Studies of aerodynamics of supersonic generic round shaped bodies

Wanchai, Jiajan

2014

Wanchai, J. (2014). Studies of aerodynamics of supersonic generic round shaped bodies.
Doctoral thesis, Nanyang Technological University, Singapore.

<https://hdl.handle.net/10356/58905>

<https://doi.org/10.32657/10356/58905>

**NANYANG
TECHNOLOGICAL
UNIVERSITY**

**STUDIES OF AERODYNAMICS OF SUPERSONIC
GENERIC ROUND SHAPED BODIES**

WANCHAI JIAJAN

SCHOOL OF MECHANICAL AND AEROSPACE ENGINEERING

2014

**STUDIES OF AERODYNAMICS OF SUPERSONIC
GENERIC ROUND SHAPED BODIES**

WANCHAI J.

2014

**STUDIES OF AERODYNAMICS OF SUPERSONIC
GENERIC ROUND SHAPED BODIES**

WANCHAI J.

WANCHAI JIAJAN

SCHOOL OF MECHANICAL AND AEROSPACE ENGINEERING

A thesis submitted to the Nanyang Technological University
in partial fulfillment of the requirement for the degree of
Doctor of Philosophy

2014

Acknowledgments

First of all, I would like to thank my supervisor, Dr Randy Shek-Ming Chue for his excellent guidance and generous support during the course of my graduate study. His phenomenal knowledge and experience in this field of studies have enabled me to gain valuable insights and lessons that are applicable in my future endeavors. I am also grateful to Prof. Simon Ching-Man Yu, my co-supervisor for continuously providing me encouragement, guidance and support in many ways.

This research is supported by DSO National Laboratories of Singapore under contract DSOCL10329. I gratefully thank DSO for their support and helpful discussions. I would further like to thank Siauwei Long for his help on my wind tunnel experiment measurement system setup. I appreciate the comments and valuable suggestions for the instrumentation, in sharing his idea through the experiment period.

A Special thanks to Dr. Tue Nguyen who has worked with me for the wind tunnel testing period and provided invaluable advice and support with FLUENT. I have also gained much from his enthusiasm, valuable discussions, and friendship.

I would also like to thank my friends and colleagues for their friendship, discussions, collaboration and assistance in this research.

Finally, I would like to express my deepest love and appreciation to my wife Kanokwan, who has been solidly behind me during all phases of my life.

Contents

Acknowledgments.....	i
Contents	ii
List of Tables	vi
List of Figures	vii
List of Nomenclature and Symbols.....	xiv
Abstract.....	xvi
Chapter 1 Introduction	1
1.1 Background and Motivation	1
1.2 Objective of Study	4
1.3 Thesis Contributions.....	5
1.4 Thesis Outline.....	7
Chapter 2 Literature Review	9
2.1 Background.....	9
2.2 Aerodynamic Characteristics.....	10
2.2.1 Static Aerodynamic Coefficients.....	10
2.2.2 Dynamic Aerodynamic Coefficients	14
2.3 Flight Stability of Generic Rounds	18
2.4 Aerodynamic Prediction	21
2.4.1 Theoretical Method	21
2.4.2 Computational Fluid Dynamics	24
2.4.3 Wind Tunnel Testing	26
2.4.4 Shock waves Boundary Layer Interaction (SWBLI).....	29
2.5 Aerodynamic Performance Improvement	33
Chapter 3 Methodology and Validation Study	35
3.1 Governing Equations	35

3.2 Theoretical Work	37
3.3 Computational Approach	40
3.4 Numerical Procedures	42
3.4.1 Grid Generation	42
3.4.1 Boundary Conditions	43
3.5 Validation Study	45
3.5.1 Computational Benchmark Models	45
3.5.2 Grid Convergence Study	48
3.5.3 Pressure Distribution Comparisons.....	49
3.5.4 Static Aerodynamic Coefficients	51
3.5.5 Dynamic Aerodynamic Coefficients.....	54
Chapter 4 Aerodynamic Performance Improvement.....	61
4.1 Optimization Procedure	61
4.1.1 Initial Design.....	63
4.1.2 Drag Computation based on Semi-Empirical Method.....	63
4.1.2 Optimization Technique.....	64
4.1.3 Stability Analysis	66
4.2 Optimization Demonstration.....	68
4.2.1 Benchmark Configurations and Design Considerations	68
4.2.2 Drag Minimization.....	68
4.2.3 Comparison with the Previous Optimization Technique	74
4.2.4 Stability Analysis	75
4.2.5 Stability Improvement	77
4.3 Effect of Axially Symmetric Flow	86
4.4 Boattail Juncture Shaping Technique	94
4.4.1 Computational Benchmark Models	95
4.4.1 Effect of Boattail Juncture Shaping on Aerodynamic Drag	97

4.4.1 Effect of Boattail Juncture Shaping on Gyroscopic Stability.....	100
4.4.1 Effect of Boattail Juncture Shaping on Dynamic Stability	103
Chapter 5 Wind Tunnel Testing.....	108
5.1 Overview of Experiment	109
5.2 Wind Tunnel Facility and Instrumentation.....	109
5.3 Measurement Parameters.....	111
5.4 Experimental Setup and Considerations.....	112
5.4.1 Test Conditions.....	112
5.4.2 Balance Load Limits.....	113
5.4.3 Model Scaling (Model Design)	114
5.4.4 Test Section Considerations	116
5.4.5 Structural Analysis	116
5.5 Model and Balance Installation	117
5.5.1 Model Assembly.....	117
5.5.2 Pressure Transducers Installation	119
5.5.3 Moment Center of Balance.....	121
5.6 Test Conditions and Procedures	122
5.7 Experimental Uncertainties	122
5.8 Results and Discussions	125
5.8.1 Static Aerodynamic Coefficients.....	125
5.8.2 Effects of Reynolds Number on Aerodynamic Characteristics.....	138
5.8.3 Flow Visualization.....	140
Chapter 6 Conclusions and Future Work.....	143
6.1 Summary.....	143
6.1.1 Aerodynamic Study and Prediction.....	143
6.1.2 Aerodynamic Performance Improvement	144
6.1.2 Aerodynamic Testing	146

6.2 Future Work	147
APPENDIX A	149
APPENDIX B.....	154
APPENDIX C.....	160
APPENDIX D	170
References	176

List of Tables

Table 3.1: Wind tunnel testing and aeroballistics flight conditions of five-different projectiles.....	48
Table 3.2: Percent differences of aerodynamic coefficients of 155 mm M549 round from the medium-sized mesh in grid resolution study.....	49
Table 4.1: Breakdown of pressure drag coefficient on the optimum and benchmark rounds.....	73
Table 4.2: Physical characteristics of the modified optimum 155 mm flight bodies ..	78
Table 4.3: The physical properties of three-different generic rounds.....	96
Table 4.4: Breakdown of pressure drag coefficient on the two different rounds	98
Table 4.5: Breakdown of normal force coefficient on the optimum and benchmark rounds.....	103

List of Figures

Figure 1.1: The standard 155 mm spin-stabilized rounds: 155 mm M549 (left) and 155 mm M107 (right) [46].	2
Figure 2.1: The configurations of typical generic rounds	9
Figure 2.2: The round body under statically unstable condition	12
Figure 2.3: The demonstration of overturning moment induces the precession	13
Figure 2.4: The demonstration of spin-induced boundary layer distortion	14
Figure 2.5: The steady coning motion of a spin-stabilized round	17
Figure 2.6: The stability criterion bound of spin-stabilized rounds	21
Figure 2.7: The four patterns of SWBLI [76]: a) weak interaction, b) medium interaction, c) strong interaction, and d) very strong interaction.	30
Figure 2.8: The experimental data of projectile at transonic speed (Mach 0.94) [75]: a) a shadowgraph and b) pressure distribution along round body	31
Figure 3.1: The flow chart of McCoy's semi-empirical codes	39
Figure 3.2: The examples of 2-D and 3-D model meshes a) 2-D axisymmetric mesh for zero-lift drag coefficient and b) full 3-D mesh for spinning conditions.	43
Figure 3.3: The spin direction and circumferential angle along the body surface	44
Figure 3.4: An illustration of a coning motion simulation	45
Figure 3.5: The configuration of generic rounds for pressure distributions measurement, a) Secant-Ogive Cylinder (SOC) and b) Secant-Ogive Cylinder Boat Tail (SOCBT), dimensions in body diameter ($D=57.15\text{mm}$)	46
Figure 3.6: The configuration of generic rounds for forces and moments calculations, a) M910 sub caliber, dimensions in cm; b) Army-Navy Spinner Rocket (ANSR) ($l=7D$)	

body diameter, $D=20\text{mm}$), dimensions in body diameter; c) 155 mm M549 projectile, dimensions in body diameter ($D=155\text{mm}$).....	46
Figure 3.7: The 3-D model meshes for pressure distributions measurement: a) Secant-Ogive Cylinder (SOC) and b) Secant-Ogive Cylinder Boat Tail (SOCBT).....	47
Figure 3.8: The 2-D and 3-D model meshes for forces and moments calculations a) M910 full 3-D mesh, b) ANSR full 3-D for spinning, c) 2-D axisymmetric mesh with rotating band, one-half 3-D mesh (non-spinning), and full 3-D mesh (spinning) for 155 mm M549 standard round.....	47
Figure 3.9: The comparisons of surface pressure distributions of SOC using CFD and wind tunnel test data (Mach 3, 4.2° angle of attack).....	50
Figure 3.10: The comparisons of surface pressure distributions of SOCBT using CFD and wind tunnel test data [63] (Mach 3, 10.4° angle of attack)	51
Figure 3.11: The static aerodynamic coefficients for ANSR (non-spinning and spinning) at 2° angle of attack versus Mach number, a) drag coefficient, b) normal force derivative, c) pitching moment derivative, d) center of pressure	56
Figure 3.12: The static aerodynamic coefficients for M910 sub caliber (spinning) at 3° angle of attack versus Mach number, a) drag coefficient, b) normal force derivative, c) pitching moment derivative, d) center of pressure.....	57
Figure 3.13: The static aerodynamic coefficients for 155 mm M549 projectile (non-spinning and spinning) at 2° angle of attack versus Mach number, a) drag coefficient, b) normal force derivative, c) pitching moment derivative, d) center of pressure	58
Figure 3.14: The dynamic aerodynamic coefficients for spinning versus Mach number, a) Roll damping moment derivative for ANSR, b) Magnus moment derivative for ANSR, c) Roll damping moment derivative for M910, d) Magnus moment derivative for M910	59

Figure 3.15: The dynamic aerodynamic coefficients for 155 mm M549 round: a) Magnus moment derivative and b) pitch damping moment sum coefficient at 2° angle of attack.	60
Figure 4.1: The flow chart of the optimization procedure.....	62
Figure 4.2: (a) Drag coefficient of optimal round configurations at various Mach numbers. (b) Comparison of drag coefficient on optimum 155mm round and fixed optimum geometry using the optimization method. Total length = $5.645D$	70
Figure 4.3: The optimum 155 mm round. The nose shape is based on Newtonian impact theory (total length $5.645D$). C.G. is the center of gravity.....	70
Figure 4.4: Comparison of the drag coefficient of the benchmark and optimum 155 mm rounds.	71
Figure 4.5: Mach contours and pressure distributions over two-different flight bodies (Mach 2 and $\alpha = 0^\circ$). a) Optimum 155mm round and b) benchmark round.	73
Figure 4.6: Comparison of predicted drag using the present optimization method, Nicolas' optimization technique [49], and CFD. Optimum shape with body length of $5D$	75
Figure 4.7: Comparison of the dynamic stability factor between the optimum shape and the benchmark round, $PD/V = 0.314$	76
Figure 4.8: Mach contours and pressure distributions on the optimum body at Mach 2 and $\alpha = 2^\circ$	77
Figure 4.9: The physical characteristics of the modified optimum 155 mm flight bodies.	78
Figure 4.10: Comparison of the drag coefficient of the modified generic rounds.	79
Figure 4.11: Comparison of the normal force and pitching moment derivatives on the modified generic rounds.	81

Figure 4.12: Comparison of the gyroscopic stability factor on the modified generic rounds, $PD/V = 0.314$	82
Figure 4.13: Comparison of the dynamic aerodynamic coefficients on the modified generic rounds, a) pitch damping moment coefficient ($C_{Mq} + C_{M\dot{\alpha}}$) and b) Magnus moment derivative, $PD/V = 0.314$	84
Figure 4.14: Comparison of the dynamic stability factor on the modified generic rounds, $PD/V = 0.314$	85
Figure 4.15: The comparison of boundary layer velocity profiles over the modified optimum 155mm round (1.5D) using two different simulations: Steady and Unsteady simulations (at zero AOA, Mach 3): a) $X/D=4.95$ and b) $X/D= 5.6$	87
Figure 4.16: The comparison of boundary layer velocity profiles over the round bodies between SOC and SOCBT projectile at three different locations (at angle of attack 4.3° , Mach 3): a) $X/D=4.9$, b) $X/D= 5.6$, and c) $X/D=5.95$	88
Figure 4.17: The formation of weak embedded shocks over the SOCBT projectile at angle of attack of 10.4° (Mach 3).	89
Figure 4.18: Two different bodies: SOCBT projectile (left) and SOCBT 2D symmetric body (right)	89
Figure 4.19: the comparison of pressure distributions along two different bodies (zero AOA): a) SOCBT projectile (round body) and b) 2-D symmetric body	90
Figure 4.20: The comparison of the flow over the boattail configuration of two different bodies using the same boattail angle 7° (Mach 2, zero angle of attack): a) round body and b) 2D symmetric body.	91
Figure 4.21: Streamline coordinates for axially symmetric flow.	92
Figure 4.22: Mach number contours and pressure variation along two different streamlines. Simulation of optimum shape at Mach 2 and $\alpha = 0^\circ$	93

Figure 4.23: The configuration of modified SOCBT rounds, a) SOCBT (MOD1), b) SOCBT (MOD2), c) SOCBT (MOD3), and d) SOCBT (MOD4), dimensions in body diameter ($D=57.15\text{mm}$).....	96
Figure 4.24: Comparison of zero-lift drag coefficient of the SOCBT rounds at various Mach numbers	98
Figure 4.25: a) Mach number contours of SOCBT, b) Mach number contour of SOCBT MOD3, and c) pressure distributions along SOCBT and SOCBT MOD 3, Mach 2, zero angle of attack	99
Figure 4.26: The comparison of drag coefficient versus angle of attack for four different SOCBT rounds, Mach 2	100
Figure 4.27: Comparison of normal force and pitching moment derivatives of four different SOCBT rounds, $\text{AOA}=4.2^\circ$	101
Figure 4.28: Comparison of the gyroscopic stability factor of four different SOCBT rounds under the same flight conditions, $\text{AOA}=4.2^\circ$, $PD/V = 0.283$	102
Figure 4.29: The pressure distributions along two different rounds at Mach 2, $\text{AOA}=4.2^\circ$: a) SOCBT and b) SOCBT (MOD 3)	103
Figure 4.30: Comparison of the dynamic stability factor on the modified generic rounds, $PD/V = 0.283$	104
Figure 4.31: The dynamic aerodynamic coefficients of three different generic rounds: a) Magnus moment derivative and b) pitch damping moment sum coefficient under $PD/V = 0.283$, $\text{AOA} = 4.2^\circ$	105
Figure 4.32: a) Magnus force coefficient and b) Magnus force center of pressure for the three-different generic rounds (SOCBT and modified SOCBT rounds) under $PD/V=0.283$, $\text{AOA}=4.2^\circ$	107
Figure 5.1: The Singapore Wind Tunnel Facility (SWIFT)	111

Figure 5.2: The available total pressure for the SWIFT wind tunnel	112
Figure 5.3: The drawing of two different wind tunnel models (1.668 scale): 155 mm M549 round (above) and optimum 155 mm geometry (bottom) (diameters = 92.88 mm)	115
Figure 5.4: The CAD model of two different rounds (1.668 scale) of the standard 155 mm M549 round (above) and optimum 155 mm geometry (bottom).....	118
Figure 5.5: The assembly of three different components of model and sting balance	118
Figure 5.6: The model and sting balance installation inside the wind tunnel test section	119
Figure 5.7: The two different configurations of pressure transducers installation over two different wind tunnel models (1.668 scale): Configuration A (6-probe) (above) and Configuration B (4-probe) (below).....	120
Figure 5.8: The balance moment center and center of gravity location of two different rounds: the Standard 155 mm M549 model (above) and an Optimum 155mm model (below) ($D=92.88\text{mm}$).....	121
Figure 5.9: The computational mesh of wind tunnel models (1/1.668 scale): a) axisymmetric mesh of the optimum round and b) 3-D grid of standard 155 mm round.	125
Figure 5.10: Drag minus base drag coefficient of two different rounds: a) optimum geometry and b) standard 155 M549 round.....	126
Figure 5.11: Total zero-lift drag coefficient of two different rounds: a) optimum geometry and b) standard 155 mm M549 round.....	128
Figure 5.12: Aerodynamic characteristics of the standard 155 mm M549 round: a) normal force derivative coefficient and b) pitching moment derivative coefficient. .	130

Figure 5.13: Aerodynamic characteristics of the optimum 155 mm round: a) normal force derivative coefficient and b) pitching moment derivative coefficient	131
Figure 5.14: The simple axisymmetric body ($D= 63.5$ mm).....	133
Figure 5.15: Comparison of the base pressure coefficient along the base	134
Figure 5.16: Mach number contours and streamlines behind the base	135
Figure 5.17: Comparison of the velocity profile at the center line behind the base....	135
Figure 5.18: Comparison of aerodynamic drag minus base drag coefficient of two different wind tunnel models based on wind tunnel testing	137
Figure 5.19: The comparison of unstable pitching moment of two different wind tunnel models based on wind tunnel testing	138
Figure 5.20: Effects of Reynolds number on aerodynamic coefficients of optimum 155 mm geometry at Mach 2; a) pitching moment coefficient and b) normal force coefficient.....	139
Figure 5.21: Effects of Reynolds number on zero-lift drag minus base drag of optimum 155 mm geometry at Mach 2.....	139
Figure 5.22: Effect of Reynolds number on drag components of optimum 155 mm geometry at Mach 2.	140
Figure 5.23: Comparison of schlieren images with Mach number contours of two different rounds: a) standard 155 mm M549 round, at Mach 4 and b) optimum 155 mm round at Mach 2.....	142

List of Nomenclature and Symbols

C_D	: drag coefficient (drag force / qS)
C_{D0}	: zero-lift drag coefficient (drag force / qS)
C_{DB}	: base drag coefficient (base drag / qS)
C_{DBT}	: boat tail pressure drag coefficient (boattail pressure drag / qS)
C_{DF}	: skin friction drag coefficient (skin friction drag / qS)
C_{DW}	: nose pressure drag coefficient (nose wave drag / qS)
C_{La}	: lift coefficient derivative (lift force / $qSD \sin \alpha$)
C_{Ma}	: pitching moment coefficient derivative (pitching moment / $qSD \sin \alpha$)
$C_{Mq} + C_{M\dot{\alpha}}$	: pitch-damping moment sum coefficient
C_N	: normal force coefficient (normal force / qS)
C_n	: side force coefficient (side force / qS)
$C_{np\alpha}$	: Magnus moment coefficient derivative (Magnus moment / $qSD(PD/V) \sin \alpha$)
C_{Na}	: normal force coefficient derivative (normal force / $qSD \sin \alpha$)
Re	: Reynolds number based on body diameter ($\rho VD/\mu$)
d_B, d_M	: base diameter and meplat diameter of the flat blunted nose tip (in D)
D	: body diameter (mm)
I_x, I_y	: moments of inertia (kg/m^2)
k_x	: axial radius of gyration, $\sqrt{I_x / mD^2}$
k_y	: transverse radius of gyration, $\sqrt{I_y / mD^2}$
L_{BT}	: boat tail length (in body diameters)
L_{CYL}	: cylindrical length (in body diameters)
L_N	: nose length (in body diameters)

P	: spin rate (rad/sec)
m	: mass, kg
M	: Mach number
N	: normal coordinates
q	: dynamic pressure, $1/2\rho V^2$ (N/m ²)
r, s, θ	: radial, streamwise, and tangential (or circumferential) coordinates
R	: ogive radius of projectile nose (in body diameters)
R_T	: tangent ogive radius (in body diameters)
S	: round shape cross-sectional area, m ²
S_g, S_d	: gyroscopic and dynamic stability factor
V	: freestream velocity (m/s)
W	: circumferential velocity (Eq. 2.1) and streamwise velocity (Eq. 4.8)
x	: axial coordinates
x_{cp}	: normal force center of pressure location, $x_{cg} - (C_{Ma}/C_{Na})$
x_{cg}	: center of gravity location (in body diameters)
A	: angle of attack (pitch) (degree)
β	: boat tail angle (degree)
Δ	: sine of total angle of attack (degree)
Γ	: ratio of specific heats; cosine of total angle of attack
$\dot{\phi}$	: coning rate (rad/sec)
Ξ	: complex variable of total angle of attack, $\alpha + i\Psi$
Ψ	: angle of side slip (yaw) (degree)
ρ	: density (kg/m ³)

Abstract

The purpose of this thesis is to study the external flow over the spin-stabilized rounds at supersonic speeds which is beyond the nominal flight of typical round bodies. The main work of this thesis can be divided into three parts: aerodynamics study and prediction, aerodynamic performance improvement, and aerodynamic testing. For the aerodynamics study, two different methodologies - semi-empirical method and computational fluid dynamics (CFD) - were employed to predict the aerodynamic behaviors and also used to provide better understanding of the physical flow over generic rounds under both non-spinning and spinning conditions. The validation study showed that these approaches can be used with reasonable accuracy to design the round geometry.

For the aerodynamic performance improvement, an optimization technique coupled with CFD was developed to optimize the 155 mm round. This technique is new, simple, and more accurate compared with previous techniques in the literature. Moreover, including the stability analysis and improvement into the optimization cycle was proven to be crucial to ensure that the designed shape is realistic. The optimized body provides up to 15% drag reduction and 46% increase in gyroscopic stability while remaining dynamically stable over the whole range of the operating Mach numbers. In addition to the optimization procedure, to further enhance the aerodynamic performance of the round bodies, boattail juncture contouring technique was investigated using CFD. The generic rounds with juncture contouring demonstrated a further reduction in drag by up to 5.4%, while remain within the allowable range of gyroscopically static and dynamic stability.

For the aerodynamic testing, to validate the superior performance of an optimum geometry obtained using the optimization procedure, the new optimum round and the standard 155 mm M549 body were tested in the large-size, high flow quality supersonic wind tunnel under non-spinning conditions. The wind tunnel test successfully demonstrated the accuracy and effectiveness of the optimization procedure.

Chapter 1

Introduction

1.1 Background and Motivation

The purpose of supersonic generic round shape design is to create a model that can provide the best performance to support the requirements of various missions. The generic round shape is one of many types of supersonic flight bodies that is widely used in the aeronautical community such as missiles, guided and unguided projectiles. Understanding the aerodynamics and design concept of these flight bodies has been an interesting topic for aerospace engineers since it is the key for enhancing the aerodynamic performance and accuracy. In addition, the results of aerodynamics are also used for structural analysis, including material requirements and selections. However, understanding the flow physics of generic rounds at supersonic regimes is complicated due to its involvement with the compression-expansion wave over the body surface and the strong recirculation flow at the round base. Moreover, the complex flow field over generic rounds under spinning motion is even more difficult to predict accurately by both numerical and experimental approaches. Consequently, these form the main difficulties in the preliminary design stage. Thus, the challenge of supersonic generic round design has become the motivation of this research.

To better understand the motivation behind this research, consider the following hypothetical, but practical, situation. Let's say there is an existing supersonic flight

body that is designed for a certain nominal range of flight speeds. How would the body be redesigned if the flight capability is to be extended to include higher flight speeds? How would the shape be optimized to get the best aerodynamic performance? And what are the flow physics that drive the performance?

The approach taken here is not to select complex flight body geometry, but to select a relatively generically shaped body as a representative example. The generic round shaped body chosen for the studies is a standard 155 mm diameter round projectile as shown in Figure 1.1.

Figure 1.1: The standard 155 mm spin-stabilized rounds: 155 mm M549 (left) and 155 mm M107 (right) [46].

This generic round shape is the typical spin-stabilized round body and it is designed for transonic to low supersonic flight regime. The maximum speed of this round is about two and a half times of the speed of sound (Mach 2.5), depending on the launch velocity. There are several key advantages of this selection. Firstly, the standard round has an axisymmetric shape that is relatively simple and such shape can be found in many flight bodies in supersonic flows. In contrast, supersonic aircrafts and missiles would be far more complex and their exact geometries would be difficult to obtain in the public domain. It is also desirable to limit the current studies without the complications of fins and wings. Secondly, the standard projectile geometry as

well as the experimental data of its performance can be readily found in open literature. The existing shape should already have a reasonably high performance in supersonic speeds, as its flight velocity is intended to be supersonic. Thirdly, the size of the standard round makes it an excellent candidate for wind tunnel experiments. The full-scale, or near-full-scale, body can be readily tested in wind tunnels without the added effects of scaling. Its full size has dimensions that also permit force measurements to be conducted with high accuracy.

There are two key parameters that are related to aerodynamic performance – drag and flight stability. The aerodynamic drag of the rounds can be separated into three parts: pressure drag (or wave drag), skin friction drag, and base drag. For flight stability, the gyroscopic static stability and dynamic stability are the main concerns of typical spin-stabilized generic rounds. These parameters are dependent on the aerodynamic coefficients of the generic round shape bodies such as pitching moment and Magnus effect coefficient. The aerodynamic drag components and the other aerodynamic characteristics of the round provide hindsight for potential areas for drag reduction and stability improvement. Thus, in order to enhance the performance of generic rounds, the ability to predict the aerodynamic behavior is the critical part for preliminary design stage.

For the aerodynamic prediction, there are three basic methods available for evaluating the aerodynamic coefficients: semi-empirical method, numerical method (CFD), and experimental method. In this research, the completed sets of three-different methods are applied to investigate the aerodynamics characteristics of the round bodies. In addition, these methods are also used to evaluate and design an optimum geometry that provides the best performance.

1.2 Objective of Study

The initial objective of this research is to study the aerodynamic characteristics of generic round-shaped bodies in supersonic flows using the large-size, high quality flow wind tunnel SWIFT (Singapore Wind Tunnel Facility) [1].

As mentioned earlier, the generic round shaped body selected for the studies will be a 155 mm M549 gun round (see Figure 1.1). The supersonic speed range of interest in this research is between flight Mach numbers of 1.8 to 4. This is considered as a typical range in the supersonic regime, with the lower value (Mach 1.8) being not too low to cause transonic issues, while the upper limit (Mach 4) being not too high to involve hypersonic problems. This speed range is also within the test capability of the SWIFT wind tunnel.

To investigate the aerodynamic characteristics of the generic round, experiments will be carried out at various supersonic speeds. The experiments will explore the aerodynamic performance at the nominally expected flight velocities of the round, as well as beyond. The nominal flight velocities to be studied will include the initial muzzle velocity immediately after launching of the round (approximately Mach 2.5), to the final decelerated velocity after the round has travelled a typical distance. Experiments will also examine the aerodynamic performance when the flight velocity is increased beyond the nominal values, up to Mach 4, which is the upper operating limit of the SWIFT facility.

However, the investigation of the 155 mm round using wind tunnel testing is expensive and time consuming. To reduce the experimental cost, prior to wind tunnel testing, a semi-empirical method and computational fluid dynamic (CFD) are used to predict the aerodynamic characteristics. The experiments will then be used to validate predicted results. The semi-empirical method and CFD can provide better

understanding of how the geometry will affect the aerodynamic and flight performance of round shaped flight bodies. Moreover, these methods can be also applied to design and examine an optimum shape at different flight Mach numbers and angles of attack (AoA) by minimizing drag and increasing stability, which would then lead to an increase in the flight range.

Therefore, the overall objective of the research is to study the external flow over the generic round shaped bodies at various flight conditions. Moderate Mach numbers of 1.8 to 4 and angles of attack of up to 5° are the flight conditions being considered.

The specific objectives are:

- To evaluate the aerodynamic characteristics of the generic 155 mm round shape using theoretical methods, CFD, and wind tunnel testing.
- To study the spinning effects on rotating generic round shape bodies using CFD.
- To gain an in-depth understanding of the physical flow over typical supersonic spin-stabilized rounds.
- To develop a novel method that can analyze and enhance the aerodynamic performance of the typical spin-stabilized rounds over a wide range of flight conditions.

1.3 Thesis Contributions

This thesis provides several contributions to both the development of a new methodology to design better round geometries and the understanding of the flow physics over supersonic spin-stabilized rounds.

In order to enhancing the performance of generic rounds, a coupled method of optimization technique and CFD were developed to optimize the round bodies to

obtain both drag reduction and stability improvement. This technique is new and useful for aeroballistics engineers because the optimization algorithm is simple, accurate, and the optimum shape can be obtained rapidly. Moreover, including the stability analysis into the optimization cycle was proven to be crucial to ensure that the designed shape is realistic.

The new optimum geometry obtained previously using optimization procedure and the standard 155 mm M549 body were first tested in the large-size; high quality flow supersonic wind tunnel at beyond the nominal flight conditions. No such data was reported in the literature previously. Therefore, the wind tunnel tests of these flight bodies provide the additional data for future validation. Moreover, the test results show that the aerodynamic performance of optimum round is better than the standard round, which confirms the accuracy and effectiveness of the optimization procedure.

For the flow physics study, the adverse pressure gradient on the conical boattail was described by the equation of motion (for inviscid and irrotational flow) in the cylindrical coordinates and illustrated in the simulation results. These pressure changes may seem counter-intuitive because the direction of change is opposite to what is expected downstream of an expansion fan. Moreover, the negative lift corresponding to this phenomenon over conical boattail rounds at finite angles of attack was also explained. An in-depth understanding of the flow phenomenon over the boattail leads to the motivation to enhance the aerodynamic performance of the round bodies. The contouring of the juncture of the conical boattail was then considered and employed. This was a popular shaping technique for boattail pressure drag reduction [2], especially in the subsonic and transonic regimes where this drag component dominates the total drag. However, the combination of drag reduction and stability analysis has

not been studied for this technique. In the current study, the effect of boattail juncture contouring technique on the aerodynamic performance of the generic rounds was investigated using CFD.

1.4 Thesis Outline

This thesis is divided into five chapters as follows:

- Chapter II Literature Review: This chapter is dedicated to illustrate the relevant literature and the past works related to the current research.
- Chapter III Methodology and Validation study: The theoretical methods and CFD techniques adopted in this study are described in this chapter. The validation studies of these methods on various round shaped bodies are also presented.
- Chapter IV Aerodynamics Performance Improvement: An optimization technique coupled with CFD was developed to optimize the spin-stabilized geometry to obtain the best aerodynamic performance in terms of drag and stability. The details of this optimization procedure will be discussed in this chapter. In addition, an in-depth explanation of the flow physics over the generic rounds and the negative lift characteristics over the conical boattail is discussed. Finally, to further enhance the aerodynamic performance of the rounds, the boattail juncture contouring technique is presented.
- Chapter V Wind Tunnel Testing: The wind tunnel facility, experimental settings and the test conditions are presented. The experiments of the standard 155 mm M549 round and optimum 155mm geometry in the supersonic wind tunnel (SWIFT) are then discussed. The wind tunnel testing data are used to validate the computational results obtained in Chapter 3 and Chapter 4. In addition, the comparison of the aerodynamic performance of these rounds will be also discussed.

- Chapter VI Conclusion and Future work: This chapter provides a summary of the entire process and pinpoints the important findings in this thesis.

Chapter 2

Literature Review

In this chapter, the background knowledge of the aerodynamic characteristics and stability of typical spin-stabilized generic rounds is discussed. Next, the relevant studies that are related to the capability of theoretical prediction methods, computational fluid dynamic (CFD techniques), and experimental approach (wind tunnel testing) are described. Finally, the background study of the methodology for aerodynamic performance improvement is presented.

2.1 Background

In the present study, the flow and the aerodynamic behaviors of spin-stabilized round shaped bodies are considered. A typical round body consists of a nose, a cylindrical body and a boattail afterbody as illustrated in Figure 2.1. The aerodynamics of these round shaped bodies can be characterized by two sets of parameters: static and dynamic aerodynamic coefficients.

Figure 2.1: The configurations of typical generic rounds

2.2 Aerodynamic Characteristics

2.2.1 Static Aerodynamic Coefficients

The aerodynamic characteristics of a spin-stabilized round shape under static conditions can be separated into six parameters based on six degrees of freedom: drag (or axial force), normal force, side force, pitching moment, rolling moment, and yawing moment. From wind tunnel testing under static conditions [3], the side force and the last two aerodynamic moments are usually small compared with the others. Thus, it is not necessary to account for these characteristics for the aerodynamic study of spin-stabilized round shapes.

2.2.1.1 Drag

In general, the drag on supersonic spin-stabilized round shaped bodies can be separated into three main components: pressure drag, skin friction drag, and base drag.

A. Pressure drag

For the supersonic generic round shaped bodies, the pressure distribution over the body's surface exerts normal forces, which when summed and projected in the free-stream direction, are referred to as pressure drag. There are three forms of pressure drag on typical round shaped bodies: wave drag due to the compression shock wave on the nose shape, pressure drag due to the expansion waves on the boat tail afterbody, and the pressure drag acting on the base, which is commonly known, as base drag. For the aerodynamic drag of generic round bodies, the base drag is separated from the pressure drag since it is convenient for aerodynamics analysis.

B. Skin friction drag

All fluids have viscosity, which causes the shearing of one fluid layer over another. The build-up of this viscous shearing force over the whole moving body is called skin friction drag [4, 5].

C. Base drag

The base drag is generated due to the strong expansion wave at the upstream corner of the base and recirculation flow at the base, which consequently yield a significantly low base pressure. The pressure difference between the forward part of the round and the base causes a net force that acts in the opposite direction to the motion of the generic round. Base drag is a major contributor to the round's total drag [6, 7].

2.2.1.2 Normal Force

In the same way as for the axial force, the total magnitude of the normal force depends on the distribution of pressure and shear stress over the body surfaces. Consequently, the normal force coefficient can be represented by the summation of two factors: pressure and viscous effects. However, for small angles of attack, the viscous effect is relatively small compared with the pressure effect. Thus, the normal force that is generated on round shape bodies is mostly produced from the pressure distributions around the surface. From the pressure measurements over the round shape bodies in experiments [8], it was found that the nose part of the round shaped body is the main contributor of positive normal force (or lift) while the boat tail afterbody generates negative lift [8, 9].

2.2.1.3 Pitching Moment (Overturning Moment)

The pitching moment coefficient (or overturning moment) depends on the same factors as the normal force coefficient. In general, most of the typical supersonic generic round shaped bodies that have high spinning motion are inherently statically unstable because the center of pressure of these flight bodies are located forward of the center of gravity (CG). Hence, a sudden disturbance introduced to the round will generate a moment about the CG causing the generic round to be unstable [10]. This is clearly shown in Figure 2.2. Thus, in order to stabilize these shapes, it is necessary to have additional force components or a turning moment (gyroscopic moment) over the bodies. The additional force components can be generated using fins at the rear part of generic rounds and the gyroscopic moment can be produced using spinning. The two different techniques are called *fin stabilization* and *spin stabilization*, respectively. As mentioned earlier, this thesis is mainly focused on the spin-stabilized rounds.

Figure 2.2: The round body under statically unstable condition

Here, the explanation of the spinning motion of the round body, which provides a gyroscopic moment to prevent the overturning moment, is discussed. Originally, there is no generation of overturning moment if the flight path direction of the spinning body (clockwise direction) coincides with the center of gravity and the axis of the round body (1st stage in Figure 2.3). Once the disturbance is introduced to the

round (2nd stage), the round nose is moved up (finite angle of attack). A normal force is generated (perpendicular to the round axis), which consequently yields the overturning moment about the center of gravity as shown in Figure 2.3 (3rd stage). This moment induces a spiral motion about the direction of flight (clockwise direction from the rear view), which is called gyroscopic precession or moment-induced precession [11]. The gyroscopic response due to the precession will be 90 degrees out of phase and will displace the spinning body in a perpendicular plane to the applied moment in which another overturning moment and another 90 degrees out of phase precession will result [10]. Thus, it is clear that this motion can prevent the large pitch-angle which would be caused by the disturbance. It should be noted that the flight trajectory of the spin-stabilized rounds are mostly in precession rather than a straight line. The phenomenon of precession is commonly seen in the spinning top. If the angular velocity of rotation and the magnitude of the overturning moment are constant, this motion is known as coning motion.

Figure 2.3: The demonstration of overturning moment induces the precession

2.2.2 Dynamic Aerodynamic Coefficients

When considering the spinning flight condition for spin-stabilized round shapes, the dynamic characteristics, namely roll damping moment, pitch damping moment and Magnus effect, need to be accounted for because these aerodynamic characteristics affect dynamic stability. Thus, it is necessary to accurately predict these characteristics, especially the pitch damping moment and Magnus effect, which are the main factors for dynamic stability [12].

2.2.2.1 Magnus Effect

As mentioned earlier, a round body without fins requires spinning to ensure gyroscopic stability during flight. However, this motion, when combined with a non-zero angle of attack, leads to the distortion of the boundary layer, which consequently yields an asymmetric pressure distribution with respect to the pitch plane as can be seen in Figure 2.4.

Figure 2.4: The demonstration of spin-induced boundary layer distortion

Thus, a side force is generated, which is commonly known as Magnus force [13]. The mathematical description of this physical phenomenon was first published by Martin [14] who successfully applied the boundary layer equations to explain the Magnus force on a cylindrical spinning body which is inclined at a small angle of attack. However, the boundary layer equations in Cartesian coordinates form are complicated. Thus, to simplify the description, Kelly [15, 16] transformed these equations for steady state and incompressible flow into the cylindrical coordinate system. Based on the mathematical derivation of Kelly, it was found that the pressure gradient normal to the round surface is a function of the circumferential boundary layer velocity as expressed in Eq. (2.1). It should be noted that this equation is well-known equation that has been used previously in literatures [4, 5].

$$\frac{\partial p}{\partial r} = \frac{\rho w^2}{r} \quad (2.1)$$

As shown in Eq. (2.1), it is clear that the asymmetric pressure distribution with respect to the pitch plane occurs since the boundary layer velocity profile along circumferential direction, w , is distorted by the spinning motion. In addition to the asymmetric pressure distribution, the other components which causes the Magnus force is the circumferential wall shear stress

$$\tau_{\theta} = \mu \left(\partial w / \partial r \right)_{r=r_{surface}}. \quad (2.2)$$

Since the gradient $(\partial w / \partial r)$ is not a constant around the circumference, a net side force due to this shear stress is generated. It is obvious that in the non-spinning case, the boundary layer velocity profile w is symmetric with respect to the pitch plane, thus, the net contribution of the Magnus force is zero. Normally, the Magnus force is relatively small compared with the normal force, typically 1/10 to 1/100 dependent on the geometry and spin rate. However, the Magnus force leads to the Magnus moment

which can be of sufficient magnitude to cause the projectile to become dynamically unstable [13]. Here, the definition of the Magnus moment coefficient derivative can be expressed as

$$C_{np\alpha} = \frac{\text{Magnus moment}}{qSD[PD/V]\sin\alpha} \quad (2.3)$$

where PD/V is the non-dimensional spin rate of the body.

2.2.2.2 Pitch Damping Moment

In the actual flight of spin-stabilized rounds, the combination of spinning motion about the longitudinal axis of the body and the motion about the center of gravity cause the oscillation of the round bodies, which consequently yield an additional moment of aerodynamic moment called the pitch damping moment [8]. Studies have shown that the pitch damping moment is proportional to the angular velocity of the body around its center of gravity [8]. While the evaluations of the static aerodynamic coefficients, roll damping, and Magnus moment derivative are straight forward, the pitch damping force and moment coefficients require a more complex mathematical derivation since these aerodynamic coefficients are dependent on the other aerodynamic characteristics.

Based on the derivation of linear flight mechanics, which was developed by Murphy [17], it is shown that the aerodynamic side force and moment acting on a round in steady coning motion can be related to the pitch damping force and moment coefficients. In the derivation, the moments in the coning reference frame can be expressed as below

$$C_M + iC_n = C_{M\alpha}\delta + \left[i\left(\frac{PD}{V}\right)C_{np\alpha} + i\left(\frac{\dot{\phi}D}{V}\right)[C_{Mq} + \gamma C_{M\dot{\alpha}}] \right] \delta \quad (2.4)$$

where PD/V and $\dot{\phi}D/V$ are the non-dimensional spin rate and non-dimensional

coning rate, respectively. In a steady coning motion, the round performs a rotation at a constant angular velocity about a line which is parallel to free stream velocity vector and passed through the center of gravity [18]. This is illustrated in Figure 2.5.

Figure 2.5: The steady coning motion of a spin-stabilized round

From the Eq. (2.4) above, it can be shown that the in-plane moment (real part) results only from the pitching moment (C_M), while the total side moment (C_n) (complex part) consists of contributions from the Magnus moment ($C_{np}\alpha$) and pitch-damping moment sum coefficient ($C_{Mq} + \gamma C_{M\dot{\alpha}}$). Here, it is required to evaluate the pitch damping moment sum coefficient. There are two methods that are available to investigate this coefficient: lunar coning methodology and combined spinning and coning methodology. In the current study, the lunar coning approach is chosen since it is assumed that the angular velocity of the round results purely from the rotation of the round body about the free stream velocity vector [18].

According to the lunar coning methodology, the relation between the coning rate, $\dot{\phi}$, and spinning rate, P or ω can then be written as $P = \dot{\phi} \cos \alpha = \dot{\phi} \gamma$. Thus, for small angle of attack (α), the final form to evaluate the pitch damping moment sum coefficient can be written as follows:

$$C_n = \delta \left(\frac{\dot{\phi} D}{V} \right) \left[C_{np}\alpha + (C_{Mq} + C_{M\dot{\alpha}}) \right] \quad (2.5)$$

To obtain the pitch-damping moment sum coefficient $C_{Mq} + \gamma C_{M\dot{\alpha}}$, the side moment due to steady coning motion C_n and the Magnus moment coefficient $C_{np\alpha}$ are first calculated. The detail of the evaluation of pitch damping moment will be discussed later in Chapter 3.

2.3 Flight Stability of Generic Rounds

Prior to designing the spin-stabilized round bodies, the stability criterion of these flight bodies is required. The stability criterion for the spin-stabilized rounds was first developed by Murphy [17], who transformed the linear flight mechanics equation to the form of complex variable of the total angle of attack ξ , which is expressed as follows:

$$\xi'' + (H - iQ)\xi' - (R + iQX)\xi = -iQG \quad (2.6)$$

where H , Q , R , X and G represents the so called Murphy's coefficients. These coefficients are developed from the aerodynamic and dynamic solutions of a body of revolution [17]. Since the complex solution of Eq. (2.6) is linear, the solution can be given as:

$$\xi = \left(K_{10} e^{i\Phi_{10}} \right) e^{(\lambda_1 + i\Phi'_1)s} + \left(K_{20} e^{i\Phi_{20}} \right) e^{(\lambda_2 + i\Phi'_2)s} + \xi_g \quad (2.7)$$

where $K_{j0} e^{i\Phi_{j0}}$, $j=1,2$, is the amplitude of the disturbances due to the initial condition, ($s=0$). Based on the solution of the linear system in Eq. (2.7), the gyroscopic frequencies effect Φ'_j , and damping coefficients λ_j obtained from the homogeneous solution, will vary with the relative magnitudes of the Murphy's coefficients [17]. The formulation of gyroscopic frequencies, Φ'_j , and damping coefficient, λ_j , can be written as

$$\Phi'_j = \frac{1}{2} \left[Q \pm \sqrt{Q^2 - 4R} \right], \quad j=1,2, \quad (2.8)$$

$$\lambda_j = -\frac{I}{2} \left[H \mp \frac{Q(2X-H)}{\sqrt{Q^2-4R}} \right], \quad j=1,2. \quad (2.9)$$

For real values of both Φ_j' and λ_j in Eq. (2.8) and Eq. (2.9), the value in the square root term must be positive, i. e, $Q^2-4R>0$ or $4R/Q^2<1$, which can be used to determine the gyroscopic stability criterion as follow

$$\text{Let } \frac{4R}{Q^2} = \frac{1}{S_g}$$

$$\text{Thus } \frac{1}{S_g} < 1 \text{ or } S_g > 1 \quad (2.10)$$

Since Murphy's coefficients Q and R are represented by the aerodynamic coefficients and the moment inertial, the gyroscopic factor can be determined by

$$S_g = \frac{Q^2}{4R} = \frac{\left(\frac{I_x^2}{I_y} \right) \left(\frac{PD}{V} \right)^2}{\left(2\rho SD^3 C_{M\alpha} \right)} \quad (2.11)$$

where

$$Q = \left(\frac{I_x}{I_y} \right) \left(\frac{PD}{V} \right)$$

$$R = \left(\frac{\rho SD^3}{2I_y} \right) C_{M\alpha}$$

In addition to the gyroscopic stability criterion, a coupled requirement for stability is that the damping coefficients λ_1 and λ_2 need to be negative to satisfy the condition of unique solution $\lim_{s \rightarrow \infty} \xi \rightarrow 0$. This is needed so that the final solution of the complex linear form in Eq. (2.7) can be solved. Thus, this second condition evolves to be the so called dynamic criterion of stability S_d as follows:

$$S_d = \frac{2X}{H} \quad (2.12)$$

with

$$X = C_{L\alpha} + k_x^{-2} C_{np\alpha}$$

$$H = C_{L\alpha} - C_D - k_y^{-2} (C_{Mq} + C_{M\dot{\alpha}})$$

where k_x and k_y are the axial radius and transverse radius of gyration, respectively.

When using aerodynamic coefficients, the final form of the dynamic stability factor can be expressed as

$$S_d = \frac{2(C_{L\alpha} + k_x^{-2} C_{np\alpha})}{C_{L\alpha} - C_D - k_y^{-2} (C_{Mq} + C_{M\dot{\alpha}})} \quad (2.13)$$

It can be seen that S_d depends on the aerodynamic characteristics defined in the previous section (section 2.2). To satisfy the second condition, the damping coefficients (Eq. (2.9)) have to be negative, which yields the following inequalities

$$\left[\frac{Q^2 (S_d - 1)^2}{Q^2 - 4R} \right] < 1 \quad \text{and} \quad H > 0 \quad (2.14)$$

when combining the above inequalities (Eq. (2.14)) with the gyroscopic stability criterion (Eq. (2.10)), the new form of dynamic stability criterion can be expressed as

$$\frac{I}{S_g} < S_d (2 - S_d) \quad (2.15)$$

From the stability criterion of spin-stabilized rounds, it can be noted that the dynamic stability ensures gyroscopic stability, but the converse is not necessarily true. This inequality can be shown graphically in Figure 2.6. From this figure, the inequality in Eq. (2.15) implies that S_d must be in the range between 0 and 2. Furthermore, to ensure the stability of spin-stabilized rounds, the stability factor must fall within the stability criterion bound plotted in Figure 2.6.

Figure 2.6: The stability criterion bound of spin-stabilized rounds

2.4 Aerodynamic Prediction

Section 2.2 and 2.3 show that the drag and stability factors are related to the aerodynamic characteristics of the round bodies. Thus, prior to analyzing the round performance, the aerodynamic coefficients are first needed. As mentioned earlier, there are three basic methods available to investigate the aerodynamic behaviors: theoretical method, CFD, and experiment. Thus, the literature studies of aerodynamic prediction which is related to these methods are discussed in the following sections. Once the accuracy of these methods is confirmed, it can then be used to design the round geometry.

2.4.1 Theoretical Method

The evaluations of aerodynamic characteristics on generic round shaped bodies using theoretical methods have been studied for many years. Most of the studies attempted to develop methods that can be used to predict aerodynamic characteristics

accurately. In 1933, the Taylor-McColl equation based on the axisymmetric Euler's equation was developed to solve the aerodynamic drag (wave drag) on simple cone round shape bodies [19]. Because the solutions of these equations are exact, the aerodynamic results obtained using these theories were very accurate compared with experiments. However, these methods were limited to only simple cone shapes at zero angle of attack.

To extend the validity of these analysis, the first-order method, second order method, and the hybrid theory of Van Dyke based on linearized theory were developed to solve the aerodynamic characteristics (wave drag, normal force, and pitching moment) on arbitrary round shaped bodies [20, 21]. These methods were derived from Euler's equations under the assumptions of slender body, low angle of attack, irrotational flow, and the transformation of non-linear equations to linear ones. All these methods provided reasonably accurate static aerodynamic results. However, the dynamic aerodynamic coefficients cannot be accurately predicted using these theories. Moreover, these methods were limited to the prediction of the aerodynamics over the point-nose round shape bodies at the low supersonic regime. To overcome this difficulty, the second-order shock expansion theory plus modified Newtonian impact theory was developed to allow the handling of blunt-nosed configurations for moderate to high supersonic Mach numbers [22, 23, 24].

Although the drag can be accurately predicted using theoretical methods, these methods can be only used to evaluate the wave drag which excludes the skin friction drag and the base drag. For skin friction drag, for most practical configurations, the boundary layer begins in a laminar state and transitions at some point to turbulent flow. In many situations, for supersonic flows, the boundary layer will be turbulent over ninety percent of the round shape body for large size generic rounds [25]. Thus, we

can assume the flow over the entire body to have turbulent boundary layers. Under this assumption, the Van Driest turbulent boundary layer theory, which was developed to predict the turbulent skin friction coefficient over a flat plate, was then employed on round shaped bodies [26]. However, although Van Driest's theory can provide reasonable skin friction results, Van Driest's equation is difficult to evaluate because it requires an iterative numerical solution. To solve the Van Driest equation, Prandtl's empirical formula [27] was used for fully turbulent boundary layers based on the correlation of Van Driest's results and experimental database. Not only does this empirical method provide a rapid solution but it also provides good agreement with Van Driest's complete theory for compressible turbulent flow.

For the base drag, many studies were devoted to solve for this using theoretical methods. There are still no satisfactory theories available because the physical flow at the base area is very complicated and difficult to predict. Nevertheless, in 1951, a good empirical method based on Chapman's solution was proposed to predict the base drag. The empirical method was developed from the interpolation of many experimental database [6]. From the study of Chapman, the base pressure depends strongly on the local Mach number and the type of boundary layer just upstream of the base. The use of this empirical method provides reasonable base drag results.

From the theoretical and empirical methods mentioned above, it can be noticed that not all of the aerodynamic characteristics can be calculated using only theoretical method or empirical database alone. Thus, it is necessary to combine them to obtain a more complete theory. In this approach, the combination of theoretical and empirical techniques is called semi-empirical method. There are many semi-empirical codes for the aerodynamic characteristics calculation such as PRODAS (Projectile Rocket Ordnance Design & Analysis System), AP05 (or Aero-Prediction 2005), and Missile

Datcom [28, 29]. However, the availability of these codes in the scientific community is restricted.

2.4.2 Computational Fluid Dynamics

Numerical methods or computational fluid dynamics (CFD) have been widely used to predict the aerodynamic characteristics of round shaped bodies for more than 20 years. Not only can they accurately predict the flow over complex geometries, they can also clearly provide the flow details over the bodies. There are three basic numerical approaches to evaluate the aerodynamic characteristics over round shaped bodies: finite difference, finite element, and finite volume methods. All these techniques have their advantages and disadvantages depending on the complexity of the geometry and the time required to obtain a solution. However, for fluid flow problems, the finite different and finite volume methods are the more common tools.

In 1986, the finite different scheme based on the Beam-Warming implicit algorithm was applied to solve the two-dimensional axisymmetric thin-layer Navier-Stokes equations over supersonic round shaped geometries [30]. The turbulence model of Baldwin and Lomax was used to evaluate the skin friction drag [31]. The zero-lift drag components of pressure drag obtained from the computational solutions were in good agreement with the experimental database [32]. However, the base drag solution had significant discrepancies over the low supersonic regime because the turbulence model is not designed for the large wake region that occurs at the base area. Moreover, the flow over the base area is very complicated to evaluate.

As mentioned previously, the base drag is one of the main contributors to the total drag that affects the aerodynamic performance. Thus, the accuracy of the base drag is critical. To improve the accuracy of the base drag calculation, many studies attempted to apply different Reynolds-Average Navier-Stokes equation (RANS) turbulence

models to evaluate the physical flow over the base area. Tucker and Shyy [33] used several variations of the two-equation k - ϵ model to obtain improved response to the mean strain rate and compressibility effects. Suzen et al. [34] applied several RANS turbulence models on a two-dimensional base. Forsythe et al. [35] also tested several RANS models on two-dimensional and axisymmetric bases. Most of these studies provided reasonable results for the base drag on both two-dimensional and axisymmetric flows. In addition, Menter's model [36] and Wilcox's k - ω model [37] performed far better than the other turbulence models. However, although all of the turbulence models that have been used to evaluate the base drag provided reasonable results, these studies focused only on the zero-lift drag prediction and excluded the other aerodynamic characteristics.

Therefore, many recent studies [38, 39, 40, 41, 42] were devoted to addressing the other aerodynamic behaviors over round shaped bodies. Steady and unsteady RANS, Large Eddy-Simulation (LES) and Detached-Eddy Simulation (DES) were employed using the finite volume scheme to evaluate both the static and dynamic aerodynamic characteristics over generic rounds (Army-Navy Spinner Rocket (ANSR), M910 sub-caliber, and 0.5 body diameters projectiles). The discretization of second-order accuracy was used. The cubic k - ϵ and realizable k - ϵ turbulence models in the CFD⁺⁺ commercial code package have been used in the RANS solutions. The full three-dimensional model mesh needs to be used to capture the flow field at small but finite angles of attack.

From the validation study of these simulations based on available aeroballistics range data [43, 44], it was found that the steady-state CFD simulation alone is adequate to predict the dynamic roll-damping moment, pitch damping moment, and the static aerodynamic characteristics on generic rounds at all Mach number range. For

the Magnus effect, the steady-state RANS also provides reasonable accuracy at moderate to high supersonic speeds. However, the steady-state RANS yielded significant discrepancies at the low supersonic, transonic, and subsonic speeds because of the non-linear Magnus moment [12].

From the literature mentioned above, it is shown that the realizable k- ϵ RANS turbulence model provides reasonable aerodynamic characteristics, while the computational cost to obtain the solution is also less than those of LES and DES. However, solving the wall boundary layer is still the weak point of this turbulence model. Because the k- ϵ family of turbulence models suffers from the drawback that it is invalid at the wall, near-wall functions ($y^+ < 5$) have to be used to avoid the difficulty of integrating the equations to the wall. Therefore, these methods may not be appropriate for wall-bounded turbulent boundary layers. In addition, the solutions might be affected by Magnus effects because the Magnus force and moment are very sensitive to the details in the wall boundary layer [39]. To improve the accuracy of the k- ϵ model, the Menter's hybrid model was developed by blending the standard k- ϵ model with the k- ω turbulent model in the near wall region. The resulting hybrid model is the Shear Stress Transport (SST) k- ω turbulence model which can provide accurate results in the wall-bounded turbulent boundary layer flows [36]. Thus, in this study, the Shear Stress Transport (SST) k- ω turbulent model is used to predict the aerodynamic characteristics on the round shaped bodies. The details of this turbulence model will be discussed later in Chapter 3.

2.4.3 Wind Tunnel Testing

There are two common experimental techniques for testing spin-stabilized round bodies: wind tunnel testing and aeroballistics range (spark range). As mentioned earlier, in this study, since aeroballistics range testing is not available, wind tunnel testing is

chosen as the main approach for aerodynamic investigation. There are several advantages of wind tunnel testing compared to the spark range and CFD. Firstly, the aerodynamic forces and moments on the round are measured directly using a balance. Thus, the aerodynamic forces and moments obtained from this approach are more realistic than from CFD. Secondly, the experimental cost of wind tunnel testing is cheaper than that of aeroballistics range. Moreover, the experimental set up is simpler. However, wind tunnel testing is more time-consuming compared with CFD and usually has some limitations on the measurements compared with the aeroballistics range. For example, the dynamic aerodynamic coefficients such as pitch and roll damping and Magnus force and moment coefficients are difficult to obtain in a wind tunnel since a complex mechanism is required to generate the spinning motion. In addition, the aerodynamic drag is also difficult to measure accurately because the base drag is strongly influenced by the base flow at the mounting region.

For the previous work of wind tunnel testing, to investigate both static and dynamic aerodynamic characteristics, many studies [3, 48, 63, 68] attempted to test the generic round bodies in supersonic wind tunnels under both non-spinning and spinning conditions. Kayser and Sturek (1978) [63] tested a spinning Secant-Ogive-Cylinder (SOC) in a supersonic wind tunnel (test section area: 330mm x 381mm) at the Ballistic Research Laboratory to investigate the aerodynamic behaviors. These experiments were conducted at Mach 3 for different angles of attack. Force measurements (from balance) and surface pressure measurements were performed. In addition, pitot probes were used to measure the boundary layer profiles over the body surface. However, only the Magnus effect results and pressure distributions along the body were published.

To obtain a complete study of aerodynamic coefficients, Platou (1979) [68] tested a series of non-conical boattail of the standard 155 mm M549 (1/3 scale) in a supersonic wind tunnel at the Naval Surface Weapons Center (NSWC) under both non-spinning and spinning conditions. These non-conical shapes were modified from the standard 155 mm M549 round to improve the aerodynamic performance. Two different flight regimes were considered for this experiment- transonic and supersonic speeds (Mach 1.75 and 2.25). A spinning test was also performed to investigate Magnus effects. The experimental results showed that the modified 155 mm rounds yielded a lower unstable pitching moment, while significantly reducing the Magnus moment derivative as compared to the standard round. However, the aerodynamic drag results were not provided in this test since it was difficult to measure the base pressure accurately.

To address the aerodynamic drag measurement, Sivan and Jermy (1989) [3] tested a 1/3 scale of the standard 155 mm round in the S-1 wind tunnel (test section: 0.36m x 0.36m) at the Flight Mechanics and Aeropropulsion Division of ARL-Salisbury. Tests were conducted for various Reynolds numbers of 1.0×10^6 to 4.0×10^6 , angle of attack from 0° to 20° , Mach numbers ranging from 0.7 to 1.8, and spin rates from 0 to 400 rev/s. The test results indicated that the spinning motion has a small effect on the static aerodynamic coefficients, while a significant effect on side force and yawing moment was obvious. In addition, a base pressure measurement was also successfully performed to correct the aerodynamic drag. A good agreement was observed between the aeroballistics range and the wind tunnel data.

For more recent wind tunnel studies, Fillipone and Ibrahim (2010) [48] tested the Secant-Ogive-Cylinder- Boattail (SOCBT) with a slot cavity on the conical boattail to compare the aerodynamic performance with the standard SOCBT. The tests were

performed in the supersonic wind tunnel (test section area: 0.15m x 0.214m) at the Goldstein Laboratory at the University of Manchester. Three different Mach numbers for zero angle of attack - 1.36, 1.65, and 1.83 - were considered as the test conditions. Two different techniques: surface pressure measurement and force measurement were employed. However, no corrections were applied to the measured drag force to account for the interference created by the sting.

2.4.4 Shock waves Boundary Layer Interaction (SWBLI)

The interaction of shock waves with boundary layers over the projectiles (round bodies) is a basic fluid-dynamics phenomenon that usually appears in transonic and supersonic flows. This phenomenon can have a significant influence on the projectile performance and often leads to extremely undesirable effects such as drag rise, massive flow separation, shock unsteadiness, and projectile instabilities [75]. In general, there are four patterns of SWBLI: weak interaction (no boundary layer separation), medium interaction (very small boundary layer separation), strong interaction (sizable separation bubble on the foot of shock wave) and very strong interaction (complicated wave system or multiple shock wave). The demonstration of these patterns are shown in Figure 2.7. The difference of these patterns is dependent on many parameters such as the configuration of the models, the state of boundary layer, and flight regimes. The detail of the boundary layer shock interaction theory can be seen in Anderson, J.D. [4].

Figure 2.7: The four patterns of SWBLI [76]: a) weak interaction, b) medium interaction, c) strong interaction, and d) very strong interaction.

There are some literature studies devoted to study the effect of SWBLI on the performance of the round bodies. Reklis et al. [75] performed the experiment to study the flow phenomenon over the conical boattail at transonic regimes. They found that the flow around an artillery projectile flying at a transonic speed presents a difficult and interesting problem. The general features of the flow pattern over the round body in terms of shadowgraph can be seen in Figure 2.8a. As shown in this figure, the discontinuities in surface curvature (at the juncture between the nose – cylindrical body and cylindrical body – conical boattail) over the round body generate rapid expansions that consequently cause the flow to become supersonic. The supersonic regions thus created terminate in shock waves whose structure is complicated by interaction with the viscous boundary layer. This boundary layer is particularly thick over the boattail where it has grown almost to the point of separation [75]. Regarding

this flow phenomenon, the strong adverse pressure gradient is generated as shown in Figure 2.8b. In addition, the unsteady shock is also occurred over the round body. This leads to obtain a large rise in drag and also provide more unstable flight when the rounds operate at finite angle of attack [13, 75]. That shows reason why the aerodynamic behaviors at transonic regimes become non-linear [4].

Figure 2.8: The experimental data of projectile at transonic speed (Mach 0.94) [75]: a) a shadowgraph and b) pressure distribution along round body.

However, at supersonic speeds, since the compression shock (or Mach waves) over the conical boattail is very weak and the boundary layer (over small boattail angle) is also relatively thin compared with at transonic regimes, the SWBLI has very small effect on the aerodynamic characteristics over round bodies. That shows reason why the aerodynamic behaviors over the round bodies (under small angle of attack) at this regime remain the linear trend.

In the current study, since the experiments have been conducted at supersonic speeds well beyond the transonic range, it is confirmed that there is no SWBLI existing on the round geometries or it plays a small role to the aerodynamic characteristics. Thus, the adverse pressure gradient over the present round geometries at this regimes is not caused by SWBLI, but it is directly generated by three dimensionality effects of the round. This can be demonstrated by the steady/unsteady simulation results of the flow detail over the rounds (to be discussed latter in Chapter 4 (section 4.3)).

From a numerical point of view, to simulate the SWBLI over the round bodies, there are many studies attempted to capture this flow phenomenon using RANS approach which incorporate configuration-dependent flow physics [12, 39, 40]. They found that RANS models fail to capture the high level of unsteadiness in the shock system observed in the experiments and this approach cannot provide a good prediction of root-mean-square pressure and Magnus effect at the surface. Recently, however, large-eddy simulation (LES) and direct numerical simulations (DNS) approach have been applied to the SWBLI problem with significant success. However, since this transient methodology required a very fine mesh and very small time step size to control the stability of numerical solution (especially, at supersonic speeds), the computational cost of this methodology is very high and also time consuming. Thus, in

the current study, since it is known that SWBLI plays a small role to the round performance and the aerodynamic behaviors over the round bodies at supersonic regimes are linear, the RANS approach can be used with confidence to provide the reasonable aerodynamic coefficients.

2.5 Aerodynamic Performance Improvement

As mentioned earlier, aerodynamics drag and flight stability are two key metrics that are related to the aerodynamic performance of a supersonic round body. In the past, many studies [45, 46, 47, 48] attempted to minimize the total drag using different shaping techniques in order to achieve better performance. Since it is known that the base drag is the major contributor to the total drag, the focus of these studies was mainly on base drag reduction. Viswanath [45] demonstrated the effectiveness of base cavities, ventilated cavities, multi- stepped afterbodies, and boattail configuration for base and net afterbody drag reduction. Suliman et al. [46] employed CFD to study the effects of the conical boat tail, the base cavity, and base bleed configurations on the drag reduction of the 155 mm M549 projectile. Ibrahim and Filippone [47, 48] applied the slot cavities and porous surface on a conical boattail to evaluate their effectiveness on base drag reduction using both experiments and CFD. Shrewsbury (1968) applied the boattail juncture shaping technique at the upstream corner of conical boattail to reduce boattail pressure drag [2]. This shaping technique was designed based on the combination of conical boattail shape and circular arc profile at the juncture between the cylindrical body and the boattail.

Besides shaping techniques, optimization is also frequently used to minimize drag. One of the most sophisticated optimization techniques was proposed by Nicolas [49] in which the pressure drag is computed based on the modified Newtonian impact

theory [50], the skin friction drag is evaluated using the turbulent boundary layer theory [51] and the base drag is computed from empirical correlations. The overall drag is then minimized using an Eulerian optimization scheme. This technique required iteration of the total drag equation until the solution yields the final optimum shape. Although this method is useful for shape optimization, it involves complicated evaluation of multiple sets of equations. Iteration is also time-consuming and the use of a first-order method in the technique may have led to inaccuracies. Furthermore, the optimum round shape obtained may not be practical for spin-stabilized bodies as it has no cylindrical portion and will thus generate large wobbles in the launch tube leading to unstable flight.

From the literature studies mentioned above, although the drag can be minimized using both shaping techniques and optimization method, these drag minimization approaches cannot guarantee the stability of the round during flight. Thus, in order to design realistic spin-stabilized rounds, the stability issue needs to be addressed into the preliminary design stage.

Chapter 3

Methodology and Validation Study

In this chapter, two different methodologies to study the aerodynamics of generic round bodies, namely, semi-empirical method and computational fluid dynamics (CFD) are first presented. Next, the ability of these approaches to predict the static and dynamic aerodynamics characteristics of spinning and non-spinning round bodies is then investigated. Finally, the semi-empirical and numerical results are validated with existing aerodynamic data obtained from both wind tunnel testing and aeroballistics range.

3.1 Governing Equations

The motion of continuum fluid is governed by the Navier-Stokes equations (NS), which represent the conservation of mass, momentum, and energy. For turbulent flow in which the turbulent scales are much larger than the mean free path of the fluid [52], the NS equations can be solved numerically using direct numerical simulation (DNS) without any turbulence model. However, the computational cost of DNS grows very fast with the Reynolds number, especially for the large Reynolds numbers encountered in industrial applications. Consequently, the requirement of computational resource of DNS is beyond the capacity limit of computer power. Thus, to reduce the computational cost, Reynolds-averaged Navier-Stokes equations are commonly used to simulate a practical flow in practical situation. The Reynolds-averaged Navier–

Stokes equations (or RANS equations) are time-averaged equations in which the instantaneous quantities in these equations are decomposed into its mean properties and fluctuating quantities. This idea was first proposed by Osborne Reynolds (1895) [53]. Based on this concept, the fluid properties, ϕ , are defined as the sum of the mean part and fluctuation part, such as $\phi = \bar{\phi} + \phi'$ or $\phi = \tilde{\phi} + \phi''$ (using ensemble average and Favre time averaging). By inserting the decomposition of ensemble and Favre time averaging into the Navier-Stokes equations, the Reynolds Averaged Navier-Stokes equations (RANS) can be obtained as follows:

Mass conservation:

$$\frac{\partial(\bar{\rho})}{\partial t} + \frac{\partial(\bar{\rho} \tilde{u}_i)}{\partial x_i} = 0 \quad (3.1)$$

Momentum conservation:

$$\frac{\partial(\bar{\rho} \tilde{u}_i)}{\partial t} + \frac{\partial(\bar{\rho} \tilde{u}_i \tilde{u}_j)}{\partial x_j} = \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} [\bar{\tau}_{ij} - \overline{\rho u_i'' u_j''}] \quad (3.2)$$

Energy conservation:

$$\frac{\partial(\bar{\rho} \tilde{E})}{\partial t} + \frac{\partial}{\partial x_j} [\bar{\rho} \tilde{H} \tilde{u}_j + \overline{\rho u_i'' u_j''} \tilde{u}_i] = \frac{\partial}{\partial x_j} (\bar{\tau}_{ij} \tilde{u}_i) - \frac{\partial}{\partial x_j} [\bar{q}_j + \bar{q}_j^t] + \bar{\rho} D^k \quad (3.3)$$

where $\tilde{u}_i$ and u_i'' represent the i -th component of the mean and fluctuating fluid velocity, $\bar{q}_j^t = -\overline{\rho u_j'' h''}$ is the turbulent heat flux, and $\bar{\rho} D^k = \frac{1}{2} \overline{\rho u_j'' u_i'' u_i''} - \overline{\rho u_i'' \tau_{ij}}$ represent the turbulent transport of turbulent kinetic energy and molecular diffusion, respectively.

The total energy and enthalpy are defined as

$$\tilde{E} = \tilde{e} + \frac{1}{2} \tilde{u}_j \tilde{u}_j + \tilde{k} \quad \text{and} \quad \tilde{H} = \tilde{h} + \frac{1}{2} \tilde{u}_j \tilde{u}_j + \tilde{k} \quad (3.4)$$

and the turbulent kinetic energy:

$$\tilde{k} = \frac{1}{2} \overline{\rho u_i'' u_i''}. \quad (3.5)$$

To solve the above compressible RANS equations, the Reynolds stress tensor, $-\overline{\rho u_i'' u_j''}$ must be modeled. In this study, the Eddy Viscosity Models (EVM) based on the Boussinesq hypothesis [53] is employed to close the system of equations. From this hypothesis, the Reynolds stress is proportional to the rate of strain of the time averaged (mean) velocity as expressed below.

The Reynolds stress tensor:

$$-\overline{\rho u_i'' u_j''} = -2\mu_t \left(\frac{1}{2} \left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right) - \frac{1}{3} \frac{\partial \tilde{u}_j}{\partial x_j} \delta_{ij} \right) + \frac{2}{3} \tilde{\rho} \tilde{k} \delta_{ij} \quad (3.6)$$

where δ_{ij} is the Kronecker delta and the proportionality constant (μ_t) is called eddy viscosity (or turbulent viscosity). The eddy viscosity is computed from the turbulence model.

3.2 Theoretical Work

Most of the theoretical works which have been used for aerodynamic prediction of pressure drag and skin friction drag were developed from the full NS and RANS as shown in Eqs. (3.1) to (3.3). Since it is known that the full Navier-Stokes equation and Reynolds Average Navier-Stokes cannot be solved analytically, there is a need to simplify these equations. For example, the full steady, compressible, Navier-Stokes equations can be transformed to the perturbation-velocity potential equations (linearized-method) [7] using the assumption of inviscid, irrotational flow, and small perturbations to provide a close form with analytical solution. Another example is that

Van Driest [26] simplified the two-dimensional compressible RANS to the boundary layer equation using the assumption of zero pressure gradients on a flat-plate to obtain the turbulent skin friction. However, as mentioned earlier, the theoretical method or empirical correlation alone is not enough to provide a complete set of aerodynamic characteristics. Thus, the semi-empirical method is needed to fulfill the requirements.

In the present study, the semi-empirical method based on McCoy's solution [55] is selected to be the tool to evaluate the aerodynamic zero-lift drag because of its simplicity and the use of explicit calculations [55]. Moreover, this method is one of the very few semi-empirical tools available in the open literature. The drag components of the round shapes consist of pressure drag or wave drag, skin friction drag, and base drag. All of these drags can be calculated separately by using individual formulations based on the theory and empirical database of generic round shapes. For the pressure drag calculation (nose wave drag and boattail pressure drag), McCoy developed a simple explicit formulation based on the correlation database of Van Dyke [56] and the Taylor-McColl theory [57] using similarity rules. As the result, McCoy's solution is very easy to implement for pressure drag predictions. For skin friction and base drag calculations, Prandtl's empirical formula [27] and McCoy's empirical method [55] are applied to evaluate the turbulent skin friction and base drag, respectively. To evaluate the total zero-lift drag, a program was written in MATLAB (the set of McCoy's equations and the MATLAB codes details are available in Appendices A and B, respectively). The flow chart of McCoy's semi-empirical codes is shown in Figure 3.1.

The semi-empirical codes start with the input parameters of the round geometries and free stream Mach number. The codes consist of three-main programs: pressure drag codes, skin friction drag codes, and base drag codes which separately calculate

the aerodynamics drag components. The codes are followed by the output program, which provides the function of summation of three-different drag components as shown in Figure 3.1.

It should be noted that this efficient method provides solutions in seconds and can be used for fast prediction of the aerodynamic zero-yaw drag components on round shaped bodies. In addition to the semi-empirical codes, since the McCoy's semi-empirical method is simple to evaluate the zero-lift drag coefficient, this code will be used as the objective function of an optimization procedure for designing the optimum 155 mm generic round that can provide minimum total drag. This topic will be discussed later in Chapter 4.

Figure 3.1: The flow chart of McCoy's semi-empirical codes

3.3 Computational Approach

In the current study, the steady, compressible, Reynolds Averaged Navier-Stokes equations (RANS) in section 3.1 are solved numerically using the finite volume method. This methodology is already developed in the form of CFD code package. In this study, a CFD code, FLUENT [58], has been applied to investigate both the static and dynamic aerodynamic characteristics over the generic rounds at various flight conditions. For the specific calculations in this study, the density-based axisymmetric and three-dimensional flow solvers are selected. The axisymmetric solver is employed for the round body at zero angle of attack and the three-dimensional solver is used for finite angles of attack. The spatial discretization is based on a cell-centered finite volume scheme. The convective fluxes are discretized by Roe's upwind flux splitting method and second order accuracy in space is achieved by Green-Gauss reconstruction. The viscous fluxes are discretized by the central difference method. The flow is assumed to be quasi-steady and the mean flow equations are integrated in time by a fully implicit scheme.

To close the RANS equations, the Shear Stress Transport (SST) $k-\omega$ turbulence model developed by Menter [36] is selected because this model is very well validated in various flows and was rated in a NASA technical memorandum as one of the most accurate RANS models for aerodynamic applications [59]. As mentioned in Chapter 2, the SST turbulence model includes two main novelties:

- It is a combination of the advantages of the standard $k-\omega$ model [37] (Wilcox, 1988) in the inner boundary layer and the advantage of the standard $k-\epsilon$ model [60] (Jones and Launder, 1972) in the outside region of the boundary layer.

- A limiter of the shear stress in adverse pressure gradient regions is introduced.

The SST model was developed to effectively blend the robust and accurate formulation of the k - ω model in the near wall region with the free-stream independence of the k - ϵ model in the far fields. To blend these models together, the standard k - ϵ model was transformed into the k - ω formulation. Based on this concept, the SST model can provide more accurate and reliable predictions for a wider class of flows such as those under adverse pressure gradient, highly separated flows, and boundary layer flows [60].

The transport equations of the Shear Stress Transport (SST) k - ω model which has a similar form to the standard k - ω model are expressed in Eq. (3.7) and Eq. (3.8), respectively.

$$\frac{\partial(\bar{\rho}k)}{\partial t} + \frac{\partial(\bar{\rho}u_j k)}{\partial x_j} = -\bar{\rho}u_i''u_j'' \frac{\partial u_i''}{\partial x_j} - \beta_k \tilde{k} \omega + \frac{\partial}{\partial x_j} \left[(\bar{\mu} + \sigma_k \mu_t) \frac{\partial \tilde{k}}{\partial x_j} \right] \quad (3.7)$$

and

$$\frac{\partial(\bar{\rho}\omega)}{\partial t} + \frac{\partial(\bar{\rho}u_k \omega)}{\partial x_k} = -\alpha_\omega \bar{\rho}u_i''u_j'' \frac{\partial u_i''}{\partial x_j} \frac{\partial \bar{u}_i}{\partial x_j} \frac{\omega}{\tilde{k}} - \beta_\omega \bar{\rho}\omega^2 + \frac{\partial}{\partial x_k} \left[(\bar{\mu} + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_k} \right] + \sigma_d \frac{\bar{\rho}}{\omega} \frac{\partial \omega}{\partial x_k} \frac{\partial \tilde{k}}{\partial x_k} \quad (3.8)$$

where the first terms, second terms, and the third terms on the right hand side of Eq. (3.7) and Eq. (3.8) represent the production of k , ω , the diffusion term respectively, while the last term in Eq. (3.8) represents the cross-diffusion term. And the closure coefficients of these equations have the values as $\sigma_k = 0.85$, $\beta_k = 0.09$, $\alpha_\omega = 0.556$, $\beta_\omega = 0.075$, $\sigma_\omega = 0.5$, $\sigma_d = 0$. The eddy viscosity (μ_t) is formulated as follows:

$$\mu_t = \bar{\rho} \cdot \frac{\tilde{k}}{\omega} \cdot \frac{1}{\max \left[\frac{1}{\alpha^*}, \frac{\Omega \cdot F_2}{a_1 \cdot \omega} \right]} \quad (3.9)$$

Ω represents a mean rate of rotation and F_2 is the blending function. The term, α^* , is given by,

$$\alpha^* = \alpha_0^* \cdot \left(\frac{\alpha_0^* + Re_t / R_k}{1 + Re_t / R_k} \right) \quad (3.10)$$

where

$$Re_t = \frac{\bar{\rho} \cdot \tilde{k}}{\mu \cdot \omega}, \quad R_k = 6, \quad \alpha_0^* = \frac{\beta_t}{3} \quad \text{and} \quad \beta_t = 0.072$$

More additional details of the full formulation of the SST model can be found in Menter [36].

3.4 Numerical Procedures

3.4.1 Grid Generation

The computational domain and grid topology used for the simulation of spin-stabilized round bodies are discussed in this section with the focus on the grid generations that are relevant to computations performed in the subsequent chapters. In this study, there are three different types of model meshes for simulating the flow: 2-D axisymmetric for zero-lift drag coefficient, 3-D symmetric (half-model) for static aerodynamics under non-spinning condition, and full 3-D for aerodynamic coefficients under spinning conditions. The 2-D axisymmetric and 3-D multi-block structured grids were generated by a commercial grid generator Pointwise (V16).

The grid points in the wall normal direction are appropriately scaled to achieve y^+

equal to 0.5 at the wall. This value is required for accurate predictions of Magnus effects, according to DeSpirito and Heavey [39]. The grid point stretching ratio at the wall is kept at or below 1.25. The grid points in the streamwise direction are uniformly distributed except at the sharp corners where they are clustered densely to resolve the shock and expansion waves. The transverse grid lines were created in such a way that they are always approximately perpendicular to the wall surface. The distance from the inflow to the leading edge of the round, the round base to the outflow, and body wall to the far field of these models are at least $0.15D$, $6D$, and $8D$, respectively. Here, examples of the generated grids are illustrated in Figure 3.2.

Figure 3.2: The examples of 2-D and 3-D model meshes a) 2-D axisymmetric mesh for zero-lift drag coefficient and b) full 3-D mesh for spinning conditions.

3.4.1 Boundary Conditions

Regarding the boundary conditions used in the current study and subsequent chapters, the inflow and outflow boundaries are set as the pressure and temperature values based on the available wind tunnel testing and aeroballistics range conditions. The turbulent intensity and viscosity ratio at inflow condition are set as 0.5% and 10, respectively. At the walls, no-slip and isothermal (288 K) conditions are enforced. For the spinning case, a moving wall condition is used based on the given spin rate where

the body surface is rotating about the longitudinal axis of the round. Here, the spin direction and circumferential angle θ along the round surface are shown in Figure 3.3. It should be noted that the cross flow direction and spin direction are the same over the range of $\theta = 0^\circ$ - 180° , whereas for the range of $\theta = 180^\circ$ - 360° , the cross flow is in the opposite direction of the spin motion.

Figure 3.3: The spin direction and circumferential angle along the body surface

As discussed in chapter 2, to study the negative effect of the precession motion such as pitch damping moment sum coefficient ($C_{Mq} + C_{M\dot{\alpha}}$) and the changes of boundary layer, the Lunar coning methodology of Weinacht et al. [18] has been chosen as the main approach. Here, the computational domain of the pitch-damping moment sum coefficient calculation is set to rotate about an axis which is parallel to free stream velocity and coincident with the round's C.G. using the coning rate as shown in Figure 3.4. The rotating reference frame methodology is applied to perform the steady-state simulation. To obtain the pitch damping coefficient, the side moment due to steady coning motion C_n and the Magnus moment coefficient $C_{np\alpha}$ are first calculated using CFD. Equation (2.3) is then applied to solve for the pitch damping coefficient. The details to evaluate the pitch damping coefficient can also be found in the open literature such as by Weinacht et al. [18, 61] and by DeSpirito et al. [62].

Figure 3.4: An illustration of a coning motion simulation

3.5 Validation Study

3.5.1 Computational Benchmark Models

In the present study, five different available generic rounds, namely, Secant-Ogive Cylinder Boat Tail (SOCBT) [63], Secant-Ogive Cylinder (SOC) [63], Army-Navy Spinner Rocket (ANSR) [43], M910 sub-caliber [44], and 155 mm M549 Projectile [64] are considered to be the computational models for conducting the preliminary aerodynamic studies. The configurations of these round shapes are shown in Figure 3.5 and 3.6, respectively. In addition, the grid generation of these models are also shown in Figure 3.7 and 3.8. The simulation results of these models are compared with the experiment for validation. It should be noted that the experimental results of the first two models are obtained from the wind tunnel test of Sturek [63], while the aerodynamic characteristics of the last three models are obtained from the aeroballistics range [43, 44, 64]. The flight test conditions of these models are tabulated in Table 3.1. After the accuracy of the numerical method is confirmed, this method can then be applied to design the optimum round geometry.

Figure 3.5: The configuration of generic rounds for pressure distributions measurement, a) Secant-Ogive Cylinder (SOC) and b) Secant-Ogive Cylinder Boat Tail (SOCBT), dimensions in body diameter ($D=57.15\text{mm}$)

Figure 3.6: The configuration of generic rounds for forces and moments calculations, a) M910 sub caliber, dimensions in cm; b) Army-Navy Spinner Rocket (ANSR) ($l=7D$ body diameter, $D=20\text{mm}$), dimensions in body diameter; c) 155 mm M549 projectile, dimensions in body diameter ($D=155\text{mm}$)

Figure 3.7: The 3-D model meshes for pressure distributions measurement: a) Secant-Ogive Cylinder (SOC) and b) Secant-Ogive Cylinder Boat Tail (SOCBT)

Figure 3.8: The 2-D and 3-D model meshes for forces and moments calculations a) M910 full 3-D mesh, b) ANSR full 3-D for spinning, c) 2-D axisymmetric mesh with rotating band, one-half 3-D mesh (non-spinning), and full 3-D mesh (spinning) for 155 mm M549 standard round

For the aerodynamics characteristics evaluation, the first two models (SOCBT and SOC) are investigated only for the pressure distributions along the surface at finite angles of attack. For the next three models, the aerodynamic forces and moments under both static and dynamic conditions are evaluated to validate with existing aeroballistics range data [43, 44, 64]. It is useful to note that the moments obtained from CFD simulations later on are all referenced from the center of gravity.

Table 3.1: Wind tunnel testing and aeroballistics flight conditions of five-different projectiles

Flight Conditions	Generic Rounds			
	SOC/SOCBT	M910	ANSR	155 mm M549
Mach number testing	3	1.4 - 4.5	1.4 - 2.5	1.4 - 2
Angle of attack (degree)	2°, 4.2°, 10.4°	3°	2°	0°, 2°
Free stream static pressure (Pa)	8112.658	101,325	101,325	101,325
Free stream static temperature (K)	111.714	288	288	288
Non-dimensional spin rate, PD/V	0.283	0.173	0.63	0.314

3.5.2 Grid Convergence Study

A grid convergence study was carefully performed to ensure that the numerical results are insensitive to the grid resolution. This was done for all test cases except for the study of the M910 round. The model mesh of M910 was designed by referring the previous mesh size which was designed in such a way that the density of the grid can resolve the flow over the critical area as mentioned in Section 3.4.1. In this section, the 155 mm M549 round is selected as an example model to investigate grid sensitivity. The final axisymmetric grid of this model consists of 65,900 cells and the final 3D grid consists of 4,800,526 cells. An example grid resolution study of the 155 mm M549 round is tabulated in Table 3.2. From the grid independence study, it can be

seen that the flow is well resolved with the fine mesh and refining the mesh further would not provide significantly different aerodynamic coefficients.

Table 3.2: Percent differences of aerodynamic coefficients of 155 mm M549 round from the medium-sized mesh in grid resolution study.

Mesh	Maximum percent difference (%) over all Mach number range (AOA = 2°)			
	C_D	C_N	C_M	$C_{np\alpha}$
Coarse (1,127,352 cells) - Medium (2,846,300 cells)	2.3	2.32	4	6
Medium (2,846,300 cells) - Fine (4,800,526 cells)	1.2	0.5	1.5	2.5

3.5.3 Pressure Distribution Comparisons

As discussed earlier, the pressure distributions over two-different round shapes (SOC and SOCBT) are evaluated using CFD and the computational results are validated with wind tunnel test data at two flight conditions: 1) Mach 3, AOA=4.2° and 2) Mach 3, AOA=10.4°. Both flight conditions were conducted under non-spinning conditions. The results of pressure distributions over these round shapes are shown in Figures 3.9 and 3.10.

Figure 3.9 illustrates the pressure distributions along the SOC surface under the angles of attack of 4.2°. The comparisons are shown at three different circumferential angle of round surface: Lee-ward side (0°), Wind-ward side (180°), and Mid-plane (90° or 270°). It is found that the CFD provides a good agreement compared with wind tunnel test data [63] at all azimuthal stations.

Figure 3.9: The comparisons of surface pressure distributions of SOC using CFD and wind tunnel test data (Mach 3, 4.2° angle of attack)

At a larger angle of attack (10.4°), the comparisons of pressure distributions over SOCBT using are shown in Figure 3.10. It can be seen that excellent agreement is obtained between CFD and wind tunnel testing data [63], except the CFD pressure distributions near the junction region between nose-cylinder and cylinder-conical boattail which are slightly lower than wind tunnel data. However, based on the pressure distributions results, it can be proved that the CFD approach can be used with confidence to predict the shock trends over round body.

Figure 3.10: The comparisons of surface pressure distributions of SOCBT using CFD and wind tunnel test data [63] (Mach 3, 10.4° angle of attack)

Furthermore, from the pressure distributions along two-different round bodies as shown in Figure 3.9 and 3.10, it can be noticed that the imbalance between pressure coefficients over the lee-ward and wind-ward side yields a large positive normal force (or lift) over these round shape bodies. And it is also clear that the largest contribution of the normal force is generated by the secant-ogive nose. Moreover, it can be seen that the SOCBT provides a larger positive normal force (or lift) than the SOC round. This is because this round body flies at a higher angle of attack.

3.5.4 Static Aerodynamic Coefficients

The static aerodynamic coefficients of three-different round bodies: ANSR, M910, and 155 mm M549 round are investigated using CFD under both spinning and non-spinning conditions. The computational results of these round shapes are validated with the aeroballistics range data [43, 44, 64]. The static aerodynamic coefficients of three-different round bodies are shown in Figures 3.11 to 3.13.

3.5.4.1 Drag (or Axial Force)

The aerodynamic drag coefficient of ANSR, M910, and 155 mm M549 round are shown in Figures 3.11a, 3.12a, and 3.13a, respectively. These graphs show the drag coefficients as a function of Mach number at small angles of attack (3° for M910, 2° for ANSR, and 0° for 155 mm M549 round). It is found that the drag coefficient results for ANSR and for 155 mm M549 round agree well with aeroballistics range data [43, 44, 64] (within 5%) throughout all Mach numbers and the semi-empirical zero-lift drag results for 155 mm M549 round are within 10 % difference from free-flight data and CFD, while the drag results on M910 is 10% (over-predicted) at low supersonic speeds as shown in Figure 3.12a. The over-prediction in CFD may be due to a fully turbulent boundary layer being assumed in the solutions. In the real flight, the flow over the body surface may consist of both laminar and turbulent boundary layers with transition occurring at some point along the body. In addition, because the configuration of M910 sub caliber is very small, most of the flow over this round shaped body may be laminar.

3.5.4.2 Normal Force

Figures 3.11b, 3.12b, and 3.13b, shows the normal force coefficient derivative, $C_{N\alpha}$, as a function of Mach number at small angles of attack on the ANSR, M910, and 155 mm M549 generic round, respectively. At 2° angle of attack, it is found that the CFD provides a good agreement (within 3%) on normal force coefficient derivative of ANSR and 155 mm M549 round, while the normal force coefficient derivative is within 5% on M910 at angle of attack of 3° . It is also found that the normal force coefficient derivative of these rounds tends to increase once the Mach number is increased (in the low Mach number regime), while the normal force coefficient derivative tends to slightly reduce in the high Mach number regime as can be seen in

Figure 3.12b. This is because the dynamic pressure at high Mach number has a larger effect on normal force coefficient than at low supersonic speed.

3.5.4.3 Pitching Moment and Center of Pressure

The pitching moment coefficient derivative, $C_{M\alpha}$, about the center of gravity and center of pressure are plotted against the Mach number in Figures 3.11c, 3.11d, 3.12c, 3.12d, 3.13c, and 3.13d. It is found that the pitching moment coefficient derivative results of three different rounds are in good agreement with the experiments – they are within 5% for the pitching moment and within 10 % percent difference for the center of pressure throughout all Mach numbers. Moreover, it is also found that the pitching moment coefficient derivative of ANSR is larger than the others. This is because this round body has a short nose length and a long cylindrical body. Subsequently, the center of gravity (C.G.) is located far from the resultant normal force which located at center of pressure. Normally, the normal force is generated from the nose region. Thus, the center of pressure is mostly located at the nose region as can be seen in Figure 3.11d, 3.12d and 3.13d. Since the ANSR provides a highly unstable pitching moment (positive pitching moment coefficient derivative, $C_{M\alpha} > 0$), this effect reduces gyroscopic stability. That is why the ANSR requires a very high non-dimensional spin-rate to provide gyroscopic stability as compared to M910 and the 155 mm M549 round.

It should be noted that the pitching moment coefficient derivative tends to decrease with increasing Mach number. This is because the resultant force is also increased. Subsequently, increasing resultant force will results in reducing static margin (distance between center of pressure and center of gravity). Finally, the pitching moment coefficient derivative is decreased.

Regarding the effects of spinning, by comparing the static aerodynamic

characteristics of these rounds in the non-spinning and spinning cases, it is found that static aerodynamic characteristics obtained using CFD are within 4% difference between the non-spinning and spinning conditions. Thus, it can be concluded that the spinning condition has a very small effect on the static aerodynamic coefficients. For the study of the effect of precession motion, since the precession motion (coning rate) is much smaller than spin rate, the changed of boundary layer is very small compared with the changes of boundary layer under spinning motion (small angles of attack). Thus, the state of boundary layer under coning motion (at small angles of attack) has a very small effect on the performance of the round bodies. This statement is also supported by the literature studies of Weinatch et al. [18]. They found that the results of the negative effect of coning motion using CFD under inviscid flow is about the same as that in viscous flow. However, it is expected that spinning and precession motions have a large effect on the side force, yawing moment (Magnus effect) and rolling moment [3] which consequently affect dynamic stability.

3.5.5 Dynamic Aerodynamic Coefficients

The main difficulty of the validation study for dynamic aerodynamic coefficients of three different rounds is the Magnus effect calculation. This is because the validation of Magnus effect requires a large number of mesh cells, which consequently yields a very costly computation. Moreover, as discussed in Chapter 2, the Magnus effect is very difficult to predict accurately using steady-state CFD calculation for low supersonic and transonic flows. However, in the present study, the CFD calculation focused only on Magnus effect calculation at moderate supersonic speeds. The results of dynamic aerodynamics coefficients are shown in Figures 3.14 and 3.15.

Figure 3.14 shows the dynamic aerodynamic coefficients (Magnus moment coefficient derivative, $C_{np\alpha}$ and roll damping coefficient, C_{lp}) as a function of Mach

number at small angles of attack on the ANSR model and M910 sub caliber. The Magnus moment derivative and pitch damping moment sum coefficient for the 155 mm M549 generic round are shown in Figure 3.15. The CFD results of these shapes are within 17% difference for Magnus effect and within 15% for roll damping at low supersonic speeds ($M < 1.8$), while the pitch damping moment sum coefficients of the 155 mm M549 round agree better with the aeroballistics range (within 10%). According to common practice of CFD calculations of Magnus effect and roll damping, 20% errors are deemed reasonable results when compared with aeroballistics range [40]. However, the errors in Magnus effect and roll damping can be reduced to within 10% in the supersonic range of Mach 2 to 4 as shown in Figure 3.14c and 3.14d. Thus, the CFD technique that has been used in this study is considered sufficient for evaluating both static and dynamic aerodynamic coefficients in the moderate supersonic range.

a)

b)

c)

d)

Figure 3.11: The static aerodynamic coefficients for ANSR (non-spinning and spinning) at 2° angle of attack versus Mach number, a) drag coefficient, b) normal force derivative, c) pitching moment derivative, d) center of pressure

Figure 3.12: The static aerodynamic coefficients for M910 sub caliber (spinning) at 3° angle of attack versus Mach number, a) drag coefficient, b) normal force derivative, c) pitching moment derivative, d) center of pressure

Figure 3.13: The static aerodynamic coefficients for 155 mm M549 projectile (non-spinning and spinning) at 2° angle of attack versus Mach number, a) drag coefficient, b) normal force derivative, c) pitching moment derivative, d) center of pressure

a)

b)

c)

d)

Figure 3.14: The dynamic aerodynamic coefficients for spinning versus Mach number, a) Roll damping moment derivative for ANSR, b) Magnus moment derivative for ANSR, c) Roll damping moment derivative for M910, d) Magnus moment derivative for M910

a)

b)

Figure 3.15: The dynamic aerodynamic coefficients for 155 mm M549 round: a) Magnus moment derivative and b) pitch damping moment sum coefficient at 2° angle of attack.

Chapter 4

Aerodynamic Performance Improvement

In this chapter, to improve the aerodynamic performance of the generic round bodies, an optimization technique coupled with computational fluid dynamic (CFD) is developed to optimize the round body to obtain the best aerodynamic performance in terms of minimum drag and high stability. The procedure of this method is discussed in the first two sections. In the next section, the flow physic study of generic rounds is also addressed. Finally, based on the understanding of the flow mechanism over the boattail, the boattail juncture shaping technique is employed to further enhance the aerodynamic performance.

4.1 Optimization Procedure

As mentioned earlier, the aerodynamic drag and flight stability are the main factors for the aerodynamics performance on the standard 155 mm round shape. Thus, to improve the aerodynamic performance of this round flight body, the present work proposes a new, simpler and more accurate optimization procedure in which the stability issue is incorporated through Computational Fluid Dynamics analysis (CFD) compared with the earlier theoretical optimization technique.

The optimization procedure is outlined in Figure 4.1. The procedure begins with the initialization of a generic round geometry. The drag experienced by this body at zero angle of attack is then computed using a McCoy's semi-empirical method [55].

Next, the optimization method is applied to minimize the drag based on the specified constraints. This is achieved iteratively where the drag is re-evaluated whenever a new geometry is obtained. After the optimization is finished, the static and dynamic aerodynamic coefficient derivatives of the optimum shape are determined by means of CFD in order to analyze the stability. If the stability requirement is not met, further changes are made on the geometry to find a best compromise between minimum drag and high stability. Finally, experimental data and CFD are used to validate the accuracy of the resulting configuration. Each step in the optimization procedure is explained in more details in the subsequent section.

Figure 4.1: The flow chart of the optimization procedure

4.1.1 Initial Design

An initial design of a generic round is needed before the optimization method can be employed. As shown in Figure 2.1, a typical round body consists of a nose, a cylindrical body and a tail. While the design of the cylindrical body is straight forward, the shape of the nose and the tail deserve some attentions. As the wave drag on the nose shape is the main contributor to the pressure drag, the nose profile will first be examined. In the present study, the von Karman ogive and power law nose profiles [65, 66] are both considered as the initial nose shape configurations because they provide minimum wave drag at supersonic speed and hypersonic speed, respectively. The von Karman ogive was derived based on slender body theory and the power law profile was computed from Newtonian impact theory. The detailed formulations are given in the Appendix A.

Regarding the tail shape, there are a number of different concepts that can be used to reduce the drag generated at the base region such as the use of boattail afterbody, slot cavity on the base, and base bleed [45, 46, 47, 48]. In this study, the conical boattail was selected because it provides superior base drag reduction [46] and its mechanical structure is stronger than the other concepts. While the conical boattail afterbody was investigated extensively in the literature [3, 45, 46, 64], this shape has yet to be fully optimized in combination with the nose and the cylindrical body. The initial conical boattail configuration is assumed to have no boattail angle.

4.1.2 Drag Computation based on Semi-Empirical Method

In addition to the initial shape, the optimization procedure also requires a way to evaluate the zero-lift drag after every iterative step. This is done in this study by employing the semi-empirical method from McCoy [55]. As discussed in Chapter 3, the overall drag of this semi-empirical method is divided into three components,

namely, the pressure drag, the skin friction drag and the base drag. Each component is then calculated by explicit formulations based on theory and empirical database of generic round shape. The detailed formulations are given in the Appendix A.

In comparison to the drag computation method from Nicolas [49] that was discussed in Chapter 2, the use of McCoy's method in an optimization loop is new and provides several advantages. First of all, the explicit formulations of pressure drag and skin friction drag are simpler, thus, the solutions can be obtained more rapidly. Second, the pressure drag computation proposed by McCoy is based on a second-order correlation which is more accurate than the first-order formulation of Nicolas [49].

4.1.2 Optimization Technique

The goal of the optimization method is to determine an optimum shape (or optimum round configuration) that can provide minimum drag at zero angle of attack. In the present study, the Active Set Algorithm: medium-scale optimization and Sequential Quadratic Programming (SQP) [67] is selected to be the main approach to optimize the objective function because of its simplicity, speed and accuracy in comparison to the Eulerian optimization scheme of Nicolas [49]. Furthermore, the current optimization technique is also suitable for a linear objective function which is computed from McCoy's semi-empirical method. The SQP optimization problem can be expressed as follows:

$$\left. \begin{array}{l} \min_{\mathbf{d} \in \mathbf{x}_k} \left[\frac{1}{2} \mathbf{d}^T \mathbf{H}_k \mathbf{d} + \nabla f(\mathbf{x}_k) \cdot \mathbf{d} \right] \\ \text{Subject to } \nabla g(\mathbf{x}_k) \cdot \mathbf{d} + g(\mathbf{x}_k) = 0 \end{array} \right\} \quad (4.1)$$

where $\mathbf{H}_k$ is the Hessian matrix, $\mathbf{d}$ is the steepest descent, $f(\mathbf{x})$ is the object function, $g(\mathbf{x})$ is the constraint function and k is the first solution. The bold face symbols

represent vector quantities. To obtain the extremum, Eq. (4.1) needs to be put in the Lagrangian form as expressed below

$$\Lambda(\mathbf{d}) = \frac{1}{2} \mathbf{d}^T H_k \mathbf{d} + \nabla f(\mathbf{x}_k) \cdot \mathbf{d} - \lambda (\nabla g(\mathbf{x}_k) \cdot \mathbf{d} + g(\mathbf{x}_k)) \quad (4.2)$$

The constrained minimum of Eq. (4.2) occurs at $\nabla \Lambda(\mathbf{d}) = 0$ (derivative of Eq. (4.2) with respect to $\mathbf{d}$), thus we have

$$H_k \mathbf{d} - \lambda \nabla g(\mathbf{x}_k) = -\nabla f(\mathbf{x}_k) \quad (4.3)$$

where λ is the Lagrangian multiplier. Eq. (4.3) is the final form of SQP optimization method that needs to be solved. In the current study, the object function $f(\mathbf{x})$ of the optimization method (Eq. 4.3) is the total zero-lift drag coefficient based on McCoy's solution [55] (Appendix A). This can be expressed as a function of the generic round geometry as follows:

$$f(\mathbf{x}_k) = C_{D_0}(L_N, L_{CYL}, L_{BT}, \beta) \quad (4.4)$$

where $\mathbf{x}_k$ is the vector of the geometric parameters of the generic round configuration (nose length L_N , cylindrical body length L_{CYL} , boattail length L_{BT} , and boattail angle β , respectively). Eq. (4.4) above is subjected to the equality constraint equation (the total length of generic round L) as expressed below

$$g(\mathbf{x}_k) = L_N + L_{CYL} + L_{BT} = L \quad (4.5)$$

Eq. (4.3) above solves a quadratic programming (QP) subproblem at each iteration. The solution is used to form new iterates and can be expressed as follows

$$\mathbf{x}_{k+1} = \mathbf{x}_k + a_k \mathbf{d}_k \quad (4.6)$$

The step length parameter a_k is determined by an appropriate line search procedure so that a sufficient decrease in a merit function is obtained. The matrix H_k is a positive definite approximation of the Hessian matrix of the Lagrangian function. The code updates an estimate of the Hessian of the Lagrangian at each iteration using the BFGS formula as expressed in Eq. (4.7) below

$$H_{k+1} = H_k + \frac{\mathbf{Q}_k \mathbf{Q}_k^T}{\mathbf{Q}_k^T \mathbf{s}_k} - \frac{H_k^T \mathbf{s}_k \mathbf{s}_k^T H_k}{\mathbf{s}_k^T H_k \mathbf{s}_k} \quad (4.7)$$

where

$$\mathbf{s}_k = \mathbf{x}_{k+1} - \mathbf{x}_k$$

$$\mathbf{Q}_k = [\nabla f(\mathbf{x}_{k+1}) + \lambda \nabla g(\mathbf{x}_{k+1})] - [\nabla f(\mathbf{x}_k) + \lambda \nabla g(\mathbf{x}_k)]$$

Since the drag is calculated from McCoy's solution, the range of application of the current method is limited by the valid range of that semi-empirical method. This provides the upper and lower limits of the round geometry and flight conditions (see Appendix A). The MATLAB codes to recall the SQP optimization technique for evaluation the optimum geometry is available in Appendix B.

4.1.3 Stability Analysis

The drag optimization carried out in the previous section does not ensure stability during flight at finite angles of attack. This is because the center of pressure of the round body x_{cp} is often located in front of the center of gravity, leading to an unstable pitching moment as shown in Figure 2.2. Here, the center of pressure is calculated as the ratio of the pitching moment and the normal force coefficient with respect to the center of gravity. As discussed in Chapter 2, for a round body without control surfaces, spinning about the axial axis is normally required to overcome the stability issue. The main factors governing the stability of a spin-stabilized flight body are the

gyroscopic static stability parameter S_g and the dynamic stability factor S_d as expressed in Eq. (2.11) and (2.13).

The above stability requirements are enforced by redesigning the optimized zero-lift body through CFD analysis. It can be seen from Eq. (2.11) that the gyroscopic static stability depends on the flight body's spin rate P , moment of inertia I_x , I_y and pitching moment coefficient derivative $C_{M\alpha}$. Within this context, the bodies are assumed to be made from steel with uniform density so that the center of gravity and the moments of inertia can be calculated. If the moment of inertia remains constant, gyroscopic stability can be improved by either increasing the spin rate or reducing the pitching moment. Increasing the spin rate produces large Magnus effects on the body, which in turn decreases dynamic stability [68]. Thus, decreasing the pitching moment is the preferred approach to improve gyroscopic stability. The pitching moment can be decreased by reducing the distance between the center of pressure and the center of gravity. Normally, the center of pressure of spin-stabilized rounds is located in front of the center of gravity. There are two possible ways to shift the center of pressure on the body – by increasing the lift on the nose region (i.e., extending the nose length) and by reducing the negative lift on the boattail afterbody [68].

For the dynamic stability factor S_d , although it is dependent on many aerodynamic coefficients as shown in Eq. (2.13), the Magnus moment derivative $C_{np\alpha}$ and the pitch damping coefficient $(C_{Mq} + C_{M\dot{\alpha}})$ are the main components that strongly influence the dynamic stability [68]. Therefore, it is usually necessary to adjust these coefficients so that the stability requirements in Eq. (2.13) are met. According to Mason et al. [69], for a good design of spin-stabilized round body, the dynamic stability factor should be about one. Furthermore, previous studies [18, 70] also showed that a drastic change of the Magnus force and the pitch damping coefficient can be obtained by changing the

shape of the conical boattail. For a fixed base-body area ratio, a longer conical boattail causes larger Magnus effect and significantly smaller pitch damping moment coefficient. This provides a convenient tool to alter the dynamic stability factor, which will be discussed later.

4.2 Optimization Demonstration

4.2.1 Benchmark Configurations and Design Considerations

In this section, the optimization procedure will be applied to design an optimum round that can provide better performance and higher stability than a benchmark configuration, which is the standard 155 mm M549 projectile [64]. The goal is to achieve improved performance for the same body diameter and length. Furthermore, the new round body should operate efficiently over a wider range of Mach numbers (from 1.5 to 4) and angles of attack (from 0° to 10°). The benchmark 155 mm round consists of a secant-ogive nose, a cylindrical body, a rotating band and a conical boattail (see in Figure 3.6c) and is well known for good performance

4.2.2 Drag Minimization

Once the input parameters of nose shape profiles, conical boattail configurations, and flight conditions are set in the total drag equation, the optimization method will be recalled to optimize the optimum 155mm round shape configurations. The object function of total zero-lift drag coefficient is solved in the current study using the upper and lower bounds of round shape configurations (boattail length = $2D$ where $D = 155$ mm and boattail angle = 10°). The constraint is the total length of the body, which must be the same as that of the benchmark configuration ($L = 5.645D$). The upper and

lower limits are based on the valid range of the semi-empirical method. Moreover, it was found during the optimization that the optimum body geometry obtained using the power law nose shape resulted in a total drag that is slightly less than that for the von Karman ogive for all Mach numbers. Therefore, only the results using the power law nose shape are discussed below.

The resulting optimum round body at various Mach number is shown in Figure 4.2a in terms of minimum zero-lift drag and optimum nose length. It can be seen that, at every individual Mach number, minimum drag is obtained when the round body has no cylindrical portion ($L_{CYL} = 0$). Furthermore, the optimum nose length and boattail angle increase with increasing Mach number. Based on this result, a least-square fitting is performed to determine a fixed optimum geometry that can provide zero-lift drag closest to the minimum value over the whole range of Mach numbers. The final round body has a nose length (L_N) of $3.8D$ and a boat tail angle of 5° and produces a total drag that is within 2% of the minimum value at every Mach number. Comparison of drag coefficients obtained by this fixed optimum body and the actual minimum drag at each Mach number is illustrated in Figure 4.2b where close agreement can be observed. The geometry of the optimum round body is given in Figure 4.3.

Figure 4.2: (a) Drag coefficient of optimal round configurations at various Mach numbers. (b) Comparison of drag coefficient on optimum 155mm round and fixed optimum geometry using the optimization method. Total length = 5.645D.

Figure 4.3: The optimum 155 mm round. The nose shape is based on Newtonian impact theory (total length 5.645D). C.G. is the center of gravity.

As the next step, CFD is used to validate the accuracy of the semi-empirical drag evaluation and to compare the performance of the optimum round body to the benchmark configuration. It should be noted that the numerical procedures of CFD to investigate the aerodynamic performance have already been discussed in Chapter 3. Figure 4.4 shows the drag predictions by CFD and the semi-empirical method for both the benchmark configuration and the optimum design. The symbols are the measurements of the benchmark configuration taken from the free-flight aeroballistics range data [3, 64]. From the computational and semi-empirical results of the benchmark configuration obtained previously in Chapter 3, it can be seen that the CFD

results agree well with the free-flight data to within 5 % (over-predicted) and the semi-empirical method results are within 10% difference from the free-flight data and CFD. Thus, it is concluded that the current optimization procedure can provide not only quick solutions but also reasonable accuracy. Furthermore, from the comparison of the aerodynamic performance between the optimum geometry and the benchmark round, it is found that the optimum geometry yields lower drag by up to 24% throughout all Mach numbers. Since the drag coefficient varies linearly at small angle-of-attack, it is expected that this optimum shape also provides minimum drag at small pitching angles. Therefore, the reduced drag of the optimum body implies a longer flight range than the benchmark round body.

Figure 4.4: Comparison of the drag coefficient of the benchmark and optimum 155 mm rounds.

The general flow features and the reason why the optimum shape provides lower drag than the benchmark round are briefly described here. The Mach number contours and pressure distributions over the optimum and benchmark configurations are illustrated in Figure 4.5. This flow is characterized by several shock waves and expansion waves emanating from the sharp corners of the bodies. At the base, there

exists a strong recirculation region associated with the sudden expansion and recompression waves. The pressure rises across the shock wave and drops across the expansion wave. Along the nose, the cylindrical body and the conical boattail, pressure is gradually increased or decreased. These changes are due to the three-dimensionality of the flow and will be explained in more details in Section 4.3. Moreover, as shown in Figure 4.5b, it should be noted that the fluctuation of pressure coefficient over the standard round is caused by the rotating band configuration. The configuration of rotating band is shown in Figure 3.6c. From this figure, it can be seen that the rotating band consists of many discontinuities of the curve over the round body that causes strong compression shocks and rapid expansion waves at supersonic speeds. This leads to obtain the sudden rises and abruptly drops of the pressure coefficient across this area". The positive pressure coefficient on the nose and the negative pressure coefficient on the boattail and the base are the contributors of the pressure drag and these coefficients are given in Table 4.1.

It can be seen that the pressure drag coefficient on the nose of the optimum body is significantly smaller than that of the benchmark round due to weaker shock at the leading edge. Furthermore, the base pressure drag of the optimum body is also significantly smaller because the base area is smaller and the average base pressure is larger. These reasons explain the superior performance of the optimum shape. It should be noted that the pressure drag is dominant over the skin friction drag, which is only 10 – 13 % to the total drag as shown Table 4.1.

Figure 4.5: Mach contours and pressure distributions over two-different flight bodies (Mach 2 and $\alpha = 0^\circ$). a) Optimum 155mm round and b) benchmark round.

Table 4.1: Breakdown of pressure drag coefficient on the optimum and benchmark rounds

Drag Coefficient		Round shape	
		Optimum 155mm round	Benchmark 155mm M549 round
Pressure drag coefficient	Nose (Nose drag/qs)	0.053	0.098
	Cylindrical body (Cylindrical body drag/qs)	-	0.0007
	Boattail (Boattail drag/qs)	0.05185	0.046
	Base (Base drag/qs)	0.0689	0.1
Skin friction drag coefficient	(Skin friction drag/qs)	0.0265	0.0353

4.2.3 Comparison with the Previous Optimization Technique

Nicolas [48] also employed his optimization procedure to design a similar round body. However, the length constraint of the body was $5D$, which is smaller than the length of the optimum shape obtained in the previous section ($5.645D$). Since it is interesting to compare the performance and accuracy of the current optimization procedure and that proposed by Nicolas, another optimum shape with an overall body length of $5D$ was determined using the present method. It was found that this shape is almost identical to the shape obtained by Nicolas [49], indicating that both methods provide similar predictions of the optimum geometry. It should be noted, however, that the current technique is simple and the result is obtained rapidly, after approximately 10 iterations. The current MATLAB implementation (version R2008) requires in between 1 to 2 seconds on a normal personal computer to obtain an optimum geometry for a given flow Mach number. Further time reductions can be expected using Fortran or C++. In addition to the time advantage of the current technique, the aerodynamic prediction was found to be much closer to what is predicted by CFD as shown in Figure 4.6. This is because the pressure drag is computed from a second-order correlation which is more accurate than the first-order formulation of Nicolas [49]. However, since the drag computation of the current optimization procedure is developed from the correlation method, the valid range of round geometries and Mach number (input parameters) is lower than the exact solution of Nicolas. Another new feature of this procedure is the stability analysis and improvement, which will be discussed in the subsequent section.

Figure 4.6: Comparison of predicted drag using the present optimization method, Nicolas' optimization technique [49], and CFD. Optimum shape with body length of $5D$.

4.2.4 Stability Analysis

Although the optimum round body with a body length of $5.645D$ shown in the previous section is the best configuration for drag minimization, the shape may not be practical if it is to be spin-stabilized because the lack of a cylindrical portion would result in large wobbles in the launch tube. Moreover, as shown in Figure 4.3, the center of gravity is located in front of the maximum diameter position, which will generate a large moment in the launch tube. Subsequent to launching, the body will further encounter dynamic instability during flight. This is illustrated in the stability criterion plot (Figure 4.7) where it can be seen that the optimum round is dynamically unstable at high Mach numbers (Mach 3 and 4), while the benchmark geometry is still within the stability bound throughout all Mach numbers. Therefore, the optimum shape needs to be modified to achieve better stability.

Figure 4.7: Comparison of the dynamic stability factor between the optimum shape and the benchmark round, $PD/V = 0.314$.

Prior to improving the stability of the optimum shape, it is useful to identify and understand the general flow features of the round body at finite angles-of-attack. Figure 4.8 shows the Mach number contours and the pressure distribution around the optimum shape at Mach 2 and $\alpha = 2^\circ$. The flow features are very similar to those at $\alpha = 0^\circ$ (see in Figure 4.5); except that the shock and expansion strengths are not the same on the lee-side and the wind-side and the recirculation region at the base is not symmetric over the body centerline anymore. The pressure difference between the lee-side and the wind-side results in a positive lift on the nose area and a negative lift on the boattail area. The presence of a region with negative lift may seem counter-intuitive. However, this is entirely possible due to the pressure increase on the boat tail surface downstream of the expansion fan. As mentioned earlier, the reason for this pressure rise will be explained in details in Section 4.3. For the spinning round, the spinning motion at finite angle-of-attack induces boundary layer distortions, leading to an asymmetric pressure and skin friction distributions with respect to the pitch plane. This consequently results in a side force called Magnus force. While normally an order

of magnitude smaller than the lift and drag forces, the Magnus force causes an induced Magnus moment, which is an important parameter in defining the dynamic stability of the round body.

Figure 4.8: Mach contours and pressure distributions on the optimum body at Mach 2 and $\alpha = 2^\circ$.

4.2.5 Stability Improvement

The design will first be modified to reduce wobble if the body is to spend part of its travel in the launch tube. This is accomplished by adding a cylindrical part and protruding it forward to cover the center of gravity location, and by extending it to the rear part of the body in order to reduce the negative lift which in turn increases the total lift. Increasing the length of the cylindrical portion of the body, however, will increase the pressure drag and skin friction drag. Therefore, further optimization of the modified optimum shape will be carried out to obtain the best compromise between low drag and high stability. Moreover, for a good body configuration design, the payload or volume of the flight body needs to be kept as high as possible. In the present study, in order to increase stability, the optimum 155 mm body is modified into three different new shapes that have the same nose length of $3.4D$ to ensure that the center of gravity falls within the cylindrical body. The three body shapes have

cylindrical body lengths consisting of $0.4D$, $0.8D$, and $1.5D$. These bodies are then optimized using the optimization procedure to evaluate the optimum boattail angle for minimum drag. From the results of the optimization method, the optimum boattail configurations of the three modified bodies are 5° , 6° , and 7° , respectively, as shown in Figure 4.9 and the physical characteristics of the modified optimum shapes are tabulated in Table 4.2. It can be seen that, when the cylindrical portion is longer, the center of gravity moves downstream and the moments of inertia and the volume are larger.

Figure 4.9: The physical characteristics of the modified optimum 155 mm flight bodies.

Table 4.2: Physical characteristics of the modified optimum 155 mm flight bodies

Domain	M549 155 mm round	Modified optimum 155 mm flight body (Body diameter (D)=155 mm)		
		Cylinder length= $0.4D$	Cylinder length= $0.8D$	Cylinder length= $1.5D$
Total length (D)	5.645	5.645	5.645	5.645
Center of gravity (C.G.)	3.5D	$3.44D$	$3.479D$	$3.59D$
Moment of inertia I_x (kg m ²)	0.15	0.1124	0.1233	0.1524
Moment of inertia I_y (kg m ²)	1.93	1.437	1.5024	1.697
Volume (m ³)	0.0106	0.00898	0.00938	0.01016

After optimizing the three different modified rounds, CFD is again used to verify the aerodynamic performance of these shapes with respect to the benchmark 155 mm round. Figure 4.10 shows the numerical results and the comparison of aerodynamic performance in terms of the zero-lift drag coefficient on various modified optimum bodies using both CFD and the semi-empirical method. It is found that the modified optimum body with a cylindrical length of $0.4D$ can reduce total drag by up to 20% throughout the Mach number range, while the bodies with cylindrical body lengths of $0.8D$ and $1.5D$ can reduce the total drag by 15% and 10%, respectively as compared with the standard 155 mm round. As expected, a longer cylindrical portion leads to less drag reduction. It is also found that the aerodynamic drag predictions using the semi-empirical method compared well with CFD to within 3% difference.

Figure 4.10: Comparison of the drag coefficient of the modified generic rounds.

Figure 4.11a illustrates the normal force (i.e., lift) derivative on different modified bodies and the benchmark round at small angles of attack ($\alpha = 2^\circ$ to 5°). It can be seen that the modified optimum body with cylindrical length of $1.5D$ yields the largest total lift derivative (about 5% larger than the standard round and 20–26% larger than the other modified shapes), while both of the other modified optimum bodies ($0.4D$ and $0.8D$ cylindrical length) provide smaller total lift derivative than the benchmark round. This is so because the longer conical boat tails of the modified shapes with $0.4D$ and $0.8D$ cylindrical lengths produce larger negative lift than the benchmark round although the nose of these bodies are longer. Since the pitching moment is dependent on both the lift force and the distance between the center of gravity and center of pressure, the pitching moment derivatives with respect to the center of gravity for all the modified bodies are about the same despite the strong difference in the lift derivatives. These bodies yield smaller pitching moment derivatives (about 17–29%) in comparison to that produced by the benchmark round. For the modified optimum body with $1.5D$ cylindrical length, the pitching moment derivative is the smallest compared with bodies of other cylindrical lengths at Mach numbers of 1.8 to 2 and is slightly higher at Mach 2 to 4 as shown in Figure 4.11b.

a)

b)

Figure 4.11: Comparison of the normal force and pitching moment derivatives on the modified generic rounds.

Figure 4.12 shows the comparison of the gyroscopic stability factor for different flight bodies by using the same non-dimensional spin rate ($PD/V = 0.314$) as that for the free-flight aeroballistics range data for the reference 155 mm M549 round [64]

and calculated using Eq. (2.11). It is found that the modified optimum body with a cylindrical body length of $1.5D$ results in the largest gyroscopic stability factor. The gyroscopic stability is increased over that for the benchmark round by about 46% using the modified shape with $1.5D$ cylindrical body, 15% for cylindrical body length of $0.8D$, while the $0.4D$ cylindrical body shape has a slightly smaller gyroscopic stability than the reference round. This is because the ratio of moments of inertia of the $0.4D$ cylindrical body is significantly smaller in comparison to that of the other bodies. Furthermore, the longer boat tail of the $0.4D$ cylindrical body may induce large Magnus effects, which in turn reduce dynamic stability. It is concluded that, for stability reason, the cylinder length of the modified optimum shape should be in range of $0.8D$ to $1.5D$, which can provide drag reduction up to 15% and increase of gyroscopic static stability up to 46% for the same spin rate employed in the free-flight aeroballistics range [64] over the Mach numbers studied. Moreover, these modified body shapes can provide a volume or payload that is almost the same as the benchmark round (see in Table 4.2).

Figure 4.12: Comparison of the gyroscopic stability factor on the modified generic rounds, $PD/V = 0.314$.

However, the high gyroscopic stability obtained previously using two modified optimum rounds does not ensure dynamic stability during flight. Thus, to design a realistic spin-stabilized round, the dynamic stability needs to be analyzed. Shown in Figure 4.13 is the comparison of the Magnus moment derivative and the pitch damping moment coefficient of three different rounds; standard 155 mm M549 round and two modified optimum rounds with different cylindrical body lengths ($0.8D$ and $1.5D$). It should be noted that the results for the modified optimum body with a cylinder length of $0.4D$ are not shown in the figure since the payload of this round is relatively small compared with the benchmark configuration. From Figure 4.13, it is found that the Magnus effects calculated using the current computational approach agree well with the aeroballistics range data [64], while the pitch damping moment coefficient prediction also falls within the accuracy range of the available data. In addition, it is also found that the modified optimum geometry (cylinder length = $1.5D$) provides the lowest Magnus effect compared with the others, while the largest Magnus moment derivative is generated by the standard 155 mm round. As expected, when comparing the two modified optimum rounds which have about the same base diameter ratio (d_b), the shorter cylindrical body yields a larger Magnus force which leads to the generation of a larger Magnus moment derivative, while the pitch damping moment coefficient is smaller as can be seen in Figure 4.13a. The Magnus moment derivative and the pitch damping moment coefficient are reduced by about 29% and increased by up to 7%, respectively, when comparing the modified optimum body with a cylinder length of $1.5D$ to that with a shorter cylindrical body ($0.8D$).

Figure 4.13: Comparison of the dynamic aerodynamic coefficients on the modified generic rounds, a) pitch damping moment coefficient ($C_{Mq} + C_{M\dot{\alpha}}$) and b) Magnus moment derivative, $PD/V = 0.314$.

Once the Magnus moment derivative and pitch damping coefficient are obtained, the dynamic stability factor (S_d) can be determined. Figure 4.14 shows the comparison of the dynamic stability factors of three different flight bodies computed using Eq. (2.13) and they are plotted in terms of the dynamic stability criterion according to Eq. (2.15). It is found that the two modified optimum rounds are dynamically stable throughout all design Mach numbers and that the stability results cluster far below the

stability boundary as can be seen in Figure 4.14. Thus, the results demonstrate that both modified optimum geometries (cylinder length = $0.8D$, $1.5D$) can be used without any concerns on flight stability.

Figure 4.14: Comparison of the dynamic stability factor on the modified generic rounds, $PD/V = 0.314$.

4.3 Effect of Axially Symmetric Flow

This section is dedicated to the explanation of an intriguing phenomenon observed earlier: the effects of axially symmetric geometry on the flow field. Understanding this phenomenon will explain the gradual change of the wall pressure on the cylindrical body and the boat tail. These pressure changes may seem to be counter-intuitive because the direction of change is opposite to what is expected downstream of an expansion fan. For example, one can see that the pressure drop across the expansion fan on the boattail is followed by a gradual increase in Figure 4.5 and 4.8. This phenomenon is also responsible for the negative lift on the boattail, which makes the round body more unstable when flying at finite angle-of-attack. To explain an in-depth detail of this flow physics, the six different assumptions which would cause adverse pressure gradient are listed as follow

- Flow separation over the conical boattail
- Embedded shock at finite angle of attack
- The difference between planar (2D) and axisymmetric flow
- Viscous effect compared with inviscid calculation
- Shock Wave Boundary Layer Interaction (SWBLI)

From the assumptions as mentioned above, to examine the flow details case by case, the CFD steady-state simulation as discussed in chapter 3 is applied. The SOCBT projectile (see Figure 3.5b) and optimum 155mm round (see Figure 4.9) are used as example round geometries. In addition, a transient flow simulation is also performed to examine any unsteady flow phenomenon which can be used to compare directly with the steady one.

Based on CFD calculation as shown in Figure 4.15 and 4.16, at supersonic speeds (Mach 2 and 3), it is found that the flow over the conical boattail with small boattail

angle for the two different models remains attached at zero to finite angles of attack. In addition, it is interesting that both of the steady and unsteady simulations provide the same trend of boundary layer profile as shown in Figure 4.15. This means that the flow over the rounds is quite steady. However, there are slightly differences of these boundary layer profiles over this region as shown in Figure 4.15. The differences would be caused by the flow unsteadiness at the base region. Nevertheless, from these figures, it is evidence that there are no flow separations existing on the current geometries. Thus, the adverse pressure gradient over the conical boattail is not caused by the flow separation.

Figure 4.15: The comparison of boundary layer velocity profiles over the modified optimum 155mm round (1.5D) using two different simulations: Steady and Unsteady simulations (at zero AOA, Mach 3): a) $X/D=4.95$ and b) $X/D=5.6$

Figure 4.16: The comparison of boundary layer velocity profiles over the round bodies between SOC and SOCBT projectile at three different locations (at angle of attack 4.3° , Mach 3): a) $X/D=4.9$, b) $X/D= 5.6$, and c) $X/D=5.95$

For the generation of weak embedded shocks over the round, it is found that the embedded shocks are formed over the round only at finite angles of attack beyond 7° . The formation of weak embedded shocks at two different locations are shown in Figure 4.17. Thus, since the adverse pressure gradient is generated at both zero and finite angles of attack, the generation of the embedded shock is definitely not related to this phenomenon.

Figure 4.17: The formation of weak embedded shocks over the SOCBT projectile at angle of attack of 10.4° (Mach 3).

To study the difference between the symmetric and axisymmetric flow over the boattail configuration, CFD based on the steady full-viscous and inviscid calculation are used. The two different bodies: 2D symmetric body and round shape are chosen as example geometries. These bodies have the same boattail angle of 7° as shown in Figure 4.18.

Figure 4.18: Two different bodies: SOCBT projectile (left) and SOCBT 2D symmetric body (right)

From the CFD results as shown in Figure 4.19, at zero angle of attack, it is found that viscous equation and inviscid calculation provide the same trend of pressure

distribution over boattail configuration of two different bodies. From this figure, it can be seen clearly that the static pressure is gradually increased (adverse pressure gradient, $dp/dx > 0$) along the cylindrical body and conical boattail for the round body, while the static pressure is almost constant along these area (zero pressure gradient, $dp/dx=0$) for the 2D symmetric body. Thus, it is evident that the gradual increase of static pressure over the expansion flow region is a special flow phenomenon only for the round shape body. This clearly shows that the SWBLI is not the underlying reason for the flow phenomenon since the pressure rise is not due to viscous boundary layers or their separation. It is the round geometry that introduces a pressure gradient even in inviscid flow.

Figure 4.19: the comparison of pressure distributions along two different bodies (zero AOA): a) SOCBT projectile (round body) and b) 2-D symmetric body

To explain the reason why the adverse pressure gradient is generated only for the round shape body, Figure 4.20 shows the comparison of numerical flow visualization over the boattail configuration of these bodies. It is found that the stream line (flow direction) over the 2D symmetric body is parallel to the wall and the compression Mach waves cannot be generated over this area (see in Figure 4.20b), while the flow direction over the conical boattail tends to slightly diverge from the wall due to the

three dimensionality effect. The slight change in the flow direction generates the weak compression Mach waves which consequently yields the gradual increasing of the static pressure along conical boattail as shown in Figure 4.20a.

Figure 4.20: The comparison of the flow over the boattail configuration of two different bodies using the same boattail angle 7° (Mach 2, zero angle of attack): a) round body and b) 2D symmetric body.

To explain the adverse pressure gradient over the conical boattail mathematically, the equations of motion for an inviscid, irrotational flow in the axisymmetric streamline coordinates [71] is considered:

$$\frac{M^2 - 1}{w} \frac{\partial w}{\partial s} - \frac{\partial \theta}{\partial n} = \frac{\sin \theta}{r} \quad (4.8)$$

$$\frac{1}{w} \frac{\partial w}{\partial n} - \frac{\partial \theta}{\partial s} = 0 \quad (4.9)$$

where (w, θ) defines the velocity magnitude and direction with respect to the axis of symmetry. s and n are the streamline coordinates. These variables are illustrated in

Figure 4.21 below.

Figure 4.21: Streamline coordinates for axially symmetric flow.

Across a shock wave or in the vicinity of an expansion corner where the expansion fan is infinitesimally small, the change of the velocity vector (w, θ) takes place over a very short distance so that the partial derivative of w with respect to s is much larger than $\sin \theta / r$. Therefore, the equations of motion can be simplified as follows:

$$\frac{M^2 - 1}{w} \frac{\partial w}{\partial s} - \frac{\partial \theta}{\partial n} = 0 \quad (4.10)$$

$$\frac{1}{w} \frac{\partial w}{\partial n} - \frac{\partial \theta}{\partial s} = 0 \quad (4.11)$$

The above equations are exactly the same as those for a planar flow [71], i.e., for infinitely large r . Thus, the flow in these regimes is *locally* two-dimensional and can be computed by the well-known oblique shock wave theory and Prandtl-Meyer expansion theory. For the flow far from the expansion corner, however, $\sin \theta / r$ in Eq. (4.8) cannot be neglected because the distance over which expansion occurs is rather long. This leads to the phenomenon that the pressure change across the expansion fan is dependent on the geometry of the body. This is illustrated in Figure 4.22 by investigating the pressure coefficient along two different streamlines across the expansion fan of the optimum body without the cylindrical portion. Streamline A is

close to the wall where the pressure drop can be described by Prandtl-Meyer theory. Streamline B is far away from the wall where the pressure drop is significantly smaller than what is predicted by the theory. Since the wall pressure downstream of the expansion corner is initially smaller than the pressure in the far field, a series of Mach waves are needed along the wall surface to ensure that the pressure difference will vanish far downstream of the corner. This clearly explains the gradual increase of wall pressure on a convex surface in axisymmetric geometry. It is this pressure increase that is responsible for the drag reduction obtained by using a conical boattail.

Figure 4.22: Mach number contours and pressure variation along two different streamlines. Simulation of optimum shape at Mach 2 and $\alpha = 0^\circ$.

4.4 Boattail Junction Shaping Technique

In addition to the optimization procedure, to further enhance the aerodynamic performance of the round bodies, the boattail junction shaping technique is employed. Based on the in-depth understanding of the physical flow over the boattail, it is found that when a boattail afterbody is used in a round body design, a significant reduction in the base drag is normally achieved because the base area and the pressure drop in the flow separation region are smaller.

The presence of the boattail is, however, not without undesirable effects. Firstly, the sharp corner between the cylindrical body and the boattail causes a flow expansion and a drop in pressure that increases the pressure drag. Secondly, as discussed in previous section (section 4.3), since the expansion is three-dimensional, the pressure is gradually increased along the boattail surface, leading to a negative lift when the round is operating at finite angles of attack. This negative lift provides an additional unstable pitching moment about the center of gravity of the round body. In order to account for this increased pitching moment, the round body may be required to spin faster, which consequently results in more significant Magnus effects. Furthermore, even if the spin rate is kept unchanged, the adverse pressure gradient on the boattail may enhance the Magnus effects by itself. Weinacht et al. [18] performed a numerical study using a thin-layer parabolized Navier-Stokes (PNS) solver and they found that the presence of the conical boattail may cause a rise in the Magnus moment coefficient by 15-30%. This increase may significantly influence the dynamic stability of the round.

A simple method to alleviate the negative effects of the boattail is to alter the pressure distribution along its surface. This can be done by simply adding a convex curvature at the junction between the cylindrical body and the boattail. By doing so, the pressure drop is more gradual and therefore, the pressure drag on the boattail is

less pronounced. Furthermore, the region where the flow experiences the adverse pressure gradient is reduced or eliminated and the Magnus effects would be minimized. This boattail juncture shaping technique was first used by Shrewsbury [9] who noticed that having a smoothed corner at the cylinder-boattail juncture of a basic conical boattail is generally beneficial, especially if the radius of curvature is larger than 2 times of the body diameter. This technique was used to significantly reduce the zero-lift boattail pressure drag at transonic regime since this drag component is dominant to the total drag.

However, the drag reduction obtained using this shaping technique cannot guarantee the stability of the rounds during flight. Thus, to design a realistic spin-stabilized round body, the combination of the drag reduction and stability analysis under the boattail juncture shaping technique need to be addressed into the preliminary design stage. In this section, the effect of boattail juncture shaping on the aerodynamic drag and stability of supersonic spin-stabilized rounds is investigated using CFD. Special attention is paid on the Magnus effects and dynamic stability as this is not reported elsewhere in the literature, to the best of our knowledge. The results provide a contribution to understanding the effects of the shaping technique on the drag and stability of the rounds.

4.4.1 Computational Benchmark Models

To study the effect of boattail juncture shaping technique on the aerodynamics performance and stability, the SOCBT round body (see in Figure 3.5b) is used as an example model for this analysis. This model is modified by contouring the juncture of the conical boattail using four radii of curvature: $2.5D$, $4.4D$, $8.8D$ and $13D$, respectively. The configurations and physical characteristics of these rounds are shown

in Figure 4.23 and tabulated in Table 4.3, respectively. In this context, the new modified round bodies are called SOCBT (MOD1), SOCBT (MOD2), SOCBT (MOD3), and SOCBT (MOD4). The aerodynamics characteristics of these models are evaluated using CFD for comparing the aerodynamics performance with the standard SOCBT round body.

Figure 4.23: The configuration of modified SOCBT rounds, a) SOCBT (MOD1), b) SOCBT (MOD2), c) SOCBT (MOD3), and d) SOCBT (MOD4), dimensions in body diameter ($D=57.15\text{mm}$).

Table 4.3: The physical properties of five different generic rounds.

Domain	Generic Round Bodies				
	(Body diameter (D)=57.15 mm)				
	SOCBT	SOCBT (MOD 1)	SOCBT (MOD 2)	SOCBT (MOD 3)	SOCBT (MOD 4)
Center of gravity (C.G.) location	3.6D	~ 3.6D	~ 3.6D	~ 3.6D	~ 3.6D
Moment of inertia I_x (kg m ²)	0.00155	0.00152	0.001502	0.001498	0.001495
Moment of inertia I_y (kg m ²)	0.0254	0.02504	0.0249	0.0238	0.0225
mass (kg)	4.45	4.42	4.4	4.38	4.35

4.4.1 Effect of Boattail Juncture Shaping on Aerodynamic Drag

The effects of boattail juncture contour shape on the aerodynamic drag of the modified versions of the SOCBT are presented in Figure 4.24. It can be clearly seen that adding the concave contour leads to smaller zero-lift drags at all tested Mach numbers. Furthermore, the total drag reduction increases with increasing radius of curvature of the contoured juncture. At the high end of the operating Mach number, however, the drag reduction is significantly smaller. These results are explained by breaking down the drag components of these round bodies as tabulated in Table 4.4. Among the pressure drags from the nose, the boattail and the base, only that of the boattail is strongly affected by the presence of the rounding off corner. Here, the boattail pressure drag can be reduced by up to 25% compared to that of the standard SOCBT round. However, the total pressure drag reduction is smaller, which is up to 5.4% at Mach 2 and 1.5% at Mach 4, because the boattail pressure drag only represents a small portion of the total drag. This contribution is smaller at higher Mach number, thus, the total drag reduction is less pronounced in that regime. The trend of total drag reduction observed in this study agrees well with the previous work of Shrewbury [2] who showed a larger total drag reduction using the juncture shaping technique at transonic speeds (Mach 0.8 to 1).

Figure 4.24 and Table 4.4 also show that a substantial drag reduction can only be obtained when the radius of curvature is sufficiently large. For the SOCBT (MOD1), the total drag reduction is negligible, which suggests that a minimum radius of curvature of $3D$ or $4D$ should be used in order to ensure that the contouring is effective. It should be noted that Shrewbury [2] suggested a smaller value of $2D$ because the Mach number considered in that work was smaller and therefore, the role of the boattail pressure drag was more important.

Figure 4.24: Comparison of zero-lift drag coefficient of the SOCBT rounds at various Mach numbers

Table 4.4: Breakdown of pressure drag coefficient on the SOCBT rounds at Mach 2 and 4

Round shape	Pressure drag coefficient (AOA=0°)				Total Pressure drag
	Nose	Cylinder	Boat tail	Base	
Standard SOCBT (Mach 2)	0.0781	-	0.0454	0.0734	0.1969
SOCBT (MOD 1) (Mach 2)	0.0783	-	0.0429	0.0735	0.1947
SOCBT (MOD 2) (Mach 2)	0.0783	-	0.0398	0.0735	0.1916
SOCBT (MOD 3) (Mach 2)	0.0783	-	0.0372	0.0735	0.1890
SOCBT (MOD 4) (Mach 2)	0.0783	-	0.0340	0.0738	0.1861
Standard SOCBT (Mach 4)	0.0643	-	0.0186	0.0373	0.1202
SOCBT (MOD 1) (Mach 4)	0.0643	-	0.0179	0.0374	0.1196
SOCBT (MOD 2) (Mach 4)	0.0643	-	0.0172	0.0373	0.1188
SOCBT (MOD 3) (Mach 4)	0.0643	-	0.0162	0.0374	0.1179
SOCBT (MOD 4) (Mach 4)	0.0643	-	0.0151	0.0376	0.1170

To explain why the boattail pressure drag decreases when the contoured juncture is employed, the flow features and the pressure distributions along two different rounds are plotted in Figure 4.25. On the standard SOCBT round, the pressure drops abruptly across the centered expansion fan emanating from the sharp corner between the cylindrical body and the boattail. Therefore, a large negative pressure coefficient is generated over the boattail, which consequently leads to a large boattail pressure drag. When the sharp corner is replaced by a smooth contoured surface, a more gradual change of pressure is facilitated through a series of expansion waves. Consequently,

the average static pressure is larger than that of the original boattail and the pressure drag is reduced. This effect is stronger when the radius of curvature is larger.

Figure 4.25: a) Mach number contours of SOCBT, b) Mach number contour of SOCBT MOD3, and c) pressure distributions along SOCBT and SOCBT MOD 3, Mach 2, zero angle of attack

In addition to the zero-lift drag, the effect of juncture contouring on the drag coefficient at finite angles of attack is also investigated. Figure 4.26 shows the comparison of the drag coefficient as a function of angle of attack (1° , 2° , and 4.2°) at Mach 2 for four different rounds: SOCBT, SOCBT (MOD2), and SOCBT (MOD3) and SOCBT (MOD4). SOCBT (MOD1) is not shown here and in the subsequent sections because the drag reduction is small. Similar to the results at zero angle of

attack, drag reduction is achieved with the shaping technique at all angles of attack. The total drag is reduced by up to 1.6%, 3.2% and 4.4% for SOCBT (MOD2), SOCBT (MOD3), and SOCBT (MOD4), respectively. A breakdown of the drag components of these round bodies is also provided in Table 4.5 where it is illustrated again that the drag reduction comes mainly from the boattail pressure drag.

Figure 4.26: The comparison of drag coefficient versus angle of attack for four different SOCBT rounds, Mach 2

Table 4.5: Breakdown of pressure drag coefficient at finite angle-of-attack on the four different rounds at Mach 2.

Round shape	Pressure drag coefficient (AOA=4.2°)				Total Pressure drag
	Nose	Cylinder	Boat tail	Base	
Standard SOCBT (Mach 2)	0.077	-	0.049	0.0965	0.2225
SOCBT (MOD 2) (Mach 2)	0.077	-	0.044	0.0965	0.2175
SOCBT (MOD 3) (Mach 2)	0.077	-	0.043	0.0968	0.2168
SOCBT (MOD 4) (Mach 2)	0.077	-	0.040	0.0972	0.2162

4.4.1 Effect of Boattail Juncture Shaping on Gyroscopic Stability

As expressed in Eq. (2.11), the normal force and pitching moment derivatives are the main aerodynamic factors in the evaluation of the gyroscopic stability. Thus, to study the effect of boattail juncture contouring on gyroscopic stability, Figure 4.27 shows the comparison of the normal force and pitching moment derivatives as

functions of Mach number at the same angle of attack ($AOA=4.2^\circ$) for four different rounds: SOCBT, SOCBT (MOD2), SOCBT (MOD3), and SOCBT (MOD4). It is found that the normal force derivative of the modified SOCBT rounds is smaller than that of the benchmark configuration by up to 4.7%, while the pitching moment is larger by up to 4.3%. Since these changes are small, these modified rounds still remain gyroscopically stable ($S_g > 1$) throughout all design Mach number as shown in Figure 4.28.

Figure 4.27: Comparison of normal force and pitching moment derivatives of four different SOCBT rounds, $AOA=4.2^\circ$.

Figure 4.28: Comparison of the gyroscopic stability factor of four different SOCBT rounds under the same flight conditions, $AOA=4.2^\circ$, $PD/V = 0.283$

The changes in normal force and pitching moment and consequently, gyroscopic stability can be explained by analyzing the pressure distributions along the body in Figure 4.29 and the force breakdown in Table 4.6. For the standard SOCBT round illustrated in Figure 4.29a, a large positive lift is normally generated by the nose and the cylindrical body, while a negative lift is generated over the conical boattail. In Figure 4.29b, for the modified SOCBT (MOD3), it can be seen clearly that the negative lift is reduced using the boattail juncture shaping technique. However, the shorter length of cylindrical portion of modified SOCBT provides significantly smaller positive lift compared with the standard SOCBT. Consequently, the total lift over the modified rounds is smaller as can be seen in the breakdown of normal force derivative of four different rounds in Table 4.6. This explains why the modified rounds provide slightly more pitching moment, and therefore, smaller gyroscopic stability compared to the standard SOCBT round even though the negative lift is reduced.

Figure 4.29: The pressure distributions along two different rounds at Mach 2, $AOA=4.2^\circ$: a) SOCBT and b) SOCBT (MOD 3)

Table 4.6: Breakdown of normal force and pitching moment coefficients on the rounds

Round shape	Normal force coefficient (Mach 2, $AOA=4.2^\circ$)				Pitching moment
	Nose	Cylinder	Boattail	Total lift	
SOCBT	0.1499	0.0589	-0.0155	0.1934	0.2796
SOCBT (MOD 2)	0.1490	0.0550	-0.0135	0.1904	0.2827
SOCBT (MOD 3)	0.1499	0.0521	-0.0125	0.1896	0.2836
SOCBT (MOD 4)	0.1496	0.0462	-0.0097	0.1860	0.2920

4.4.1 Effect of Boattail Juncture Shaping on Dynamic Stability

Figure 4.30 shows the comparison of the dynamic stability factors of the four different round bodies computed using Eq. (2.13) and they are plotted in terms of the stability criterion according to Eq. (2.15). It is found that SOCBT, SOCBT (MOD2), and SOCBT (MOD3) are dynamically stable throughout all design Mach numbers. The stability factor moves closer to the boundary with increasing Mach number, indicating that the round body is less stable at larger flight speeds. Furthermore, it can be seen that the dynamic stability is only sensitive to the presence of the contoured juncture at the low Mach number. At Mach 1.5, the SOCBT (MOD 3) is marginally

stable and the SOCBT (MOD 4) encounters dynamic instability. At higher Mach numbers, the dynamic stability factor of these modified rounds are about the same as the standard round. The results indicate that, for the Mach number range considered in this study, the radius of curvature applied on a SOCBT round should not exceed $8.8D$ for stability reasons.

Figure 4.30: Comparison of the dynamic stability factor on the modified generic rounds, $PD/V = 0.283$

To explain why the larger radius of curvature of juncture contour provides drastic change of stability for SOCBT (MOD 3) and also provides dynamic instability for SOCBT (MOD4) at low supersonic speeds, Figure 4.31 shows the comparison of the Magnus moment derivative and pitch damping moment sum coefficient on three different rounds: SOCBT, SOCBT (MOD2), and SOCBT (MOD3) as a function of various moderate Mach numbers. It is found that a large change in Magnus effect (26%) is generated for SOCBT (MOD3) at a low Mach number (Mach 1.5), while the Magnus effect is only slightly increased (3%) for SOCBT (MOD3) and it is reduced (5%) for SOCBT (MOD2) at Mach 2 and 3. At Mach 4, both modified rounds provide smaller Magnus moment derivatives than that of the standard SOCBT round (up to

5%) as shown in Figure 4.31a. For the pitch damping moment coefficient, it is found that the modified rounds provide slightly lower (within 1% difference) pitch damping moment coefficients than the benchmark round throughout the Mach numbers considered as can be seen in Figure 4.31b. Thus, it can be concluded that the drastic change of dynamic stability at the low Mach number is due to the Magnus moment derivative.

a)

b)

Figure 4.31: The dynamic aerodynamic coefficients of three different generic rounds: a) Magnus moment derivative and b) pitch damping moment sum coefficient under $PD/V = 0.283$, $AOA = 4.2^\circ$

Since the Magnus moment derivative is dependent on both the Magnus force and its center of pressure, the Magnus effect shown in Figure 4.31a can be explained by these aerodynamic characteristics. Figure 4.32 illustrates the Magnus force coefficient and the center of pressure of the three different rounds, SOCBT, SOCBT (MOD2), and SOCBT (MOD3), for various moderate supersonic speeds at the same angle of attack of 4.2° . It is found that, when boattail juncture contouring is employed, the change in Magnus force is a function of Mach number. At high Mach numbers (Mach 2 to 4) the Magnus force is reduced by up to 11.64% and 15.75% for SOCBT (MOD2) and SOCBT (MOD3), respectively, while the Magnus forces of these two modified rounds are increased up to 4% at Mach 1.5. Furthermore, at all Mach numbers, the center of pressure of Magnus force is shifted away from the center of gravity (up to 4.3% at Mach 1.5). This shift is due to a redistribution of Magnus force towards the base of the round bodies as can be seen in the breakdown of the Magnus force over the contoured juncture and boattail portion in Table 4.7.

The change in Magnus center of pressure explains why the Magnus moment derivative does not decrease significantly at high Mach number, despite the large reduction of Magnus force in this regime. This is because the effects of Magnus force and Magnus center of pressure are canceled out. At low Mach number (Mach 1.5), however, the Magnus force is enhanced while its center of pressure moves away from the center of gravity. Consequently, the Magnus moment derivative increases drastically as shown in Figure 4.31a.

Figure 4.32: a) Magnus force coefficient and b) Magnus force center of pressure for the three-different generic rounds (SOCBT and modified SOCBT rounds) under $PD/V=0.283$, $AOA=4.2^\circ$.

Table 4.7: Breakdown of Magnus force of three different SOCBT rounds (Mach 1.5)

Round shape	Magnus force coefficient ($PD/V=0.283$)($AOA=4.2^\circ$)				Total
	Nose	Cylinder	Boattail	Boattail juncture shape	
SOCBT	-0.002	-0.0029	-0.0036	-	-0.00846
SOCBT (MOD 2)	-0.002	-0.0025	-	-0.0038	-0.00825
SOCBT (MOD 3)	-0.002	-0.0019	-	-0.0040	-0.00794

Chapter 5

Wind Tunnel Testing

In the previous chapter (Chapter 4), an optimization procedure was coupled with computational fluid dynamics (CFD) to minimize the aerodynamic drag of the standard 155 mm round body without destabilizing it. It was shown that the optimized body provides up to 15% drag reduction while remains both gyroscopically and dynamically stable over a wide range of Mach numbers ($M = 1.5$ to 4). This is achieved with a reduction in volume or payload of 4%. In order to confirm the high performance of the new optimum 155 mm round, the experimental approach need to be employed.

In this chapter, the aerodynamic characteristics of an optimum 155 mm round (cylindrical length=1.5D) is first investigated experimentally in the large-size, high quality flow wind tunnel SWIFT (the Singapore Trisonic Wind Tunnel). The test of the new high performance geometry is performed at relatively high Mach number (Mach 1.8 to 4) compared with the previous wind tunnel test in literatures [3, 68]. In addition, the aerodynamic performance of the standard 155 mm M549 round is also first investigated at such speeds to compare with the optimized shape [74]. Thus, the experimental study in this chapter contributes to confirm aerodynamic performance of the new optimum 155 mm round body and it also confirms the effectiveness and accuracy of the novel optimization procedure proposed by Jiajan et al. [74].

Therefore, in this chapter, the overview of the experiment and wind tunnel facility are first discussed. The test planning and the experimental set up are then described. Next, the experimental procedures and the testing conditions are presented. Finally, the experimental results of these rounds are used to validate with the numerical results based on both actual testing and the free-flight conditions.

5.1 Overview of Experiment

Since the wind tunnel employed in this study is large in scale (large-size supersonic wind tunnel), the experimental cost is very expensive and it is also time consuming. Thus, this testing was carefully performed by the cooperation between the author and experienced engineering team in the facility. To complete the experiment, the experimental work was divided into two phases: experimental set up and wind tunnel test - data analysis. Prior to testing, the experimental set up which including test conditions, model design and fabrication, test section effect, structural analysis, model assembly and instrumentation are the main considerations of the experiment (to be discussed later). After the experimental aspects are completed, the wind tunnel test and data analysis are then performed.

5.2 Wind Tunnel Facility and Instrumentation

The experiment was carried out in the Singapore Wind Tunnel Facility (SWIFT) [1] at DSO National Laboratories, Singapore (See Figure 5.1). The SWIFT is an extensively refurbished trisonic blowdown wind tunnel capable of testing articles ranging from Mach 0.3 to 4 in a 1.2 m $\times$ 1.2 m square test section. The design Mach number and Reynolds number can be controlled based on the available total pressure which ranges from 1.7 to 17.5 bars. The design total pressure is controlled by the

pressure control component (control valve, see Figure 1). In supersonic flows, Mach number is set by the adaptable nozzle. The typical run time of the tunnel is about 20 to 50 seconds depending on the Mach number and stagnation pressure. The uncertainty on the Mach number is based upon the uncertainties of both stagnation and upstream static pressures. Based on the measured flow quality, the variation of the axial Mach number is less than 1%, the temporal static pressure is less than 0.0025% for low Mach numbers and the upstream turbulent intensity is less than 0.5%. For flow angularity, at supersonic speeds, the average pitch angle is approximately 0.05° and yaw angle of 0.05° . The total temperature in the reservoir tank is approximately the same as the standard sea level condition in Singapore (300 K).

The stagnation pressure p_t in the test section is measured by a Druck absolute pressure transducer in the settling chamber of the wind tunnel. The range of the transducer used is 7 bars. The accuracy of this transducer is typically 0.05% of full scale. The stagnation temperature T_t is measured by a thermo couple probe in the settling chamber. The base pressure p_B is measured by a Druck piezoresistive differential pressure transducer. The range of this transducer is 0.35 bars, with 0.05% of full scale. The pitching angle of the model support is measured by a resolver mounted in the mechanism. The accuracy of the pitching angle reading is 0.05° .

In the test section, the model is supported by the sting mounted on a pitch-and-roll mechanism by which the desired angles of attack can be achieved. Two different movements of sting: holding mode and sweeping mode (continuous movement) are available in the facility during a test run. For each run, the flow pattern over the body is also visualized by a schlieren system.

Figure 5.1: The Singapore Wind Tunnel Facility (SWIFT)

5.3 Measurement Parameters

As mentioned earlier, the aerodynamic characteristics of the two different flight bodies: standard 155 mm M549 round and optimum 155 mm geometry (cylindrical length= $1.5D$) are investigated experimentally to confirm the aerodynamic performance. These round bodies are shown in Figure 3.6c and Figure 4.9 in full-scale. In this study, the test plan includes force and moment measurements for non-spinning conditions only as spinning combine with precession motion tests are not available in SWIFT. Therefore, only static aerodynamic coefficients are obtained in the experiment. Nevertheless, as mentioned in Chapter 3, these aerodynamic coefficients are slightly affected by the spinning and coning motion. Moreover, the static aerodynamic coefficients alone are sufficient for assessing the aerodynamic performance in terms of drag and gyroscopic stability on the bodies.

5.4 Experimental Setup and Considerations

Prior to wind tunnel testing, there are five experimental aspects that need to be considered: test conditions, balance load limits, model scaling, test section effects, and model structural analysis.

5.4.1 Test Conditions

For the test condition consideration, the experiments need to be set up according to the specifications of the SWIFT wind tunnel. The maximum Mach number in this study is intended to be 4.0 which is the maximum Mach number attainable in SWIFT. The maximum attainable total pressure in SWIFT tunnel can be up to 17.5 bars as can be seen in the tunnel operating envelope for the total pressure (Figure 5.2). However, a maximum total pressure of 15 bars is recommended as a realistic value. The desired total pressure in the reservoir tank depends on the test-section Mach number. The total pressure also affects the Reynolds number and hence the skin friction drag. Because Reynolds number is an important parameter that needs to be duplicated in the experiments, considerations must be given to determine whether the available total pressure would allow successful matching of Reynolds numbers for the range of Mach numbers studied here.

Figure 5.2: The available total pressure for the SWIFT wind tunnel

For the force balance consideration, the wind tunnel models have to be designed according to the specifications of the balance. The load limits of the 6-DOF balance installed in SWIFT are as follows:

In order to check whether the aerodynamic forces exerted on the balance are within the allowable limits for both the standard 155 mm M549 generic round and the optimum shape models, the attainable total pressure will first be determined as an attempt to duplicate the free-flight Reynolds numbers. The isentropic relation is used to calculate the properties in the test section. The corresponding forces and moments are obtained using the CFD simulations. The aerodynamic forces, moment, and test section flow conditions for full-scale flight at sea level are tabulated in Table 1 (Appendix C). From Table 1, it can be seen that the forces and moments increase with the flight Mach number, as the free stream pressure is always at atmospheric value. For the full-scale standard 155 mm M549 round, the axial forces would exceed the load limit of the balance, while the normal force and moments are well within tolerable limits. To reduce the axial forces to within the balance limit, the model must be scaled down.

5.4.3 Model Scaling (Model Design)

The scaling of the current test models was determined based on the balance load limits and the balance configurations (diameter). According to an existing model (AGARD model B) that has previously been tested in the SWIFT tunnel, the maximum base diameter of this model is about 76.2 mm or 3-inch. The standard 155 mm generic round model is therefore scaled based on the constrained base diameter of 76.2 mm. Knowing that the conical boat tail has a boat tail angle of 7.5° , the maximum diameter (cylindrical body) can be easily calculated. The resulting 1/1.668 scaled model has a maximum diameter and a total length of 92.88 mm and 524.3 mm, respectively. The maximum diameter and the total length of the standard round are also used for the optimum geometry. The drawings of the two different rounds (1/668 scale) are shown in Figure 5.3.

Prior to fabricating the model, the detail design of the wind tunnel models is required to specify the dimensions for model assembly. There are three main requirements of this process: specification of 90% contact surface at balance, base clearance, and structural analysis. To satisfy the requirement of the tolerance with 90% contact between the adaptor and balance, the CNC (computational numerical control) machine needs to be employed for this process. This is needed so that the balance can provide the measured data correctly. In addition, to avoid contact between the model base and sting that might occur due to the vibration, the clearance between the base of the wind tunnel models and sting should be designed in length at least 5 mm. This is also very important for wind tunnel model design issue to ensure that the balance can measure the forces and moments correctly. However, the clearance should not be too large to avoid the strong influence of base flow unsteadiness which could lead to the difficulty of base pressure measurement. The detail design of two different wind

tunnel models with the base clearance and 90% contact at balance are shown in Appendix C. After the wind tunnel models detail designs are finalized, the structural analysis is then employed. The structural analysis will be discussed later.

Figure 5.3: The drawing of two different wind tunnel models (1.668 scale): 155 mm M549 round (above) and optimum 155 mm geometry (bottom) (diameters = 92.88 mm)

Note that when the model size is reduced, the flow Reynolds number over the model must still be maintained to match the full-scale flight value by adjusting the pressure, otherwise the mismatch in Reynolds number will lead to incorrect skin friction results, and hence the overall drag. Table 2 (Appendix C) lists the test conditions and forces and moments for the 1/1.668 scaled reference model if the Reynolds numbers match exactly to those for the full-scale conditions in Table 1 (Appendix C). There are two important points here: 1) the axial forces at Mach 3 and 4 still exceed the allowable limits of the balance, and 2) the total pressures called for at the required Reynolds numbers would exceed the facility total pressure values at all Mach numbers.

The test conditions are then recalculated based on the total pressure limits in the SWIFT facility, which are all lower than those required for Reynolds number matching. The forces, moment, and the corresponding test conditions are tabulated in Table 3 (Appendix C). From the results in Table 3, it is seen that the Reynolds numbers for the

scaled wind tunnel model are all below, but are still within the same order of magnitude from, those for full-scale sea level flight. Thus, it is expected that the skin friction results in the wind tunnel tests would not differ significantly from those for full-scale flight. Furthermore, because skin friction usually plays a small role in the total drag for the current Mach number range of interest, the forces and moments in the experiments should not be severely affected by the mismatch in Reynolds numbers. In any case, the aerodynamic forces are now within the allowable limits of the balance for the achievable facility total pressures. It should be noted that the similarities of the flow separation behaviour over the round bodies is not considered for the test because the experiment is conducted at a small angles of attack and the round bodies have a small boattail angle. Since the coning angle is not large, the state of the boundary layers is not likely to deviate significantly from real flight. The angles of attack tested in the experiments were also selected to reflect real flight scenarios. Thus, it is assumed that the flow over conical boattail of the wind tunnel model still remains attached at all the supersonic speeds.

5.4.4 Test Section Considerations

For the test section wall interference considerations, because the test section size is quite large ($1.2\text{ m} \times 1.2\text{ m}$), the reflection of the leading edge shock wave would not be impinging on the test article even for the full-scale model at finite angles of attack under supersonic speeds. It is therefore not necessary to consider this problem further for the 1/1.668 scale test model.

5.4.5 Structural Analysis

After the test conditions have been established and the wind tunnel model design is finalized, the remaining consideration is the structural integrity of the model

assembly. A preliminary finite element analysis (using Ansys Workbench [72]) has been carried out to calculate the stresses in the wind tunnel model. Steel is selected to be the fabrication material of the wind tunnel model which is used as one of the input parameters in the finite element analysis. The recommendation for the safety factor is that it should be greater than 5 to ensure that the model is sufficiently strong throughout the test Mach numbers. Here, the safety factor is defined using Von-Mise criterion as the ratio of the yielding stress to von-Mise stress. The numerical procedures for structural analysis of two different models are available in Appendix C. From the structural analysis results using FEM as shown in Appendix C (Figure 6 to 8), it is evident that both wind tunnel models yield safety factors greater than 5. Thus, the wind tunnel models of these rounds can be used with confidence for conducting in wind tunnel testing throughout all intended Mach numbers.

5.5 Model and Balance Installation

5.5.1 Model Assembly

As mentioned in section 5.3.2, due to the limited forces and moments that the balance can experience, the wind tunnel models have been scaled down to 1/1.668 (see Figure 5.3). As shown in Figure 5.3, it can be seen that the wind tunnel models are constructed in three pieces and the cavity inside the body is created to facilitate the installation of the force balance. These components consist of a nose, a mid-section (adaptor), and a cylindrical-boattail afterbody (round body) as shown in Figure 5.4. From Figure 5.3 and 5.4, since the nose of an optimum round has a smaller thickness, the adaptor location of this round is designed slightly further down stream compared

with the standard round to avoid the structural problem. This leads to provide the difference of balance location inside the round cavity as shown in Appendix C.

Figure 5.4: The CAD model of two different rounds (1.668 scale) of the standard 155 mm M549 round (above) and optimum 155 mm geometry (bottom)

Here, the procedures of wind tunnel model and sting balance installation is discussed. The installation begins with the connection between the sting and the balance. Next, the body tube is moved to cover the balance. The round body part is then screwed to the adaptor and the nose, respectively. It should be noted that the adaptor is used to connect the nose and balance together inside the round body. The assembly of each component and the balance with the sting is shown in Figure 5.5.

Figure 5.5: The assembly of three different components of model and sting balance

Figure 5.6 shows the photographs of the completed wind tunnel model (optimum round) installation inside the wind tunnel test section. In addition, the connection of

balance and the sting is also demonstrated. In this figure, at the rear part of the wind tunnel model, it can be seen that the base cavity is designed to locate pressure tubes inside the body.

Figure 5.6: The model and sting balance installation inside the wind tunnel test section

5.5.2 Pressure Transducers Installation

In addition to the model installation, as the attachment of the sting balance changes the base geometry and the flow in the base region, the measured force is not that of an independent flying body. In this study, the most difficult part of the wind tunnel test program is to obtain the base pressure accurately. Thus, to obtain the aerodynamic drag, the drag obtained from the wind tunnel test needs to be corrected by extracting the base drag using base pressure measurements. In order to measure base pressure, pressure transducer tubes are installed at desired locations inside the base cavity. The location of the pressure probes should be far enough from the base to ensure that the measured pressure is not influenced by the base flow unsteadiness. The suitable range of the base pressure transducers is also required for testing at relatively high Mach number. In the current experiment, two different configurations of the installation of the base pressure transducers, namely, configuration A and configuration B are considered. They are illustrated in Figure 5.7.

Figure 5.7: The two different configurations of pressure transducers installation over two different wind tunnel models (1.668 scale): Configuration A (6-probe) (above) and Configuration B (4-probe) (below).

For configuration A, six pressure probes are used in total where four probes are instrumented close to the rear end of the round body and two others are much further inside the cavity. Configuration B is similar to configuration A, except that the two pressure probes further inside are not used. The four probes close to the base are located along the circumferential direction on the balance. Their distances from the base are 4.8 cm and 10 cm for the optimum round and standard round, respectively. Several runs were performed for both configurations at the same test condition and it

was found that the measured base pressure, which is the average of the reading of every probe, is almost identical for both configurations. This result indicates that the pressure probes are sufficiently far from the base and the measurement is insensitive to the base flow. For brevity, only the results from configuration B are presented in the subsequent sections.

5.5.3 Moment Center of Balance

The center of gravity (C.G.) of the round is selected as the reference point for the pitching moment calculations. The C.G. locations of two different rounds are shown in Figure 5.8. However, according to the actual wind tunnel test, the measured pitching moment is calculated with respect to the balance moment center (B.M.C). Here, the specifications of the balance which including the total length, diameter, and balance moment center are shown in Figure 5.8. Thus, to determine the aerodynamic pitching moment with respected to the center of gravity, the pitching moments obtained from the balance need to be transferred to the center of gravity of model by using the difference of B.M.C and C.G. locations.

Figure 5.8: The balance moment center and center of gravity location of two different rounds: the Standard 155 mm M549 model (above) and an Optimum 155mm model (below) ($D=92.88\text{mm}$).

5.6 Test Conditions and Procedures

In this experiment, the tests of two different models were performed at Mach numbers ranging from 1.8 to 4. The Reynolds number is approximately up to 6.5×10^6 based on the model body diameters. In the current study, the data are also collected for Reynolds numbers ranging from 2.26×10^6 to 4.6×10^6 at Mach 2, to investigate the Reynolds number dependency on aerodynamic behaviors under both zero and finite angles of attack. The sweeping model movement is used as the primary mode to measure the force and moment changes with respect to the angle of attack in the interval of -5° to 5° . In addition, the holding position mode is also used to evaluate drag at selected angles of attack. The summary of wind tunnel test conditions of the two different models is tabulated in 5.1. The details of the test conditions for each run are provided in Appendix D.

Table 5.1 Test conditions for two different rounds

Parameter	Value
Mach number	1.8 to 4
Total pressure (Pa)	2×10^5 to 15.5×10^5
Static pressure (Pa)	0.1×10^5 to 0.3×10^5
Reynolds number	2.2×10^6 to 6.48×10^6
Total temperature (reservoir tank, K)	~ 300
Angle of attack (pitch, degree)	-5° to 5°

5.7 Experimental Uncertainties

It is known that the accuracy of aerodynamic results obtained from wind tunnel testing is related to the uncertainty of the wind tunnel flow properties and wind tunnel system. In the current study, the experimental results consist of drag coefficient (axial force coefficient), normal force coefficient, and pitching moment coefficient. There are two different sources that would affect the uncertainties of these aerodynamic

results: systematic error (or bias error) and precision error [73]. The summations of these sources are called the total uncertainty as expressed in Eq. (5.1) below.

$$\text{Total uncertainty: } [u_y]_{\text{Total}} = \left[\left(1.96 \times [u_y]_{\text{precision}} \right)^2 + \left([u_y]_{\text{bias}} \right)^2 \right]^{1/2} \quad (5.1)$$

where $[u_y]_{\text{precision}}$ and $[u_y]_{\text{bias}}$ can be represented by a Taylor series of error propagation

$$[u_y]_{\text{precision/bias}} = \left[\left(\frac{\partial y}{\partial x_1} \sigma_{x_1} \right)^2 + \left(\frac{\partial y}{\partial x_2} \sigma_{x_2} \right)^2 + \left(\frac{\partial y}{\partial x_3} \sigma_{x_3} \right)^2 + \dots + \left(\frac{\partial y}{\partial x_n} \sigma_{x_n} \right)^2 \right]^{1/2} \quad (5.2)$$

and $y = y(x_1, x_2, x_3, \dots, x_n)$ is an aerodynamic coefficient that is a function of air properties and model configurations ($x_1, x_2, x_3, \dots, x_n$).

To estimate the precision error of the measured aerodynamic coefficients, the uncertainty of the basic tunnel parameters - static pressure and total pressure - are first evaluated based on the repeatability of the test. It should be noted that the uncertainty of the tunnel parameters is referred to as the standard deviation (σ) of these data. Here, the precision of the total pressure and static pressure are expressed below:

- Total pressure, p_t : ± 0.022 %
- Static pressure, p : ± 0.023 %
- Base pressure, p_B (for drag minus base drag): ± 0.115 %
- Balance force and moment variations (approximately): ± 0.03 %

Once the uncertainties (above) are obtained, the precision component can be determined using Eq. (5.2). In addition to the precision uncertainty, to evaluate the systematic uncertainty (or bias) of the aerodynamics, the instrumental uncertainties and model configurations (diameter) uncertainty are also given as follows:

- Total pressure, p_t : ± 0.05 % of full scale
- Static pressure, p (pressure transducer): ± 0.004 % of full scale

- Base pressure transducer, p_B (for drag minus base drag): $\pm 0.05\%$
- Balance (axial force, normal force, and pitching moment): $\pm 0.092\%$, 0.067% , and 0.091%
- Pitch-angle mechanism: $\pm 0.1\%$
- Diameter (measurement error): $\pm 0.5\%$

Here, an example calculation of the total uncertainty of axial force coefficient (or drag), C_{AF} is presented. The axial force coefficient formulation can be expressed as:

$$C_{AF} = \left[\frac{8AF}{\rho M^2 \pi D^2} \right] = \left[\frac{8AF}{\frac{2\gamma}{\gamma-1} ([p]^{1/\gamma} [p_t]^{\gamma-1/\gamma} - P) \pi D^2} \right] = C_{AF}(AF, p_t, p, D) \quad (5.3)$$

Using Taylor series method, the bias and precision uncertainties can be expanded as

$$\left[u_{C_{AF}} \right]_{\text{precision bias}} = \left[\left(\frac{\partial C_{AF}}{\partial AF} \sigma_{AF} \right)^2 + \left(\frac{\partial C_{AF}}{\partial p} \sigma_p \right)^2 + \left(\frac{\partial C_{AF}}{\partial p_t} \sigma_{p_t} \right)^2 + \left(\frac{\partial C_{AF}}{\partial D} \sigma_D \right)^2 \right]^{1/2} \quad (5.4)$$

Thus, we have

$$\left[u_{C_{AF}} \right]_{\text{precision}} = \left[0.03^2 + 0.023^2 + (0.022)^2 + 0 \right]^{1/2} = \pm 0.044\% \quad (5.5)$$

$$\left[u_{C_{AF}} \right]_{\text{bias}} = \left[0.092^2 + 0.12^2 + 0.25^2 + (2 \times 0.5)^2 \right]^{1/2} = \pm 1.04\% \quad (5.6)$$

From Eq. (5.5) and (5.6), the total uncertainty of the axial force coefficient can be obtained

$$\left[u_{C_{AF}} \right]_{\text{Total}} = \left[(1.96 \times 0.044)^2 + (0.308)^2 \right]^{1/2} = \pm 1.0435\%$$

To estimate the uncertainty of normal force and pitching moment coefficient, a similar procedure is used. Since these aerodynamic coefficients are functions of angle of attack, the precision of pitch-angle needs to be included in the uncertainty analysis. Thus, the uncertainties of these aerodynamic coefficients are larger than the precision

of aerodynamic drag. Based on Eq. (5.1) to (5.3) above, the precision of these aerodynamic coefficients are expressed as follows:

- Axial force coefficient (excluding the base drag): $\pm 1.45\%$
- Normal force coefficient (approximately): $\pm 1.56\%$
- Pitching moment coefficient (approximately): $\pm 1.72\%$

5.8 Results and Discussions

In this section, the experimental results of two different wind tunnel models are presented and compared with CFD under actual test conditions. Based on these results, the aerodynamic performance can be investigated. The details of the numerical procedures have already been discussed in Chapter 3. Here, the grids of the wind tunnel models (1/1.668 scale) are shown in Figure 5.9. In addition, the actual testing conditions that are used for the CFD boundary conditions are also tabulated in Appendix D.

Figure 5.9: The computational mesh of wind tunnel models (1/1.668 scale): a) axisymmetric mesh of the optimum round and b) 3-D grid of standard 155 mm round.

5.8.1 Static Aerodynamic Coefficients

5.8.1.1 Aerodynamic Drag

As discussed in Chapter 2, for a typical round body, the base drag is the main contributor to the total drag. However, obtaining accurate base pressure or base drag in

wind tunnel testing is very difficult since the sting affects the base pressure and, consequently, the total drag. In this section, the drag minus base drag coefficient is presented. This drag coefficient is determined by extracting the base drag from the total drag of the model with the sting. The base drag is computed from the average value of the pressure probe's reading (see Section 5.4.2). The drag coefficients of two different models are shown in Figure 5.10.

a)

b)

Figure 5.10: Drag minus base drag coefficient of two different rounds: a) optimum geometry and b) standard 155 M549 round

Figure 5.10 shows the zero-lift drag minus base drag coefficient of the optimum geometry and standard 155 mm M549 round as a function of Mach number. It is found that the drag minus base drag coefficients of the two different wind tunnel models agree well with the CFD. They are within 6.5% and 5.5% for optimum geometry and standard round, respectively.

In addition to the drag minus base drag coefficient, since it is interesting to compare the total drag obtained from the wind tunnel test with the aeroballistics range data [63], the base drag computed from CFD will be added to the experimental drag minus base drag results. Consequently, the total drag can be obtained. Figure 5.11 shows the comparison of the total zero-lift drag coefficient obtained from aeroballistics range data, CFD (free-flight conditions), CFD (wind tunnel conditions), and the wind tunnel experiments. It is found that the experimental total drag results (drag minus base drag + CFD base drag) of the optimum geometry agree well with CFD for both free-flight and actual test conditions as shown in Figure 5.11a. They are within 3% throughout all Mach numbers. For the total drag of the standard 155mm round as shown in Figure 5.11b, it is found that the experimental zero-lift drag results are also closed to both the aeroballistics range data and CFD (within 6%).

Figure 5.11: Total zero-lift drag coefficient of two different rounds: a) optimum geometry and b) standard 155 mm M549 round.

5.8.1.2 Normal force and Pitching Moment Derivative

As mentioned earlier, to study the effects of angle of attack on the normal force and pitching moments, these aerodynamic characteristics are measured by sweeping the model through a range of angles of attack (-5° to 5°). The experimental results of the normal force and pitching moment as a function of angle of attack (-5° to 5°) at different Mach numbers are plotted in Appendix D. From these figures, it is shown that these aerodynamic coefficients are linear functions of angle of attack. As expected,

the trends of these experimental results are supported by the linear theory of generic rounds under small angles of attack [25].

To verify the aerodynamic performance of the two different rounds, the derivatives of these aerodynamic coefficients are evaluated with respect to the angle of attack. Figure 5.12 shows the comparison of normal force and pitching moment derivatives of the standard 155mm round using three different approaches: CFD, wind tunnel testing, and aeroballistics range as a function of Mach number at 2° . By comparing the normal force and pitching moment derivatives based on wind tunnel test data with published aeroballistics range data [64] as shown in Figure 5.12a and 5.12b, respectively, good agreement can be observed between these approaches (within 3% difference), while the numerical results based on both the free-flight (full-scale) and wind tunnel models (1/1.688 scale) also agree well with the current experimental data. They are within 3% and 6% difference for normal force and pitching moment, respectively, throughout all the Mach numbers considered as can be seen in Figure 5.12.

Figure 5.12: Aerodynamic characteristics of the standard 155 mm M549 round: a) normal force derivative coefficient and b) pitching moment derivative coefficient.

For the investigation of the normal force and the pitching moment derivative of the optimum 155 mm geometry as shown in Figure 5.13, it is found that the CFD prediction based on both scales (full-scale and 1/1.668 scale) yield good agreement with the wind tunnel test results (within 5%).

Figure 5.13: Aerodynamic characteristics of the optimum 155 mm round: a) normal force derivative coefficient and b) pitching moment derivative coefficient

5.8.1.3 Experimental Results Discussions

Regarding the experimental results as discussed in section 5.8.1.1, there are some disagreements between CFD and experiment on the aerodynamic drag (drag minus base drag coefficient) for the first round testing. This is because the operating range of the absolute pressure transducer which was used for base pressure measurement is

relatively low. This range is not proper for the high frequency unsteadiness in the base flow at high supersonic speeds. Thus, to overcome this issue, the second round of the test (or repeatability test) with the proper range of the base pressure measurement was required. In addition, several base pressure transducers are also needed for installation at different locations inside wind tunnel models as discussed in section 5.5.2. Consequently, the averages of these pressures are obtained. The results of the repeatability test are already discussed in the section 5.8.1.1 which clearly showed that a good agreement with CFD can be observed. The differences are within 6% between the current test and CFD. These differences would be caused by an inaccurate numerical base drag prediction using steady RANS.

According to the CFD study as discussed in chapter 3, it is found that the numerical aerodynamic total drag prediction of different round geometries are about 5-10% error compared with existing wind tunnel data and aeroballistic range. This error is mainly from the base drag prediction because this drag component is the main contributor to the total drag (more than 45%) (see Table 4.1). For the pressure drag prediction (or wave drag), normally the CFD can predict well on this drag component over the different portions of the round body: nose, cylindrical body, and boattail. This can be supported by the predicted pressure distributions along the body as shown in Figure 3.9 and Figure 3.10. Moreover, for the skin friction drag, since this drag component is only 10-13% to the total drag, it is impossible to cause such a level of error of aerodynamic drag. Therefore, since there are no ways to obtain the base drag correctly from the experiment (both wind tunnel testing and aeroballistic range), it is difficult to confirm the accuracy of the base drag prediction using numerical methodology.

However, in the current section, to ensure the accuracy of this aerodynamic coefficient, the RANS has been applied to simulate the pressure distributions at the base of supersonic round body. The axisymmetric body with a base diameter of 63.5 mm of Henry and Dutton, 1994 is chosen as the benchmark geometry (see Figure 5.14). This body was tested in a supersonic wind tunnel using laser dropper velocity methodology to capture the flow details behind the base. In addition, to measure the mean static pressure along the base, 19 pressure taps were used. The test was conducted at Mach 2.46 and zero angle of attack. It should be noted that the sting was connected to the model at upstream to prevent any interference with the flow field at the base. The details of flight test condition and model installation can be seen in the experimental study of Henry and Dutton, 1994 [77].

Figure 5.14: The simple axisymmetric body ($D= 63.5$ mm)

The numerical results in terms of pressure distributions along the base, flow visualization (Mach number contour and streamline), and the velocity profile at the center line behind the base are shown in Figure 5.15, Figure 5.16, and Figure 5.17, respectively. From figure 5.15, it can be seen that RANS methodology failed to predict the trend of pressure distributions correctly at the base compared with the experimental data [77]. The experiment shows that a constant pressure distribution is generated across the base line, while the CFD provides a smooth curve at the base. Since the flow at the round base is an unsteady flow phenomenon, the steady RANS

simulation might not be appropriate to capture this flow detail. According to the CFD works in the literatures [78, 79], DES and Hybrid RANS/LES are normally suggested as the main approaches to provide the correct trend of the supersonic base flow phenomenon (to be discussed later in future work).

Nevertheless, regarding the averaged base pressure coefficient obtained from the steady RANS as shown in Figure 5.15, it is found that the simulation result is reasonable. It falls within the engineering accuracy compared with the experiment (within 8.5%). Moreover, the rear stagnation point of the recirculation flow also agrees well with the test (only 2% error) as shown in Figures 5.16 and 5.17. Thus, it is clearly shown that the error of the drag coefficient which obtained in the previous chapter is normally from the limitation of base drag prediction using RANS. However, this study helps to confirm that RANS can be used with confidence to obtain mean aerodynamic base drag results.

Figure 5.15: Comparison of the base pressure coefficient along the base

Figure 5.16: Mach number contours and streamlines behind the base

Figure 5.17: Comparison of the velocity profile at the center line behind the base

For the other aerodynamic coefficients: normal force derivative and pitching moment derivative, the experimental data obtained from the first round of the test is acceptable. Since these aerodynamic coefficients are not influenced by the base flow unsteadiness, there are only 3% and 5% difference compared with the previous CFD

and aeroballistic range data for normal force derivative and pitching moment derivative, respectively. The differences of these coefficients are even smaller once they are compared with the error bounds obtained from uncertainty analysis. However, these differences might be caused by the measured error of the balance and also the error of the range between balance moment center and the round's center of gravity.

After the experimental results are confirmed, the aerodynamic performance and stability of the rounds can be analyzed. As discussed previously, only the static aerodynamic coefficients can be investigated for the current test since the spinning motion combine with coning motion testing is not available. However, the static coefficients alone are sufficient for assessing the aerodynamic performance in terms of drag and gyroscopic stability (Eq. 2.11) on the bodies. Moreover, the aerodynamic drag coefficient and normal force derivative obtained from the current test are also important parameters that can be used to determine the dynamic stability (see Eq. 2.13).

From Equation 2.11, it is clearly shown that the gyroscopic stability, S_g , is strongly affected by pitching moment derivative. If this aerodynamic behaviour is decreased, the gyroscopic stability is definitely more stable. Nevertheless, the gyroscopic stability alone does not ensure the dynamic stability of the round during flight. Thus, to complete the stability analysis of spin-stabilized rounds, the other aerodynamic parameters: Magnus moment derivative and pitch damping moment sum coefficient would be needed in future experiments (dynamic testing) to address the dynamic stability (to be discussed later in chapter 6).

5.8.1.4 Summary

The results in the preceding sections have verified the accuracy of the optimization procedure proposed in Jiajan et al. [74]. These experimental results were then discussed further to explore where the difference between numerical results and the test data come from. In addition, the experimental results and its discussions in the previous section also help to confirm that the optimum round performs significantly better than the standard 155 mm round. For clarification, the measured aerodynamic characteristics of both rounds are compared directly from Figure 5.10 to 5.13. It can be seen in Figure 5.18 that, at every Mach number, the optimum round produces a drag minus base drag that are significantly smaller than those of the standard round. The drag reduction is up to 22% for the drag minus base drag. This reduction means that the range of the round can be greatly increased. Moreover, Figure 5.19 shows that the pitching moment derivative is also around 30% smaller, which is desirable for stability.

Figure 5.18: Comparison of aerodynamic drag minus base drag coefficient of two different wind tunnel models based on wind tunnel testing

Figure 5.19: The comparison of unstable pitching moment of two different wind tunnel models based on wind tunnel testing.

5.8.2 Effects of Reynolds Number on Aerodynamic Characteristics

Figure 5.20 shows the effects of Reynolds number on the normal force and pitching moment coefficient on the optimum geometry at Mach 2. It is found that the effect of Reynolds number on these aerodynamic coefficients is very small. However, for the variation of zero-lift drag correction with Reynolds number as shown in Figure 5.21, as expected, the drag minus base drag coefficient decreases with increasing Reynolds number. It should be noted that the reduction of this aerodynamic characteristic is mainly due to the decreasing of the skin friction drag component since it is a function of Reynolds number [27]. To demonstrate the effect of Reynolds number on the skin friction drag coefficient, Figure 5.22 shows the comparison of drag components results obtained from CFD as a function of Reynolds number. It is clearly shown that the pressure drag coefficient is not sensitive to the Reynolds number while the skin friction drag coefficient is significantly decreased (up to 7%) with increasing Reynolds number. In addition, it is also found that the CFD drag minus base drag

agree well with the experiments (within 4%) throughout all Reynolds numbers. Thus, this study and the results in the previous sections (section 5.7.1) suggest that CFD is an efficient and accurate tool to bridge the Reynolds number gap between the wind tunnel test and the aeroballistics range test.

Figure 5.20: Effects of Reynolds number on aerodynamic coefficients of optimum 155 mm geometry at Mach 2; a) pitching moment coefficient and b) normal force coefficient.

Figure 5.21: Effects of Reynolds number on zero-lift drag minus base drag of optimum 155 mm geometry at Mach 2.

Figure 5.22: Effect of Reynolds number on drag components of optimum 155 mm geometry at Mach 2.

5.8.3 Flow Visualization

The color Schlieren system with 500W Xenon light source is used for flow visualization in the test section. The internal light path is adjusted by a set of diameter 40mm and 60mm mirrors. Two concave Mirrors with diameter 500mm are installed. One is used to converge light source towards camera and the other is to collect light source emitted from the Xenon bulb. In addition, two plane Mirrors with diameter 750mm are also setup in this system (positioned on opposite side of the test section). Schlieren alignment between “500mm mirrors - 750mm mirrors - test-section” was achieved with a 0.5mW He-Ne laser source.

According to the facility concerns (structural aspect), using a longer sting is avoided for the test because a large moment arm at the sting might be generated due to the high dynamic pressure at supersonic speeds, especially at Mach 4. Consequently , this moment would cause a serious damage to the wind tunnel system. This leads to a

sting length that is shorter than the position of the observation window. Thus, only the flow field around the nose can be seen in the schlieren picture.

The comparison of schlieren photos of the flow field which were obtained during the wind tunnel tests and the Mach number contours based on numerical simulations at Mach 2 and 4 of two different models are illustrated in Figure 5.23. From this figure, the flow is characterized by a strong shock wave at the nose, several expansion fans along the body and a large recirculation region at the base of the body. It is evident that the simulation results and the schlieren-recorded flow fields are very similar at the two different Mach numbers. From Figure 5.23, based on schlieren photos, the detached shock is generated at the flat nose of the standard 155 mm round and is then followed by the oblique shock over the nose, while the attached shock over the nose of the optimum round is clearly seen in Figure 5.23b. The shock angles over the nose are approximately 20° and 33° , for standard 155 mm M549 round and optimum geometry, respectively.

a)

b)

Figure 5.23: Comparison of schlieren images with Mach number contours of two different rounds: a) standard 155 mm M549 round, at Mach 4 and b) optimum 155 mm round at Mach 2.

Chapter 6

Conclusions and Future Work

6.1 Summary

6.1.1 Aerodynamic Study and Prediction

The goal of aerodynamic prediction is to investigate the ability of two different approaches: semi-empirical method (McCoy's solution) and computational fluid dynamics (CFD) using RANS (SST $k-\omega$) to predict the aerodynamic characteristics of spin-stabilized rounds under both non-spinning and spinning conditions. In addition, another goal of this study is to provide better understanding about the physical flow over generic round bodies. Five different available generic round shaped bodies, namely: Secant-Ogive Cylinder (SOC), Secant-Ogive Cylinder Boat Tail (SOCBT), Army-Navy Spinner Rocket (ANSR), M910, and 155 mm M549 round were selected as the computational benchmark models for aerodynamic study and validation. Understanding the flow-field characteristics of these rounds can provide hindsight for aerodynamic performance improvement. According to the ability of aerodynamic prediction of two different approaches, it was demonstrated that these approaches provided a good agreement with existing wind tunnel test data and aeroballistics range data throughout a range of moderate supersonic speeds. Both static and dynamic aerodynamic coefficients obtained from these methods fall within the engineering

accuracy range (within 10%). Therefore, the semi-empirical method and computational approach can be used with confidence to design the optimum geometry, which provides a good performance.

6.1.2 Aerodynamic Performance Improvement

6.1.2.1 Optimization Procedure

To improve the aerodynamic performance of the rounds, an optimization procedure was developed to design an optimum spin-stabilized flight body which yielded minimum drag and high stability at supersonic speeds of Mach 1.5 to 4. The procedure combines a semi-empirical drag evaluation, the SQP optimization technique and the stability analysis based on the gyroscopic stability and the dynamic stability condition. It was shown that the current method, which is simple and quick, provides more accurate drag prediction and a similar optimum shape in comparison with a previous optimization technique. Furthermore, including the stability analysis and improvement into the optimization cycle was proven to be crucial to ensure that the designed shape is realistic.

The optimization procedure was demonstrated on the design of a round body. The total drag of the optimum geometry is 24% less than that of the benchmark 155 mm M549 round at the same speed. However, the optimum configuration was impractical for spin-stabilized flight because the lack of a cylindrical body section would produce large wobbles in the launch tube leading to instability. As a result, the optimum geometry was modified to address stability considerations. Three modified models consisting of cylindrical body lengths of $0.4D$, $0.8D$, and $1.5D$ were investigated regarding stability improvement. The optimization technique was first used to redesign the optimum shape and CFD was then used to evaluate the aerodynamic coefficients

and also to analyze the stability on all modified flight bodies. The cylinder length of the modified optimum shape in the range of $0.8D$ to $1.5D$ was found to yield the best combination of low drag and high stability while the volume or payload is only reduced by 4% from the benchmark M549 round. The optimized body provides up to 15% drag reduction and 46% increase in gyroscopic stability while remaining dynamically stable over the considered range of the operating Mach numbers. It is concluded that the optimization procedure is an efficient and useful tool in designing generic round bodies.

6.1.2.2 Flow Physics Study

For the flow physics study, the flow mechanism which provides an adverse pressure gradient over the cylindrical body and conical boattail was explained by the equation of motion (inviscid and irrotational flow) in the axisymmetric form and simulation results. While this phenomenon is well-known, it is this that is responsible for the negative lift on the boat tail, which makes the round body more unstable when flying at finite angle-of-attack. The in-depth understanding of the flow mechanism over conical boattail is very useful to explore the potential area for enhancing aerodynamic performance and stability improvement.

6.1.2.3 Boattail Juncture Shaping Technique

Based on the in-depth understanding of the flow phenomenon over the boattail, to further enhance the aerodynamic performance of the round bodies, the boattail juncture shaping technique was employed. The effects of boattail juncture contouring on the aerodynamic performance and stability of supersonic spin-stabilized rounds were investigated by means of well-validated CFD. The contour design was applied to the standard SOCBT using four different radii of curvature: $2.5D$, $4.4D$, $8.8D$ and $13D$. By comparing the aerodynamic performance between the modified SOCBT rounds

and the standard SOCBT configuration, it was found that the modified SOCBT rounds can reduce drag by up to 5.4%. The drag reduction is less pronounced at larger Mach numbers. The gradual change of pressure over the contoured surface of the boattail juncture is the main reason for the reduction of the boattail pressure drag, which consequently results in total drag reduction. For the range of Mach numbers considered here, which is from 1.5 to 4, the results indicate that the radius of curvature should be at least $3D$ to obtain a significant drag reduction from juncture contouring.

Regarding stability, the effect of juncture shaping is insignificant for both gyroscopic and dynamic stabilities at high Mach numbers. At the low Mach number (Mach 1.5), however, it was observed that the round body with a large radius of curvature encounters dynamic instability. This is mainly due to the increase in Magnus moment derivative in this flight regime, which in turn comes from a combined effect of increasing Magnus force and the shifting of the Magnus center of pressure. Therefore, it is suggested that the radius of curvature should not exceed $8.8D$ to ensure dynamic stability at low supersonic Mach numbers.

6.1.2 Aerodynamic Testing

To confirm the aerodynamic performance of the optimum geometry obtained previously using the optimization procedures, the new optimum round and the standard 155 mm M549 bodies were tested in the large-size, high quality flow supersonic wind tunnel SWIFT (Singapore Wind Tunnel Facility) at different non-spinning test conditions (Mach 1.5 to 4, angle of attack 0° to 5°). The results showed that the new round body proposed by Jiajan et al. [74] provides significantly lower drag and pitching moment derivative in comparison to the standard 155 mm round. Furthermore, good agreement was observed between the CFD prediction and the experimental measurements. Therefore, the accuracy and effectiveness of the

optimization procedure of Jiajan et al. [74] is confirmed. The effects of Reynolds number on the aerodynamic characteristics of the rounds were also investigated. Finally, the current measurements form a valuable addition to the experimental database of round bodies at relatively high Mach numbers.

6.2 Future Work

Firstly, for the aerodynamic prediction, a more complete analysis of the flow over the spin-stabilized round bodies at three different regimes: subsonic, transonic, and supersonic speeds, the DES and LES are suggested as the main computational approach for aerodynamic investigations since RANS cannot provide the accurate dynamic aerodynamic coefficients such as Magnus effect at the transonic and subsonic regimes. In addition, these approaches will also help to provide more details in the flow over the round body, especially the base flow which is very important for improving the aerodynamic performance at these speeds.

Secondly, for the optimum geometry obtained from the optimization procedure, it should be noted that the modified optimum 155 mm round shape has a sharp nose tip as opposed to the flat nose of the standard 155 mm round. This configuration may be a major concern for heat transfer as well as manufacturing and operational issues. Nevertheless, in the case of a gun projectile flight, the round travels to reach the target in only four to five seconds. Thus, the heat transfer in the nose tip is not a significant problem. Regarding the issues of manufacturing operation, the nose tip of the modified optimum 155 mm round will be redesigned to be a blunt or flat nose. However, from the computational results of the standard 155 mm round as discussed in Chapter 3, it was found that the flat nose or the blunt nose has only a small effect on the

aerodynamic performance (increased drag about 1% and small effect on gyroscopic and dynamic stability). Thus, it can be guaranteed that the new modified optimum 155 mm rounds with the blunt nose still provides a superior performance over the standard 155 mm round.

Finally, for the aerodynamic testing, to investigate the effects of spinning motion on the aerodynamic characteristics of optimum geometry, the aeroballistics range test of the full-scale model is required. This approach will provide more a complete aerodynamics assessment since it is a real-flight test. This can be used to confirm the optimum geometry which was designed by optimization procedure is realistic.

APPENDIX A

THE MCCOY'S SEMI-EMPIRICAL SOLUTION

A. Nose shape profiles

The von Karman ogive nose shape profile is based on slender body theory [65] and is formulated as:

$$r = \frac{D}{2} \left(\frac{\theta - \frac{\sin(2\theta)}{2}}{\pi} \right)^{\frac{1}{2}}$$

where $\theta = \cos^{-1}(1-2x/L)$. Alternatively, the nose profile can be defined by a power law ($n = 3/4$) based on Newtonian impact theory [66]:

$$r = \frac{D}{2} \left(\frac{x}{L} \right)^n$$

B. Semi-empirical method based on McCoy

In this study, the semi-empirical method based on McCoy [55] has been created using MATLAB (version R2008) to evaluate the zero-lift drag coefficient on supersonic generic round shaped bodies. The total drag coefficient is given by the sum of the component drag coefficients for pressure, skin friction, and base drag as:

$$C_{D_o} = C_{D_w} + C_{D_{BT}} + C_{D_F} + C_{D_B}$$

Each of the above coefficients are calculated separately using individual formulations given below based on theoretical and empirical relations for generic round shapes. McCoy's method consists of three formulations for subsonic, transonic, and supersonic flows. In the current study, only the supersonic McCoy formulation is being considered. The valid Mach number range for the supersonic formulation is from 1.3 to 5, body diameters from 4 to 400 mm, boat tail angle from 0 to 10°, and boat tail length up to $2D$.

Nose pressure drag coefficient C_{Dw}

The nose pressure drag is a function of nose thickness ratio (τ) and nose shape parameters (R_T / R) as follows:

$$C_{Dw} = \frac{(c_1 - c_2 \tau^2) \left(\tau \sqrt{M^2 - 1} \right)^{c_3 + c_4 \tau}}{M^2 - 1} + \frac{\pi}{4} K d_M^2 C_{ps}$$

where

$$\tau = \frac{1 - d_M}{L_N}$$

$$c_1 = 0.7156 - 0.5313(R_T / R) + 0.5950(R_T / R)^2$$

$$c_2 = 0.0796 + 0.0779(R_T / R)$$

$$c_3 = 1.587 + 0.049(R_T / R)$$

$$c_4 = 0.1122 + 0.1658(R_T / R)$$

$$C_{ps} = \frac{2}{\gamma M^2} \left\{ \left[\frac{(\gamma + 1)M^2}{2} \right]^{\frac{\gamma}{\gamma - 1}} \left[\frac{\gamma + 1}{2\gamma M^2 - (\gamma - 1)} \right]^{\frac{1}{\gamma - 1}} - 1 \right\}$$

R = Radius of the circle that forms the ogive (for the M549 projectile, it is $18.88D$ as shown in Figure 3.3c)

R_T = Tangent radius of the ogive (R_T/R is the headshape parameter. Thus, $R_T/R = 0$ for a cone, $R_T/R = 1$ for a tangent ogive nose, and values between 0 and 1 describe various secant-ogive shapes.)

d_M = Meplat diameter, which is the diameter of the flat blunted nose tip on the projectile (in D)

C_{ps} = Stagnation pressure coefficient behind a normal shock at the flat nose region

K = Correction factor for leakage pressure (= 0.75)

L_N = Nose length (in D)

M = Freestream Mach number

Boat tail pressure drag coefficient C_{DBT}

$$C_{D_{BT}} = \frac{4A \tan \beta}{k} \left\{ \left[1 - e^{-kL_{BT}} \right] + 2 \tan \beta \left[e^{-kL_{BT}} \left(L_{BT} + \frac{1}{k} \right) - \frac{1}{k} \right] \right\}$$

where

$$A = A_1 e^{-\sqrt{\frac{2}{\gamma M^2}} L_{CYL}} + \frac{2 \tan \beta}{\sqrt{M^2 - 1}} - \frac{[(\gamma + 1)M^4 - 4(M^2 - 1)\tan^2 \beta]}{2(M^2 - 1)^2}$$

$$A_1 = \left[1 - \frac{3(R_T / R)}{5M} \right] \left\{ \frac{5\tau}{6\sqrt{M^2 - 1}} \left(\frac{\tau}{2} \right)^2 - \frac{0.7435}{M^2} (\pi M)^{1.6} \right\}$$

= Nose shape correction factor for supersonic boat tail drag coefficient

$$k = \frac{0.85}{\sqrt{M^2 - 1}}$$

L_{CYL} = Length of cylindrical body (in D)

L_{BT} = Length of boat tail (in D)

β = Conical boat tail angle

Skin friction drag C_{DF}

$$C_{D_F} = \frac{4}{\pi} C_{d_f} S_w$$

$$C_{d_f} = \frac{0.455}{(\log_{10} \text{Re}_l)^{2.58}} (1 + 0.21M^2)^{-0.32}$$

where

$S_w = S_{w_{NOSE}} + S_{w_{CYL}}$ = Wetted surface area (exclusive of the base)

$$S_{w_{NOSE}} = \frac{\pi}{2} L_N \left(1 + \frac{1}{8L_N^2} \right) \left[1 + \left(\frac{1}{3} + \frac{1}{50L_N^2} \right) (R_T / R) \right] = \text{Nose surface area (exclusive of the base)}$$

$S_{w_{CYL}} = \pi(L_T - L_N)$ = Cylindrical surface area (exclusive of the base)

$$\text{Re}_l = 23296.3 M L_T D$$

L_T = Total length (in D)

D = Body or cylinder diameter (units in mm)

Base drag C_{DB}

$$C_{D_B} = \frac{2d_B^2}{\gamma M^2} \left[1 - \frac{p_B}{p} \right]$$

$$\frac{p_B}{p} = \left[1 + 0.09M^2 \left(1 - e^{-L_{CYL}} \right) \right] \left[1 + 0.25M^2 (1 - d_B) \right]$$

where

p_B = Base pressure

p = Freestream pressure

d_B = Base diameter (in D)

APPENDIX B

THE MCCOY'S SEMI-EMPIRICAL CODES AND OPTIMIZATION CODES

A. Semi-Empirical Codes

```

%% Semi-Empirical Codes (McCoy's Solution) (Supersonic Regimes)
clc
clear all

%% Input parameters
%% (Geometries parameters)
%% Example M549 155mm projectile
LN=3.01; % Nose Lenght (Body diameters)
LT=5.650; % Total Lenght (Body diameters)
Dref=155; % Reference diameters (mm)
Dm=0.09; % Meplat (Body diameters)
RTR=0.5; % Nose shape parameter
Db=0.848 % Base diameter (Body diameters)
LBT=0.580; % Boattail Length (Body diameters)
Lcyl=LT-LN-LBT; % Cylindrical Lenght (Body diameters)
beta=7.30*pi/180; % Boattail angle (radians)

%% Freestream Mach number
M=[1.8, 2, 3,4];

%% skin friction calculation CDSF (Supersonic) (Assume fully
turbulent)
%% boundary layer at T_free stream 288 K
for i=1:length(M)
    TA=(1-Dm)/LN;
    M2(i)=(M(i)).^2;
    Re(i)=23296.3.*M(i).*LT*Dref
    Cft(i)=0.455./(log10(Re(i)).^2.58)*(1+(0.21.*(M2(i))))).^0.32;
    Dum=1+(0.33+(0.02/LN^2)).*RTR;
    SWN=1.5708*LN*Dum*(1+(1/(8*LN^2)))
    SWCYL=3.1416*(LT-LN)
    SW=SWN+SWCYL
    CDSFT(i)=1.2732*SW*Cft(i)
end
CD_SKIN=CDSFT'

%% Base drag calculation
for i=1:length(M)
    PB2(i)=1/(1+0.2477.*M2(i)+0.0345.*M2(i).*M2(i));
    PB4(i)=(1+0.09.*M2(i)*(1-exp(LN-LT)))*(1+0.25.*M2(i).*(1-
Db));
    PBP1(i)=PB2(i).*PB4(i);
    CDB(i)=(1.4286*(1-PBP1(i)).*(Db*Db))/M2(i);
end
CD_BASE=CDB'

%% Boattail calculation
for i=1:length(M)
    EXP(i)=exp(-Lcyl.*1.1952/M(i));
    TAN=tan(beta);
    M_rt(i)=sqrt(M2(i)-1);
    A1(i)=(5*TA)/(6.*M_rt(i))+(0.5*TA)^2....
    -((0.7435/M2(i)).*(TA.*M(i)).^1.6);
    A2(i)=(1-(0.6*RTR)/(M(i))).*A1(i);
    k(i)=0.85./M_rt(i);

```



```

EXP2(i)=exp(-LBT.*k(i));
k1(i)=1./k(i);
A(i)=(A1(i).*EXP(i))+(2*TAN/M_rt(i))-((2.4.*(M2(i)).^2)...
    -(4.*(M_rt(i)).^2))*TAN^2/(2*(M_rt(i)).^4);
CDBT(i)=(4*A(i)*TAN.*k1(i)).*...
    *((1-EXP2(i))+2*TAN.*(EXP2(i).*(LBT+k1(i))-k1(i)));

end
CD_BOATTAIL=CDBT'

%% Nose wave drag
K=0.75;
for i=1:length(M)
SRE(i)=M2(i)-1;
RE(i)=sqrt(SRE(i));
FRE(i)=RE(i);
SSMC=1+0.368*(TA.^1.85);
C1=0.7156-0.5313*(RTR)+0.5950*(RTR^2);
C2=0.0796+0.779*(RTR);
C3=1.587+0.049*(RTR);
C4=0.1122+0.1658*(RTR);
RT(i)=1/(RE(i).*RE(i));
CDHT(i)=(C1-C2*(TA.^2))*RT(i)*((TA.*RE(i)).^(C3+C4*TA));

PTP(i)=((1.2.*M2(i)).^3.5).*((6/(7.*M2(i)-1.2)).^2.5);
Cps(i)=1.428571429.*(PTP(i)-1)./M2(i);
CDHM(i)=pi/4*K*Dm^2*Cps(i);
CMEP(i)=((1.122.*(PTP(i)-1)*Dm^2))./M2(i);
CDHM(i)=0.95.*CMEP(i);
CDH(i)=CDHT(i)+CDHM(i);
end
CD_WAVE=CDH'

%% Total drag calculation
for i=1:length(M)
CD0(i)=CD_WAVE(i)+CD_BASE(i)+CD_SKIN(i)+CD_BOATTAIL(i);
end
CD0

%% Print out the results
figure
plot(M,CD0,'rS--')
xlabel('Mach number')
ylabel('Cd_Total drag')

```

B. Optimization Codes

Main Codes

```
close all;clear all;clc

global M

M=2;
%% set the boundary for LN,Lcyl,LBT,beta
up=[4.8;3;2;10*pi/180];
low=[3.2;0;0;0];

%% initial guess values [Nose length; Cylindrical length; Boattail
length;%% Boattail angle]

x0=[2;1;1;0.1];

%% [x,histout,costdata] = projbfgs(x0,@opt_fun,up,low)

Aeq=[1,1,1,0]; %Equality constraint
equation(LN+Lcyl+LBT+Beta)=Aeq

beq=5.645;      % Total length (body diameters)

%% output
[x,fval,exitflag,output,lambda,GRAD, HESSIAN] =
fmincon(@MCDrag1,x0,...
Aeq,beq,[],[],low,up)

optimum_configuration=x;
Optimum_Nose_length=x(1)
Optimum_Cylindrical_length=x(2)
Optimum_Boattail_length=x(3)
Optimum_Boat_tail_angle=x(4)*180/pi
minimum_Total_drag=fval
```

Sub function codes

```
function CD0=MCDrag1(x)
global M

LN=x(1); % Nose length vector
Lcyl=x(2); % Cylindrical length vector
LBT=x(3); % Boattail length vector
beta=x(4); % Boattail angle vector

%% Input parameters

LT=LN+Lcyl+LBT; % Total length
Dref=155; % Reference body diameters (mm)
Dm=0; % no meplat
RTR=0.5; % Nose shape parameter
Db=1-2*LBT*tan(beta); % Base diameter (body diameters)
```

```

%% skin friction calculation CDSF (Supersonic) Assume fully
turbulent
%% boundary layer at T_free stream 288 K
TA=(1-Dm)/LN;
M2=(M).^2;
Re=23296.3.*M.*LT*Dref;
Cft=0.455./(log10(Re).^2.58)*(1+(0.21.*(M2))).^-0.32;
Dum=1+(0.33+(0.02/LN^2))*RTR;
SWN=1.5708*LN*Dum*(1+(1/(8*LN^2)));
SWCYL=3.1416*(LT-LN);
SW=SWN+SWCYL;
CDSFT=1.2732*SW*Cft;

CD_SKIN=CDSFT;

%% Base drag calculation

PB2=1/(1+0.2477.*M2+0.0345.*M2.*M2);
PB4=(1+0.09.*M2*(1-exp(LN-LT)))*(1+0.25.*M2.*(1-Db));
PBP1=PB2.*PB4;
CDB=(1.4286*(1-PBP1)*(Db*Db))/M2;

CD_BASE=CDB;

%% Boattail calculation
EXP=exp(-Lcyl.*1.1952/M);
TAN=tan(beta);
M_rt=sqrt(M2-1);
A1=(5*TA)/(6.*M_rt)+(0.5*TA)^2-((0.7435/M2).*(TA.*M).^1.6);
A2=(1-(0.6*RTR)/(M)).*A1;
k=0.85./M_rt;
EXP2=exp(-LBT.*k);
k1=1./k;
A=(A1.*EXP)+(2*TAN/M_rt)-((2.4.*(M2).^2)...
    -(4.*(M_rt).^2))*TAN^2/(2*(M_rt).^4);

CDBT=(4*A*TAN.*k1).*(1-EXP2)+2*TAN.*(EXP2.*(LBT+k1)-k1);

CD_BOATTAIL=CDBT;

%% Nose wave drag
K=0.75;
SRE=M2-1;
RE=sqrt(SRE);
FRE=RE;
SSMC=1+0.368*(TA.^1.85);
C1=0.7156-0.5313*(RTR)+0.5950*(RTR^2);
C2=0.0796+0.779*(RTR);
C3=1.587+0.049*(RTR);
C4=0.1122+0.1658*(RTR);
RT=1/(RE.*RE);
CDHT=(C1-C2*(TA.^2))*RT*((TA.*RE)^(C3+C4*TA));
PTP=((1.2.*M2).^3.5).*(6/(7.*M2-1.2)).^2.5;
Cps=1.428571429.*(PTP-1)./M2;
CDHM=pi/4*K*Dm^2*Cps;
CMEP=((1.122.*(PTP-1)*Dm^2))./M2;
CDHM=0.95.*CMEP;

```

```
CDH= CDHT+CDHM;  
  
CD_WAVE=CDH;  
  
%% Total drag calculation  
  
CD0=CD_WAVE+CD_BASE+CD_SKIN+CD_BOATTAIL;
```

APPENDIX C

THE FLIGHT CONDITIONS DESIGN AND STRUCTURAL ANALYSIS

Table 1: Forces, moment and test conditions for full-scale standard 155 mm M549 round flight body

Mach	P0 (N/m²)	T0 (K)	P (N/m²)	T (K)	Cd	CN	Cm	Axial (N)	Normal (N)	Moment (Nm)	Re_D
1.8	582,193	300	101,325	182.04	0.278	0.0872	0.157	1205.47	378.12	105.52	1.20E+07
2	792,812	300	101,325	166.67	0.26	0.0924	0.139	1391.88	494.65	115.34	1.50E+07
3	3,721,943	300	101,325	107.14	0.2	0.0974	0.122	2409.02	1173.19	227.77	4.28E+07
4	15,384,703	300	101,325	71.43	0.15	0.0974	0.115	3212.03	2085.68	381.70	1.07E+08

Table 2: Forces, moment and test conditions for 1/1.668 scale standard 155 mm M549 model with Reynolds numbers fully matched

Mach	P0 (N/m²)	T0 (K)	P (N/m²)	T (K)	Cd	CN	Cm	Axial (N)	Normal (N)	Moment (Nm)	Re_D
1.8	971,098	300	169,010	182.04	0.278	0.0872	0.157	722.71	226.69	67.93	1.20E+07
2	1,322,411	300	169,010	166.67	0.26	0.0924	0.139	834.46	511.36	59.62	1.50E+07
3	6,208,201	300	169,010	107.14	0.2	0.0974	0.122	1444.26	381.56	39.76	4.28E+07
4	25,661,685	300	169,010	71.43	0.15	0.0974	0.11	1925.68	189.35	17.79	1.07E+08

Table 3: Forces, moment and test conditions for 1/1.668 scale standard 155 mm M549 model at maximum facility total pressure

Mach	P0 (N/m²)	T0 (K)	P (N/m²)	T (K)	Cd	CN	Cm	Axial (N)	Normal (N)	Moment (Nm)	Re_D
1.8	750,000	300	130,530	182.04	0.278	0.0872	0.157	558.16	175.08	67.93	9.24E+06
2	820,000	300	104,800	166.67	0.26	0.0924	0.139	517.43	511.36	59.62	9.31E+06
3	1,300,000	300	35,391	107.14	0.2	0.0974	0.122	302.43	381.56	39.76	8.97E+06
4	1,500,000	300	9,879	71.43	0.15	0.0974	0.11	112.56	189.35	17.79	6.28E+06

a)

b)

The dimensions of two different wind tunnel models (1/1.668 scale) with sting balance, dimensions in mm: a) optimum 155 mm geometry and b) standard 155 mm M549 round.

Structural Analysis of Wind Tunnel Models

A preliminary finite element analysis (using Ansys Workbench) [72] has been carried out to calculate the stresses in the wind tunnel model. Steel is selected to be the fabrication material of the wind tunnel model which is used as one of the input parameters in the finite element analysis. The yielding stress of steel is approximately 250 MPa. The recommendation for the safety factor is that it should be greater than 5 to ensure that the model is sufficiently strong throughout the test Mach numbers.

For the FEM analysis, two different mesh sizes have been created to compare the differences and the boundary conditions of FEM have been set based on the CFD calculations (Chapter 3 and 4). The computational mesh sizes and boundary conditions for finite element analysis are shown in Figure 1 and 2.

Figure 1: The two different mesh sizes for structural analysis of 155 mm M549 model: coarse mesh (above) and fine mesh (bottom)

Figure 2: The boundary conditions for steady load of the standard 155 mm M549 model

From Figure 2, in terms of numerical procedures, it should be noted that the contact surface of three components (nose, adaptor, and body portion) make the physical models to be non-linear problem which is more complicate than the linear system. However, the non-linear solution can be solved by Newton and Raphson method that inside the ANSYS package. Based on this non-linear solution, the contact surface can be set as the boundary condition for FEM solution. Moreover, in Figure 2, the boundary conditions have been set up based on the steady load conditions at Mach 2 at which the critical load is generated due to the high dynamic pressure.

However, in the actual test, the unsteady critical load (starting load) can be generated at the high Mach number (Mach 4) which may create high stresses beyond the yield point. Based on the recorded data of previous testing, the starting load (normal force) at Mach 4 is approximately 6000 N. Thus, to ensure that the stress due to starting is not beyond the yielding point, FEM is again used for structural analysis. Furthermore, the boundary conditions for the unsteady load are also modified to enhance the accuracy as shown in Figure 3.

For the structural analysis of the optimum shape model, it should be noted that the structural analysis procedure is about the same as that for the standard 155 mm wind tunnel model (just changing the values of axial force, normal force, and location of center pressure). The boundary conditions of this model are shown in Figure 4. From Figure 4, it can be seen that the axial force is separated into two components to avoid the concentrated load acting on the pointed nose.

Figure 3: The boundary conditions for the unsteady load (starting load at Mach 4) for the standard 155 mm M549 model

Figure 4: The boundary conditions for the steady load for the optimum 155 mm model

Moreover, to ensure that the stress due to the starting load (approximately 6000 N, at Mach 4) of the optimum geometry is not beyond the yielding point, the modified boundary conditions for the unsteady load are also shown in Figure 5.

Figure 5: The boundary conditions for the unsteady load (starting load at Mach 4) for the optimum 155 mm model

After the residual of the FEM solution has reached 10^{-6} , the final results can be obtained for each model. The results of FEM for steady load and unsteady load of the standard 155 mm and the optimum geometry models are shown in Figures 6 to 8. From the structural analysis results using FEM as shown in Figures 6 to 8, it is evident that both wind tunnel models yielded safety factors greater than 5. Thus, the wind tunnel models can be used with confidence for wind tunnel testing throughout all intended Mach numbers.

a)

b)

Figure 6: FEM results for steady load of two different models: a) the standard 155 mm M549 model and b) the optimum 155 mm model

a)

b)

Figure 7: FEM results for unsteady load (critical load at Mach 4) of the standard 155 mm M549 model; a) von-mise stress contour and b) safety factor

a)

b)

Figure 8: FEM results for unsteady load (critical load at Mach 4) of optimum 155mm model a) von-mise stress contour and b) safety factor

APPENDIX D

THE TESTING CONDITIONS AND EXPERIMENTAL RESULTS

Wind tunnel testing conditions of two different rounds: M549 155 mm round and optimum 155 geometry

Test conditions (155mm M549 round (1/1.668 scale) D=92.88mm)								Objective
M	Pt x10E-5 Pa	Tt (K)	P (Pa)	q(Pa)	Re_dx10E-6	alpha	w/wo trip	
1.8	1.999463	300	34798.71	90198.26	2.459	(-5 to 5)	WO	Cd, CL, Cm
2	1.999463	300	25554.04	81772.93	2.268	(-5 to 5)	W/WO	Cd, CL, Cm
2.5	3.516297	300	20580.06	102900.3	3.138	(-5 to 5)	WO	Cd, CL, Cm
3	5.51576	300	15015.93	108114.7	3.798	(-5 to 5)	W/WO	Cd, CL, Cm
3.5	8.825216	300	11570.67	113392.6	4.702	(-5 to 5)	WO	Cd, CL, Cm
4	15.513075	300	10217.05	130778.2	6.480	(-5 to 5)	W/WO	Cd, CL, Cm

Test conditions (Optimum 155mm round (1/1.668 scale) D=92.88mm)								Objective
M	Pt x10E-5 Pa	Tt (K)	P (Pa)	q (Pa)	Re_dx10E-6	alpha	w/wo trip	
1.8	1.999463	300	34798.71	90198.26	2.459	(-5 to 5)	W/WO	Cd, CL, Cm
2	1.999463	300	25554.04	81772.93	2.268	(-5 to 5)	W/WO	Cd, CL, Cm
2.5	3.516297	300	20580.06	102900.3	3.138	(-5 to 5)	W/WO	Cd, CL, Cm
3	5.51576	300	15015.93	108114.7	3.798	(-5 to 5)	W/WO	Cd, CL, Cm
3.5	8.825216	300	11570.67	113392.6	4.702	(-5 to 5)	WO	Cd, CL, Cm
4	15.513075	300	10217.05	130778.2	6.480	(-5 to 5)	WO	Cd, CL, Cm

Pitching moment coefficient versus angle of attacks (-5° to 5°) at Mach numbers ranging from 1.8 to 4: Standard M549
155 mm round

Normal force coefficient versus angle of attack (-5° to 5°) at Mach numbers ranging from 1.8 to 4: Standard M549 155 mm round

Pitching moment coefficient versus angle of attacks (-5° to 5°) at Mach numbers ranging from 1.8 to 4: Optimum 155 mm round

Normal force coefficient versus angle of attacks (-5° to 5°) at Mach numbers ranging from 1.8 to 4: Optimum 155 mm round

References

- [1] Koh, P.T., Goh, E. K. R., Lim, H. B., "SWIFT - Singapore's National Aeronautical Testing Asset," *AIAA* 2005-7615, December 2005.
- [2] Shrewsbury, G.D., "Effect of Boattail Juncture Shape on Pressure Drag Coefficients of Isolated Afterbodies," *NASA TM X- 1517*, Mar 1968.
- [3] Sivan D.D. and Jerney C., "Wind Tunnel Tests of the Aerodynamic Characteristics of a 155 mm Artillery Shell," Aeronautical Research Lab., Flight Mechanics Technical Memorandum 417 (ARL-FLIGHT-MECH-TM-417), Australia, 1989.
- [4] Anderson, J.D., Jr., *Introduction to Flight: Fourth Edition*: McGraw-Hill Higher Education, Boston, MA, 2000.
- [5] White, F.M., *Viscous Fluid Flow 2nd Edition*: McGraw-Hill, 1991.
- [6] Chapman, D.R., "An Analysis of Base Pressure at Supersonic Velocities and Comparison with Experiment," *NACA Report 1051*, 1951.
- [7] Tan, W.C., "Aerodynamic Drag prediction of Emerging Projectile and Missile Configurations," Naval Post Graduate School, Monterey, California, 2010.
- [8] Krasnov, N.F., *Aerodynamics of Bodies of Revolution*: translated from the second Russian edition, The Rand Corporation, Santa Monica, California, 1970.
- [9] Anders S.P., and George, N., "An Improved Projectile Boattail II," BRL Memorandum Report No. 1886, U.S. Army Ballistics Research Laboratories, Aberdeen Proving Ground, Maryland, 1976.
- [10] Chee, M.F., "Development of an Artillery Accuracy Model," Naval Post Graduate School, Monterey, California, 2006.
- [11] Kneubuehl, B.P., *Wounds ballistics: Basics and Applications*: Springer-Verlag Berlin, Heidelberg, 2011
- [12] Desperito, J, and Siltan, S.I, "Capabilities for Magnus Prediction in Subsonic and Transonic Flight," U.S. Army PDECOM, Army Research Laboratory, Aberdeen Proving Ground, Maryland 21005-5069, 2008.
- [13] Sturek W.B., Dwyer H.A., Kayser L.D., Nietubicz C.J., Reklis R.P., and Opalka K.O., "Computations of Magnus effects for a yawed, spinning body of revolution," *AIAA Journal* 16(7): pp. 687–92, 1978.
- [14] Martins, J.C., "On Magnus Effect Caused by the Boundary Layer Displacement Thickness on Bodies of Revolution at small Angles of Attack," BRL Rep. 870,

Department of the Army Prou. 5B03-03-001, Ord. Res. and Dev. Proj. TB3-0108, Aberdeen Proving Ground, Maryland, June 1955.

- [15] Kelly, H.R., "An Analytical Method for Predicting the Magnus Forces and Moments on Spinning Projectiles," TM-IG34, U. S. Naval Ordnance Test Station, China Lake, California, 1954.
- [16] Kelly, H.R., and Thacker, G.R., "The Effect of High Spin on the Magnus Force on a Cylinder at Small Angles of Attack," NAVORD Report 5036, U. S. Naval Ordnance Test Station, China Lake, California, 1956.
- [17] Murphy C.H., "Free Flight Motion of Symmetric Missiles," Report No. 1216, Ballistic Research Laboratories, Aberdeen Proving Ground, Maryland, 1963.
- [18] Weinacht, P., Sturek, W.B., and Schiff, L. B., "Navier-Stokes Predictions of Pitch Damping for Axisymmetric Projectiles," *Journal of Spacecraft Rockets*, 34 (6), pp.753-761, 1997.
- [19] Taylor, G.I, and Maccoll, J.W., "The Air pressure on Cone Moving at High Speed," *Proc Roy Soc (London) ser A*, vol 139, pp. 278-311, 1933.
- [20] Van Dyke, M.D., "First and Second-Order Theory of Supersonic Flow Past Bodies of Revolution," *Journal of Aeronautical Sciences*, Vol. 18 No.3, pp. 161-179, March 1951.
- [21] Van Dyke, M.D., "A Study of Second-Order Supersonic Flow Theory," NACA Rept. TR 1081, April 1952.
- [22] Syvertson, C.A., and Dennis, D.D., "A Second-Order Shock Expansion Method Applicable to Bodies of Revolution Near Zero-lift," NACA TR 1323, 1957.
- [23] Eggers, A.J., and Savin, R.C., "Approximate Methods for Calculating the Flow about Nonlifting Bodies of Revolution at High Supersonic Airspeeds," NACA TR 2579, 1951.
- [24] Jacksons, C.M., Sawyer, W.C., and Smith, R.S., "A Method for Determining Surface Pressure on Blunt Bodies of Revolution at Small Angles of Attack in Supersonic Flow," NASA TN P-4865, 1968.
- [25] Moore, F.G., *Approximate Methods for Weapon Aerodynamics*: Naval Surface Warfare Center, Progress in Astronautics and Aeronautics, Volume 186, 2000.
- [26] Van Driest, E.R., "Turbulent Boundary Layers in Compressible Fluids," *Journal of the Aeronautical Sciences*, vol. 18, No. 3, pp. 145-160, 1951.
- [27] Schlichting, H., *Boundary Layer Theory*: McGraw-Hill, 2955, 1951.
- [28] Missile DATCOM is ITAR restricted:
<http://home.coosahs.net/dmarlowe/itar.htm>

- [29] Moore, F.G., and Hymer, T.C., "The 2005 Version of the Aero prediction Code: Part II- Users Guide," API Report no. 2, June 2004.
- [30] Sahu, J., "Drag predictions for projectiles at transonic and supersonic speeds. ABRL Maryland," AD-A171 462, BRL-MR-3523, 1986.
- [31] Balwind, B.S., and Lomax, H., "Thin-Layer Approximation and Algebraic Model for Separated turbulent flows," AIAA paper No-78-257, 1978.
- [32] Kayser, L.D., "Base Pressure Measurements on a Projectile shape at Mach numbers from 0.91 to 1.2," US Army Ballistic Research Laboratory, Aberdeen Proving Ground, Maryland, ARBRL-MR-03353, 1984.
- [33] Tucker, P.K., and Shyy, W., "A Numerical Analysis of Supersonic Flow over an axisymmetric afterbody," AIAA paper No. 93-2347, 1993.
- [34] Suzen, Y.B., Hoffmann, K.A., and Forsythe, J.R., "Application of Several Turbulence Models for High Speed Shear Layer Flows," AIAA paper No. 99-0933, 1999.
- [35] Forsythe, J.R., Strang, W., and Hoffmann, K.A., "Validation of Several Reynolds-Averaged Turbulence Models in a 3D Unstructured Grid Code," AIAA paper No. 2000-2552, 2000.
- [36] Menter, F.R., "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications," *AIAA journal*, 32(8), pp.1598-1650, 1994.
- [37] Wilcox, D.C., *Turbulence Modeling for CFD, 2nd Edition*: DCW Industries, La Canada, CA, 1998.
- [38] Silton, S.I., "Navier-Stokes Computations for a Spinning Projectile from Subsonic to Supersonic Speeds," ARL-TR-2850; U.S. Army Research Laboratory: Aberdeen Proving Ground, MD, September 2002.
- [39] DeSpirito, J., and Heavey, K. R., "CFD Computation of Magnus Moment and Roll Damping Moment of a Spinning Projectile," AIAA-2004-4713; Proceedings of the AIAA Atmospheric Flight Mechanics Conference, Providence, RI, 16–19 August 2004.
- [40] DeSpirito, J., and Plostins, P., "CFD Prediction of M910 Projectile Aerodynamics: Unsteady Wake Effect on Magnus Moment," AIAA-2007-6580; Proceedings of the AIAA Atmospheric Flight Mechanics Conference, Hilton Head, SC, 20–23 August 2007.
- [41] Simon, F., Deck, S., Guillen, P., and Sagaut, P., "Reynolds-Averaged Navier-Stokes/Large- EddySimulations of Supersonic Base Flow," *AIAA Journal*, 44 (11), pp. 2578–2590, 2006.
- [42] DeSpirito, J., Silton, S.I., and Weinacht, P., "Navier-Stokes Predictions of Dynamic Stability Derivatives: Evaluation of Steady State Methods," AIAA-

2008-0214; Proceedings of the 46th Aerospace Sciences Meeting, Reno, NV, January 2008.

- [43] Schmidt, L.E., and Murphy, C.H., "The Aerodynamic Properties of the 7-Caliber Army-Navy Spinner Rocket in Transonic Flight," BRL-MR-775; U.S. Army Ballistics Research Laboratory: Aberdeen Proving Ground, MD, March 1954.
- [44] Plostins, P., McCoy, R. L., and Wagoner, B. A., "Aeroballistics Performance of the 25mm M910 TPDS-T Range Limited Training Projectile," U.S. Army Ballistic Research Laboratory, BRL-MR-3886, Aberdeen Proving Ground, MD, Jan. 1991.
- [45] Viswanath, P. R., "Flow Management Techniques for Base and After body Drag Reduction," *Prog Aerospace Sci*, **32**, pp 79-129, 1996.
- [46] Suliman, M.A., Mahmoud, O.K., Al-Sanabawy, M.A., and Abdel-Hamid, O.E., "Computational Investigation of Base Drag Reduction for a Projectile at Different flight regimes," 13th International Conference on Aerospace Science & Aviation Technology, Military Technical College, Kobry Elkobbah, Cairo, Egypt, 2009.
- [47] Ibrahim, A. and Filippone, A., "Effect of Porosity Strength on Drag Reduction of a Transonic Projectile," *Journal of Aircraft*, 44, (1), pp 310-316, January 2006.
- [48] Ibrahim, A. and Filippone, A., "Supersonic aerodynamics of a Projectile with slot cavities," *Aeronautical Journal*, 114, (3416), pp 15-24, Jan. 2010.
- [49] Nicolas, J.M., "Optimal Bodies for Minimum Total Drag at Supersonic Speeds," NSWC TR-80-208 Dahlgren, VA 22448, May 1980.
- [50] Van Dyke, M.J., "The Similarity Rules for Second-Order Subsonic and Supersonic Flow," NACA Technical Note 3875, October 1956.
- [51] Van Driest, E.R., "Turbulent Boundary Layers in Compressible Fluids," *Journal of the Aeronautical Sciences*, 18, (3), pp 145-160, 216, 1951.
- [52] Aupoix, B., "Introduction to Statistical One-point Closures", Lecture notes, École de Mécanique des Fluides Numérique, France, 2009.
- [53] Reynolds, O., "On the Dynamical Theory of Incompressible Viscous Fluids and the Determination of the Criterion," *Philosophical Transactions of the Royal Society of London. A*, vol. 186, pp. 123-164, 1895.
- [54] Schmitt, F.G., "About Boussinesq's turbulent viscosity hypothesis: historical remarks and a direct evaluation of its validity," *Comptes Rendus Mécanique*, 335 (9-10), pp. 617-627, 2007.

- [55] McCoy, R., L., "MC Drag - A Computer Program for Estimating the Drag Coefficients of Projectiles," U.S. Army Armament Research & Development Command Report No. ARBRL-TR-02293 (1981): 74.
- [56] Van Dyke, M.J., "The Similarity Rules for Second-Order Subsonic and Supersonic flow," NACA Technical Note 3875, October 1956.
- [57] Taylor, G.I. and Maccoll, J.W., "The Air Pressure on a Cone Having at High Speeds," *Proc. Roy. Soc. A.*, vol. 139, pp. 278-311, 1933.
- [58] FLUENT 2006, "FLUENT 6.3 User's Guide," ANSYS, Inc.
- [59] Bardina, J.E., Huang, P.G. and Coakley, T.J., "Turbulence modelling, validation, testing and development," NASA Technical Memorandum 110446, 1997.
- [60] Jones, W.P., and Launder, B.E., "The Prediction of Laminarization with a Two-Equation Model of Turbulence," *International Journal Heat and Mass Transfer*, Vol.15, pp. 301-314, 1972.
- [61] Weinacht, P., "Navier-Stokes Predictions of Pitch-Damping for a Family of Flared Projectiles," AIAA Paper 91-3339, AIAA 9th Applied Aerodynamics Conference, Baltimore, MD, September 1991.
- [62] DeSpirito, J., Sifton, S.I., and Weinacht, P., "Navier-Stokes Predictions of Dynamic Stability Derivatives: Evaluation of Steady State Methods," AIAA-2008-0214; Proceedings of the 46th Aerospace Sciences Meeting, Reno, NV, January 2008.
- [63] Sturek, W.B., Kayser, L.D, and Yanta, W., "Measurements in the Turbulent Boundary Layer of Yawed Spinning Body of Revolution at Mach3 with a Laser Velocimeter and Impact Probe," *AIAA Journal*, No.78-824, 1978.
- [64] Kline, R., Herrmann, W.R., and Oskay, V., "A determination of the aerodynamic coefficients of the 155 mm, M549 Projectile," Technical Report No. 4764, Picatinny Arsenal, Dover, New Jersey, BD 002730L, November 1974.
- [65] von Karman, T.H., "The problems of Resistance in Compressible Fluids," *Cl. Sci. Frs. Mat journal*, 14, pp. 222 – 276, 1936.
- [66] Cole, J.D., "Newtonian Flow Theory for Slender Bodies," U.S. Air Force, Project Rand, R.M. 1633, Feb. 1956.
- [67] Kenneth, J.B., "Numerical Method for Chemical Engineering Applications in MATLAB," Cambridge University Press, New York, 2006.
- [68] Platou, A.S., and Neilson, G.I.T., "An Improved Projectile Boat Tail II," BRL Memorandum Report No. 1886, U.S. Army Ballistics Research Laboratories, Aberdeen Proving Ground, Maryland, 1976.

- [69] Mason L., Devan, L., Moore, F.G., and McMillan, D., "Aerodynamics Design Manual for Tactical Weapons", NSWC TR 81-156, July 1981.
- [70] Jenke, L.M., and Shadow, T.O., "Experimental Magnus and Static Stability Characteristics of Ballistic Projectiles with Various Boattail Angles and Lengths at Mach numbers from 0.5 through 2.5," AEDC-TR-75-40, AFATL-TR-75-52 (AD A013759), August 1975.
- [71] Liepmann, H. W., and Roshko, A., *Elements of Gas Dynamics*: John Wiley & Sons Inc., 1957.
- [72] ANSYS, Inc. website. Available: <http://www.ansys.com>.
- [73] Abernethy, R.B., and Thompson, J.W., "Handbook - Uncertainty in Gas Turbine Measurements," AEDC-TR-73-5, February 1973.
- [74] Jiajan, W., Chue, R., Nguyen, T. and Yu, S., "Optimization of Round Bodies for Aerodynamic Performance and Stability at Supersonic Speeds," *Aeronautical J*, in print.
- [75] Reklis, R. P., Sturek, W. B., and Baily, F. R., "Computation of Transonic Flow Past Projectiles at Angle of Attack," AIM 11th Fluid and Plasma Dynamics Conference, A I M Paper No.78-1182, July 1978.
- [76] Kim, H.D., and Setoguchi, T., " Shock Induced Boundary Layer Separation," 8th International Symposium on Experimental and Computational Aerothermodynamics of Internal Flows, Lyon, France, July-2007.
- [77] Herrin, J.L., and Dutton, J.C., "Supersonic Base Flow Experiments in the Near-Wake of a Cylindrical Afterbody," AIAA Journal, Vol.32, pp.77–83, Jan.1994.
- [78] Fureby, C., Nilsson, Y., and Andersson, K., "Large Eddy Simulation of Supersonic Base Flow," AIAA Paper 99-0426, Jan. 1999.
- [79] Baurle, A.R., and Tam, J.C., "Hybrid Simulation Approach for Cavity Flows: Blending, Algorithm, and Boundary Treatment Issues," AIAA Journal, Vol.41, No.8, pp.1463–1480, Aug. 2003.