in the pressure distributions in ground effect with respect to those in the unbounded flow field are shown in Figure 5.11, 5.13 and 5.15. From these Figures, the distributions of ΔC_p show similar trend at most cross sections. That is, the pressure on most area of the lower surface of the wing increases and the pressure on most area of the upper surface of the wing decreases. Both effects lead to the lift gain on the wing. For the case at $\alpha = 1.23^\circ$, the pressure distribution on the cross section near the wing root is complex due to the separation bubble, which is discussed later.

Figure 5.10 Comparison of pressure distributions with h/c at eight wing sections at $\alpha = 0^\circ$.

Figure 5.11 Pressure difference with respect to $h/c = \infty$ on eight wing sections at $\alpha = 0.49^\circ$.

Figure 5.12 Comparison of pressure distributions with h/c at eight wing sections at $\alpha = 0.49^\circ$.

Figure 5.13 Pressure difference with respect to $h/c = \infty$ on eight wing sections at $\alpha = 0.49^\circ$.

Figure 5.14 Comparison of pressure distributions with h/c at eight wing sections at $\alpha = 0.49^\circ$.

Figure 5.15 Pressure difference with respect to $h/c = \infty$ on eight wing sections at $\alpha = 0.49^\circ$.

As an example, Figure 5.16 shows the pressure contours in the plane $y/b = 0.239$ in unbounded flow field and in the ground effect. A zoomed view is taken to look at the difference more clearly. The high-pressure region is larger on the lower surface and smaller on the upper surface in ground effect. Figure 5.17 shows the velocity contours under the same condition. The zoomed view shows the high-speed region is smaller on the lower surface and larger on the upper surface in ground effect. This phenomenon can be explained by the blockage effect when the wing moves in the proximity of the ground. The convergent passage between the wing and the ground slows the speed of the flow, hence the flux across the passage. Part of the flux is forced to flow over the upper surface of the wing. This incremental flux results in increase in the velocity on the upper surface. Thus, according to the relationship between the velocity and the pressure, the pressure becomes higher on the lower wing surface and becomes lower on the upper wing surface. Hence, the lift on the wing increases with decreasing ride height. This is a reflection of the positive GE, and the chord-dominated GE. It should also be noted that the pressure difference

between the upper and lower wing surfaces is smaller near the trailing edge of the wing because of the divergent passage. This is a reflection of the chord-dominated GE.

Figure 5.16 Comparison of pressure contours OGE and IGE around a wing section in the plane $y/b = 0.239$.

Figure 5.17 Comparison of velocity contours in OGE and IGE around a wing section in the plane $y/b = 0.239$.

Figure 5.18 Variation in C_l distributions on the fuselage with h/c .

For the fuselage, the distribution of lift coefficient on the fuselage along the stream-wise direction is studied, which is shown in Figure 5.18. To view the change clearly, as the ride height decreases, the difference of the lift coefficient with a baseline case ($h/c = \infty$) is shown in Figure 5.19. When h/c changes from ∞ to 0.25, the distribution of ΔC_l shows that the changes in lift are very small because of the large variations in the pressure distribution on the lower surface of the fuselage. In addition, the order of magnitude of the pressure change on the fuselage (10^{-2}) is one order less than that on the wing (10^{-1}). As a consequence, the lift on the fuselage remains almost the same at all ride heights, compared to the lift gain on the wing.

Figure 5.19 Variation in ΔC_l distributions on the fuselage with h/c .

Table 5.1 Comparison of ΔC_L on the fuselage

α	h/c	$\Delta C_{L,nose}$	$\Delta C_{L,tail}$	$\Delta C_{L,Total}$
0°	1	0.003539	0.001216	0.004755
	0.5	0.004149	-0.00039	0.00376
	0.25	0.003996	-0.00181	0.002186
0.49°	1	0.00398	0.000633	0.004614
	0.5	0.005806	0.000457	0.006263
	0.25	0.006414	-0.00178	0.004636
1.23°	1	0.003476	-0.00291	0.000567
	0.5	0.008774	0.002077	0.010851
	0.25	0.009465	-0.00262	0.00684

When the fuselage is divided into two parts at the moment reference point: the nose part and the tail part, the lift coefficients on the two parts are integral and compared. The nose part always has a positive lift coefficient which is much larger than that on the tail part, as shown in Table

5.1. Also, there exists a very big overshoot near the nose, where the force arm is very long.

These two minor discoveries are very useful when analyzing the pitching moment on the fuselage later.

5.1.2 Influence of Ride Height on the Drag Coefficient

For the drag of the wing, the induced drag is paid a close attention by analyzing the wingtip vortex. Q-criterion is used to extract the vortex axis of the wingtip vortex. Q-criterion is one of the most widely used local vortex-identification criteria. Dallmann [37] and Hunt et al. [38] suggested a vortex definition only based upon the second invariant Q of the velocity gradient tensor. Vortices in an incompressible flow are identified as connected fluid regions with a positive second invariant of $\nabla \mathbf{u}$ (in tensor notation below the subscript comma denotes differentiation)

$$Q \equiv \frac{1}{2}(u_{ii}^2 - u_{ij}u_{ji}) = \frac{1}{2}u_{ij}u_{ji} = \frac{1}{2}(|\boldsymbol{\Omega}|^2 - |\mathbf{S}|^2) > 0$$

where $|\boldsymbol{\Omega}| = \text{trace}(\boldsymbol{\Omega}\boldsymbol{\Omega}^T) = \sum_{i,j=1}^3 \Omega_{ij}\Omega_{ij} = \frac{1}{2}|\boldsymbol{\omega}|^2$ is the Frobenius norm of the vorticity tensor, and $|\mathbf{S}|$ is the Frobenius norm of strain rate tensor. Positive Q defines the regions where the vorticity magnitude prevails over the strain-rate magnitude (rotation dominated shearing which indicates a vortex). In addition, the pressure in the vortex region is lower than the ambient pressure. In most vortices, the fluid flow velocity is greatest next to its axis and decreases in inverse proportion to the distance from the axis. Hence, the vortex axis can be determined by connecting the points with maximum velocity or minimum pressure in the vortex core in the plane perpendicular to this axis. With Q-criterion, the vortex axis can also be obtained by connecting the point with the maximum Q in the vortex core in the plane perpendicular to this axis.

Figure 5.20 shows the trajectory of wingtip vortex in span-wise direction as ride height changes. It shows that the span-wise location of the wingtip vortex core moves towards the wing root all the time in unbounded flow field. However, in ground effect, the wingtip vortex core moves from the wingtip towards the wing root first and then moves far away from the wing root IGE. At all three angles of attack, the trend is the same. This can be explained by introducing the mirror-image model of Mook et al. [39]. In the unbounded flow, the two wingtip vortices induce on each other and move downward. While in ground effect, the wingtip vortex is affected by the image circulation whose function is to induce up-wash. The wingtip vortex cores are prevented from moving towards each other in the near-field; they even separate from each other in the farther distance along the flow direction. The relative distance between the wingtip vortex and the wing increases. As a result, the induced velocity caused by the circulation due to wingtip vortex is reduced according to the Biot-Savart Law:

$$d\mathbf{V} = \frac{\Gamma}{4\pi} \frac{d\mathbf{l} \times \mathbf{r}}{|\mathbf{r}|^3}$$

According to Prandtl's lifting line theory, the induced drag decreases when the induced velocity decreases. Thus, the induced drag is also reduced on the wing. This is a consequence of span dominated ground effect. As the ride height h/c decreases from 1 to 0.25, the effect of mirror-image model is more obvious since the image circulation is getting closer. The wingtip vortex is prevented from moving towards fuselage more strongly and moves outwards faster. Hence, the induced drag decreases with decreasing ride height.

Figure 5.20 Span-wise trajectory of the wingtip vortex for various h/c

5.1.3 Influence of Ride Height on the Pitching Moment Coefficient

Finally, as far as the pitching moment is concerned, the pressure distributions on the wing are analyzed again. As is mentioned previously, the blockage effect explains the lift gain on the wing. However, the reduced flux in the passage between the wing and the ground can escape from the side of the wing. Less flux is forced to flow over the upper surface of the wing. This means that the gain in lift is mainly generated on wing sections away from the wingtip. If the wing is divided into two parts in span wise direction at the moment reference point shown in Figure 4.4, the part located in front of the moment center, which is away from the wingtip, has larger gain in C_L and smaller span area while the other part (the sections closer to wing tip and trailing edge) has smaller gain in C_L and larger span area. As a result, the moment coefficient changes very little. This is another consequence of span dominated ground effect.

As to the pitching moment of the fuselage, the distribution of C_m is analyzed on the cross sections of fuselage along the stream-wise direction, shown in Figure 5.21. The change in pitching moment coefficient, ΔC_m , based on the case OGE is calculated and compared in Figure 5.22. Again, the fuselage is divided into two parts, the nose part and the tail part at the reference

moment center and the integration of ΔC_m is calculated on the two parts, as shown in Table 5.1.

From this table, $\Delta C_{M,nose}$ is always positive (nose-up) and much larger than $\Delta C_{M,tail}$. Therefore, the gain of pitching moment on the fuselage mainly comes from the nose part.

Figure 5.21 Variation in C_m distributions on the fuselage with h/c .

Figure 5.22 Variation in ΔC_m distributions on the fuselage with h/c .

Table 5.2 Comparison of ΔC_M on the fuselage

α	h/c	$\Delta C_{M,nose}$	$\Delta C_{M,tale}$	$\Delta C_{M,Total}$
0°	1	0.005886	-0.001621	0.004266
	0.5	0.007904	8.568E-5	0.007989
	0.25	0.008145	6.353E-4	0.008780
0.49°	1	0.006455	-3.716E-4	0.006083
	0.5	0.009554	-7.075E-4	0.008846
	0.25	0.01067	0.001318	0.01199
1.23°	1	0.006263	0.006133	0.01240
	0.5	0.01217	-9.383E-4	0.01123
	0.25	0.01359	0.005025	0.01861

In Figure 5.22, it can be noticed that there exists a large overshoot near the fuselage nose at all ride heights and angles of attack. Its magnitude increases with the decreasing ride height and its surrounding part has a major contribution on the lift gain in the nose part. The pressure distribution and contours in the plane of this cross section are analyzed in detail. Figures 5.23, 5.24 and 5.25 show the pressure contours in the plane of the cross section. As the ride height decreases, the pressure increases dramatically below the fuselage. The pressure distribution along the horizontal direction on the fuselage is shown in Figure 5.26. The change between the IGE and OGE cases is calculated and is shown in Figure 5.27. With decreasing ride height, the pressure increases markedly on the lower surface of the cross section and decreases slightly on the upper surface. Therefore, the upward force increases in this cross section, which has a very long force arm. Therefore, the nose-up pitching moment increases correspondingly.

Figure 5.23 Pressure contours in the plane of overshoot plane at $\alpha = 0^\circ$.

Figure 5.24 Pressure contours in the plane of overshoot plane at $\alpha = 0.49^\circ$.

Figure 5.25 Pressure contours in the plane of overshoot plane at $\alpha = 1.23^\circ$.

Figure 5.26 Pressure distribution on the surface of fuselage in the plane of overshoot

Figure 5.27 ΔC_p distribution on the surface of fuselage in the plane of overshoot

5.1.4 Influence of Ride Height on the Separation Bubble

The separation bubbles in ground effect are compared with each other and that in unbounded flow. Figure 5.28 shows the 3-D streamlines close to the upper surface of the wing near the wing root at $\alpha = 0.49^\circ, h/c = 0.25$. The fluid elements move parallel from the leading edge of the wing. Near the trailing edge, the elements near the wing root start to separate and move away from the surface. The ones closest to the wing root are drawn to and coil in the separation

bubble. Behind the separation bubble, the affected area grows and all elements start to be involved in the wake flow.

Figure 5.28 3-D streamlines close to the upper surface of the wing near the wing root at $\alpha = 0.49^\circ$, $h/c = 0.25$.

The separation bubble can be shown with the surface streamlines on the wing and fuselage at the corner of wing root and trailing edge. In Figures 5.29, 5.30 and 5.31, the separation bubbles are compared at various ride height h/c . In Figures 5.29 and 5.30, the shape of the separation bubble changes little when the ride height decreases. However, in Figure 5.31, the size of the separation area changes with different ride heights. Big difference in the pressure difference exists in this area, e.g. the pressure distribution on the wing at the cross section $y/b = 0.15$ (Figure 5.15). Considering this phenomenon with irregular trend in the aerodynamic forces at $\alpha = 1.23^\circ$, the separation bubble is perhaps the reason. However, more advanced analysis is needed to support this point of view.

(a) OGE

(b) $h/c = 1.00$

(c) $h/c = 0.50$

(d) $h/c = 0.25$

Figure 5.29 Surface streamlines on the wing and fuselage at the corner of wing root and trailing edge at $\alpha = 0^\circ$.

(a) OGE

(b) $h/c = 1.00$

(c) $h/c = 0.50$

(d) $h/c = 0.25$

Figure 5.30 Surface streamlines on the wing and fuselage at the corner of wing root and trailing edge at $\alpha = 0.49^\circ$.

(a) OGE

(b) $h/c = 1.00$

(c) $h/c = 0.50$

(d) $h/c = 0.25$

Figure 5.31 Surface streamlines on the wing and fuselage at the corner of wing root and trailing edge at $\alpha = 1.23^\circ$.

5.2 Influence of Angle of Attack in Ground Effect

Generally speaking, in unbounded flow field, when AOA increases from low to moderate angle, lift and drag coefficients increase; pitching moment coefficient changes but not very much. To determine this conclusion is also valid in ground effect, Figures 5.32, 5.33 and 5.34 compare the aerodynamic coefficients when α changes. In ground effect, when α increases from 0° to 1.23° , the total lift increases since the lift generated on the wing and the fuselage both increase, however mainly due to the wing (Figure 5.32); the total drag increases because the drag generated by the wing and the fuselage both increase (Figure 5.33); and the nose up pitching moment increases due to a larger increase due to the fuselage and a smaller increase due to the wing (Figure 5.34). Except for the total pitching moment, all coefficients have the same trend in unbounded flow.

Figure 5.32 Variation in C_L with α .

Figure 5.33 Variation in C_D with α .

Figure 5.34 Variation in C_M with α .

5.2.1 Influence of AOA on Lift Coefficient in Ground Effect

When h/c is fixed and α increases from 0° to 1.23° , the lift on both the wing and the fuselage increases. The convergent passage under the wing becomes longer and has an increasing slope and the divergent passage under the wing becomes shorter and has a reduced slope. Thus the blockage effect is strengthened and the pressure increases on the lower surface and decreases on the upper surface with increase in α as shown in Figure 5.35 and Figure 5.36.

Figure 5.35 Pressure distribution on eight wing sections at $h/c = 1$.

Figure 5.36 Pressure difference with respect to $\alpha = 0^\circ$ on eight wing sections at $h/c = 1$.

For lift coefficient on the fuselage, Figure 5.37 compares C_l distributions along stream-wise direction on the fuselage and the corresponding change based on the case of $\alpha = 0^\circ$ is shown in Figure 5.38. In unbounded flow, when the angle of attack increases from 0 to 1.23° , the lift coefficient experiences a large gain on most cross sections of the fuselage and a small decrease on the tail part (Figure 5.38). However, in GE, the lift coefficient oscillates with a “phase shift” and larger magnitude when AOA increases (Figure 5.37). As a consequence, lift on the fuselage increases in both conditions.

Figure 5.37 Variation in C_l distributions along stream-wise direction on the fuselage with AOA.

Figure 5.38 Variation in ΔC_l distributions along stream-wise direction on the fuselage with AOA.

5.2.2 Influence of AOA on Drag Coefficient in Ground Effect

The drag coefficient on both the wing and the fuselage also increases. The drag consists of induced drag and profile drag for the wing. The profile drag generated by the wing as well as the fuselage both increase because the projection area in flow direction increases. The induced drag on the wing is proportional to the square of the lift coefficient, according to the lifting line theory of incompressible flow on finite wings:

$$C_{D,i} = \frac{C_L^2}{\pi AR}$$

Since the lift increases with the AOA, the induced drag of the wing also increases. In case of fuselage, the drag is dominated by its profile drag, which increases with the AOA. This conclusion is also the same as that for the unbounded flow field.

5.2.3 Influence of AOA on Pitching Moment Coefficient in Ground Effect

Figure 5.39 Variation in C_m distributions along stream-wise direction on the fuselage with AOA.

As to the pitching moment on the fuselage, the C_m distribution along the stream-wise direction with variation of AOA is shown in Figure 5.39. The corresponding change based on $\alpha = 0^\circ$ is calculated and compared in Figure 5.40. When AOA increases in unbounded flow, the pitching moment levels up entirely in front of the moment reference point. While in GE, the ΔC_m in the nose part oscillates dramatically, because there is “phase shift” with growing magnitude in C_m

distribution. In both conditions, the nose up pitching moment on the fuselage increases when α increases from 0 to 1.23° .

Figure 5.40 Variation in ΔC_m distributions along stream-wise direction on the fuselage with AOA.

Chapter 6: Conclusions

The Spalart-Allmaras (SA) model with a pressure based solver in ANSYS FLUENT has been used to validate the simulation of flow past a 3-D DLR-F6 wing-body in transonic flow by comparing the computations with the computations of other investigators and the experimental data as reported in the summary of the Second AIAA CFD Drag Prediction Workshop (DPW2). The validated code is then used to compute the flow past the DLR-F6 wing-body in ground effect at low subsonic Mach numbers and Reynolds numbers at various heights above the ground at different angles of attack.

When the ride height h/c decreases from ∞ to 0.25 with a fixed angle of attack, the wing-body experiences (1) a gain in lift that is generated mainly on the wing; (2) a loss in drag that is mainly because the induced drag on the wing decreases; and (3) a gain in nose-up pitching moment that is generated mainly on the fuselage. In addition, in ground effect, the change in lift is dominated by the wing primarily due to the chord dominated ground effect. The change in drag is dominated by the wing primarily because of the span dominated ground effect. The change in nose-up pitching moment is dominated by the fuselage.

When angle of attack increases from 0° to 1.23° , the wing-body experiences (1) a gain in lift that is generated mainly on the wing; (2) a gain in drag that is contributed by both the wing and the fuselage; (3) a gain in nose-up pitching moment because of a larger gain from the fuselage and a smaller loss from the wing. Although the first two conclusions are similar to that for the unbounded flow field, however the third conclusion about the pithing moment is different in ground effect compared to unbounded flow field.

Chapter 7: Future Work

Based on the simulations of the ground effect of DLR-F6 wing-body reported in this thesis, it can be concluded that the CFD methodology employed in this thesis can be applied to study the aerodynamics of wing-bodies in more complex geometries in ground effect, which is a combination of a fuselage and a three-element wing. The study of DLR-F11 in ground effect can be very helpful in understanding the process of landing and takeoff of an actual transport aircraft.

References

- [1] Qu, Q. L., Lu, Z., Liu, P. Q., and Agarwal, R. K., “Numerical Study of Aerodynamics of a Wing-in-Ground-Effect Craft,” *Journal of Aircraft*, Vol. 51, No. 3, 2014, pp. 913–924, doi:10.2514/1.C032531
- [2] Rozhdestvensky, K. V., “Wing-in-Ground Effect Vehicles,” *Progress in Aerospace Sciences*, Vol. 42, No. 3, 2006, pp. 211–283, doi:10.1016/j.paerosci.2006.10.001
- [3] Rozhdestvensky, K. V., *Aerodynamics of a Lifting System in Extreme Ground Effect*, Springer–Verlag, Berlin, 2000, Chap. 1.
- [4] Halloran, M., and O’Meara, S., “Wing in Ground Effect Craft Review,” Australia DSTO Aeronautical and Maritime Research Lab., Rept. DSTO-GD-0201, Melbourne, Victoria, Australia, Feb. 1999.
- [5] Lange, R. H., and Moore, J. W., “Large Wing-in-Ground Effect Transport Aircraft,” *Journal of Aircraft*, Vol. 17, No. 4, 1980, pp. 260–266, doi:10.2514/3.57898
- [6] Ahmed, M. R., “Aerodynamics of a Cambered Airfoil in Ground Effect,” *International Journal of Fluid Mechanics Research*, Vol. 32, No. 2, 2005, pp. 157–183, doi:10.1615/InterJFluidMechRes.v32.i2.30
- [7] Abramowski, T., “Numerical Investigation of Airfoil in Ground Proximity,” *Journal of Theoretical and Applied Mechanics*, Vol. 45, No. 2, 2007, pp. 425–436.
- [8] Hayashi, M., and Endo, E., “Measurement of Flow Fields Around an Airfoil Section with Separation,” *Japan Society for Aeronautical and Space Science Transactions*, Vol. 21, No. 52, 1978, pp. 69–75.
- [9] Agarwal, R. K., and Deese, J. E., “Numerical Solutions of the Euler Equations for Flow Past an Airfoil in Ground Effect,” 22nd AIAA Aerospace Sciences Meeting, AIAA Paper 1984-0051, Jan. 1984.
- [10] Coulliette, C., and Plotkin, A., “Aerofoil Ground Effect Revisited,” *Aeronautical Journal*, Vol. 100, No. 992, 1996, pp. 65–74.
- [11] Hsiun, C.-M., and Chen, C.-K., “Aerodynamic Characteristics of a Two-Dimensional Airfoil with Ground Effect,” *Journal of Aircraft*, Vol. 33, No. 2, 1996, pp. 369–392, doi:10.2514/3.46949
- [12] Hiemcke, C., “NACA 5312 in Ground Effect: Wind Tunnel and Panel Code Studies,” AIAA Paper 1997-2320, 15th Applied Aerodynamics Conference, June 1997.
- [13] Barber, T. J., Leonardi, E., and Archer, R. D., “A Technical Note on the Appropriate CFD Boundary Conditions for the Prediction of Ground Effect Aerodynamics,” *Aeronautical Journal*, Vol. 103, No. 1029, 1999, pp. 545–547.
- [14] Ahmed, M. R., Takasaki, T., and Kohama, Y., “Aerodynamics of a NACA 4412 Airfoil in Ground Effect,” *AIAA Journal*, Vol. 45, No. 1, 2007, pp. 37–47, doi:10.2514/1.23872
- [15] Hsiun, C. M., and Chen, C. K., “Numerical Investigation of the Thickness and Camber Effects on Aerodynamic Characteristics for Two-Dimensional Airfoils with Ground Effect in Viscous Flow,” *Trans. Japan Soc. Aero. Space Sci.*, Vol. 38, 1994, pp. 78-90.
- [16] Qu, Q. L., Wang, W., Liu, P. Q., and Agarwal, R. K., “Airfoil Aerodynamics in Ground Effect for Wide Range of Angles of Attack,” *AIAA Journal*, Vol. 53, No. 4, 2015, pp. 1048-1061.

- [17] Ahmed, M. R., and Sharma, S. D., "An Investigation on the Aerodynamics of a Symmetrical Airfoil in Ground Effect," *Experimental Thermal and Fluid Science*, Vol. 29, 2005, pp. 633-647.
- [18] Zerihan, J., and Zhang, X., "Aerodynamics of a Single Element Wing in Ground Effect," *Journal of Aircraft*, Vol. 37, No 6, 2000, pp. 1058-1064.
- [19] Mahon, S., and Zhang, X., "Computational Analysis of Pressure and Wake Characteristics of an Aerofoil in Ground Effect," *Journal of Fluids Engineering*, Vol. 127, No. 2, 2005, pp. 290-298.
- [20] Recant, I. G., "Wind-tunnel Investigation of Ground Effect on Wings with Flaps," NASATN-D-705, 1939.
- [21] Yang, W., Lin F., and Yang Z., "Investigation on Application of High-Lift Configuration to Wing-in-Ground Effect," *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*, Vol. 226, No.3, 2012, pp. 260-271.
- [22] Qu, Q. L., Wang, W., Liu, P. Q., and Agarwal, R. K., "Aerodynamics and Flow Mechanics of a Two-Element Airfoil in Ground Effect," *AIAA Paper 2015-0550*, 2010.
- [23]. Gratzler, L. B., and Mahal, A. S., "Ground Effects in STOL Operation," *Journal of Aircraft*, Vol. 19, No. 3, 1971, pp. 236-242.
- [24] Qu, Q, Ju, B., Huang, L., Liu, P., and Agarwal, R.K., "Flow Physics of a Multi-Element Airfoil in Ground Effect." in *54th AIAA Aerospace Sciences Meeting*, p. 0856, 2016.
- [25] Wieselsberger, C., "Wing Resistance near the Ground," *NACA TM No. 77*, 1922.
- [26] Moon, Y. J., Oh, H. J., and Seo, J. H., "Aerodynamic Investigation of Three-Dimensional Wings in Ground Effect for Aero-Levitation Electric Vehicle," *Aerospace Science and Technology*, Vol. 9, No. 6, 2005, pp. 485-494.
- [27] Chawla, M., Edwards, L., and Franke, M., "Wind-Tunnel Investigation of Wing-in-Ground Effects," *Journal of Aircraft*, Vol. 27, No. 4, 1990, pp. 289-293.
- [28] Cho, J., and Han, C., "Unsteady Trailing Vortex Evolution behind a Wing in Ground Effect," *Journal of Aircraft*, Vol. 42, No. 2, 2005, pp. 429-434.
- [29] Lee, T.W., "Experimental Study of Ground Effect on Finite-Wing Wake Flow Development," M.S. Thesis, Aeronautics and Astronautics Dept., National Cheng Kung University, 2002 (in Chinese).
- [30] Harvey, J., and Perry, F. J., "Flow field Produced by Trailing Vortices in the Vicinity of the Ground," *AIAA Journal*, Vol. 9, No. 8, 1971, pp. 1659-1660.
- [31] Qu, Q., Huang, L., Liu, P., & Agarwal, R. K., "Near-field Wingtip Vortex Characteristics of a Rectangular Wing in Ground Effect," in *54th AIAA Aerospace Sciences Meeting*, 2016.
- [32] Wang, Q. X. "Analyses of a Slender Body Moving Near a Curved Ground." *Physics of Fluids* 17, No. 9, 2005, p. 097102.
- [33] Airplanes, Commercial. "Statistical Summary of Commercial Jet Airplane Accidents." *Worldwide Operations 2008 (1959)*.

- [34] Rossow, C. C., Godard, J. L., Hoheisel, H., and Schmitt, V., "Investigations of Propulsion Integration Interference Effects on a Transport Aircraft Configuration," *Journal of Aircraft*, Vol. 31, No. 5 1994, pp. 1022-1030, doi: 10.2514/3.46605
- [35] Laflin, K. R., Klausmeyer, S. M., Zickuhr, T., Vassberg, J. C., Wahls, R. A., Morrison, J. H., Brodersen, O. P., Rakowitz, M. E., Tinoco, E., and Godard, J.-L., "Data Summary from Second AIAA Computational Fluid Dynamics Drag Prediction Workshop," *Journal of Aircraft*, Vol. 42, No. 5, 2005, pp. 1165-1178.
- [36] Second AIAA CFD Drag Prediction Workshop. <http://aiaa-dpw.larc.nasa.gov/Workshop2/participants.html>, dpw@cessna.textron.com, Orlando, FL, June 2003.
- [37] Dallmann, U. "Analysis of Simulations of Topologically Changing Three-Dimensional Separated Flows." In *Separated Flows and Jets*, pp. 1-14, Springer Berlin Heidelberg, 1991.
- [38] Hunt, J. C. R., A. A. Wray, and P. Moin. "Eddies, streams, and convergence zones in turbulent flows." In *Studying Turbulence Using Numerical Simulation Databases*, 2. 1988.
- [39] Mook, D., and Nuhait, A., "Numerical Simulation of Wings in Steady and Unsteady Ground Effects," *Journal of Aircraft*, Vol. 26, No. 12, 1989, pp. 1081–1089, doi:10.2514/3.45884

Vita

Ning Deng

- Degrees** M.S Mechanical Engineering, Washington University in St. Louis, United States, Jul 2017
- B.S. Aircraft Design and Engineering, Beihang University, China, June 2013
- Birth of Place** Shaoyang, Hunan, China
- Publications** Deng, Ning, Qiulin Qu, and Ramesh K. Agarwal. "Numerical Study of the Aerodynamics of DLR-F6 Wing-Body in Unbounded Flow Field and in Ground Effect." In 55th AIAA Aerospace Sciences Meeting, p. 1424. 2017.
- Deng, Ning, Qiulin Qu, and Ramesh K. Agarwal. "Numerical Study of Fuselage Aerodynamics of DLR-F6 Wing-Body in Ground Effect." In 35th AIAA Applied Aerodynamics Conference, p. 4234. 2017.