Winter 2016

The Effects of Rarefaction and Thermal Non-Equilibrium on a Blunt Body and a Bicone in Hypersonic Flow and Their Shape Optimization for Reducing Both Drag and Heat Transfer

Samuel Gardner
Washington University in St Louis

Follow this and additional works at: https://openscholarship.wustl.edu/eng_etds
Part of the Aerodynamics and Fluid Mechanics Commons

Recommended Citation
https://openscholarship.wustl.edu/eng_etds/197

This Thesis is brought to you for free and open access by the McKelvey School of Engineering at Washington University Open Scholarship. It has been accepted for inclusion in Engineering and Applied Science Theses & Dissertations by an authorized administrator of Washington University Open Scholarship. For more information, please contact digital@wumail.wustl.edu.
The Effects of Rarefaction and Thermal Non-Equilibrium on a Blunt Body and a Bicone in Hypersonic Flow and Their Shape Optimization for Reducing Both Drag and Heat Transfer

by

Samuel Taylor Gardner

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

Master of Science

December 2016
Saint Louis, Missouri
Contents

List of Tables ......................................................... iv
List of Figures ....................................................... v
Acknowledgments .................................................... viii
Abstract ............................................................. x

1 Introduction ......................................................... 1
  1.1 Motivation ................................................... 1
  1.2 Background .................................................. 2
    1.2.1 Hypersonic Flow Characteristics ......................... 2
    1.2.2 Navier-Stokes Equations Including Rotational Non-equilibrium 3
    1.2.3 Rarefaction Effects ...................................... 6
    1.2.4 Maxwell’s Slip Flow Boundary Conditions .............. 7
  1.3 ZLOW ......................................................... 8

2 Simulation of an Axisymmetric Blunt Body Flow in Equilibrium Using a One Temperature Model (NS1T) and in Thermal Non-Equilibrium using a Two Temperature Model (NS2T) ................................................. 9
  2.1 Axisymmetric Blunt Body Parameters ......................... 9
    2.1.1 Geometry ................................................. 9
    2.1.2 Mesh Generation .......................................... 10
    2.1.3 Free Stream Flow Conditions ......................... 12
  2.2 Computed Results for Axisymmetric Blunt Body in Equilibrium (NS1T) and Non-Equilibrium (NS2T) .......................................................... 13
    2.2.1 Flow Field Properties .................................. 13
    2.2.2 Surface Properties ...................................... 17
  2.3 Axisymmetric Blunt Body in Equilibrium (NS1T) vs Non-Equilibrium (NS2T): Discussion of Results .......................................................... 23

3 Simulation of an Axisymmetric Bicone Flow in Equilibrium Using a One Temperature Model (NS1T) and in Thermal Non-Equilibrium using a Two Temperature Model (NS2T) ................................................. 30
  3.1 Axisymmetric Bicone Parameters .............................. 30
3.1.1 Geometry .......................... 30
3.1.2 Mesh Generation .......................... 32
3.1.3 Free Stream Flow Conditions .......................... 33

3.2 Computed Results for Axisymmetric Bicone in Equilibrium (NS1T) and Thermal Non-Equilibrium (NS2T) .......................... 34
3.2.1 Flow Field Properties .......................... 34
3.2.2 Surface Properties .......................... 39

3.3 Axisymmetric Bicone in Equilibrium (NS1T) vs Non-Equilibrium (NS2T): Discussion of Results .......................... 44

4 Shape Optimization of a Blunt Body for Reduction of Both Drag and Heat Transfer Using a Genetic Algorithm .......................... 49
4.1 Genetic Algorithm Overview .......................... 49
4.2 Shape Optimization of a Blunt Body using a Multi-Objective Genetic Algorithm .......................... 53
4.2.1 MOGA Parameters for Axisymmetric Blunt Body Shape Optimization .......................... 53
4.2.2 Computational Resources .......................... 55
4.2.3 Blunt Body MOGA Results .......................... 55

5 Shape Optimization of a Bicone for Reduction of Both Drag and Heat Transfer Using a Genetic Algorithm .......................... 67
5.1 MOGA Parameters for Axisymmetric Bicone Shape Optimization .......................... 67
5.2 Bicone Optimization Computational Resources .......................... 69
5.3 Bicone MOGA Results .......................... 70

6 Conclusions .......................... 79

References .......................... 82

Vita .......................... 83
# List of Tables

2.1 Blunt body mesh validation study [3] ................................................. 11
2.2 Free stream parameters for blunt body simulations ............................. 13
2.3 Blunt body results with thermal equilibrium ........................................ 24
2.4 Blunt body results with thermal non-equilibrium ................................. 24
2.5 Percent difference between NS1T and NS2T simulations in various flow properties at various Knudsen numbers ........................................... 29

3.1 Free stream parameters for bicone simulations ..................................... 34
3.2 Bicone results with thermal equilibrium .............................................. 47
3.3 Bicone results with thermal non-equilibrium ....................................... 47
3.4 Percent difference in various flow properties between NS1T and NS2T simulations at various Knudsen numbers ........................................... 48

4.1 Computational resources for blunt body optimization .......................... 55
4.2 Blunt body optimization: improvements in objective values .................. 58

5.1 Computational resources for bicone optimization ............................... 69
5.2 Bicone optimization: improvements in objective values ....................... 78
List of Figures

1.1 Flow regimes and appropriate equations for modeling the flow at a given Knudsen number [1] ............................................................... 7
2.1 Geometry of the blunt body .......................................................... 10
2.2 Mesh around the blunt body .......................................................... 11
2.3 Details of the segmented mesh around the blunt body ...................... 12
2.4 Mach contours for Mach 7.1 flow about an axisymmetric blunt body .... 14
2.5 Pressure contours for Mach 7.1 flow about an axisymmetric blunt body ... 15
2.6 Translational temperature contours for Mach 7.1 flow about an axisymmetric blunt body .............................................................. 16
2.7 Translational and rotational temperature contours for Mach 7.1 flow about an axisymmetric blunt body with non-equilibrium temperature model .... 18
2.8 Translational and rotational temperatures along the stagnation line for Mach 7.1 flow about an axisymmetric blunt body in thermal non-equilibrium ... 19
2.9 Comparison of pressure coefficients on the surface of an axisymmetric blunt body in Mach 7.1 flow in equilibrium and thermal non-equilibrium .... 20
2.10 Comparison of heat transfer on the surface of an axisymmetric blunt body in Mach 7.1 flow in equilibrium and thermal non-equilibrium .......... 21
2.11 Comparison of skin friction $\tau (\frac{N}{m^2})$ on the surface of an axisymmetric blunt body in Mach 7.1 flow in equilibrium and thermal non-equilibrium ........ 22
2.12 Variation of maximum heat transfer with Knudsen number for an axisymmetric blunt body in Mach 7.1 flow in equilibrium (NS1T) and thermal non-equilibrium (NS2T) ........................................... 25
2.13 Variation of drag coefficient with Knudsen number for an axisymmetric blunt body in Mach 7.1 flow in equilibrium (NS1T) and thermal non-equilibrium (NS2T) .................................................. 26
2.14 Variation of shock standoff distance with Knudsen number for an axisymmetric blunt body in Mach 7.1 flow in equilibrium (NS1T) and thermal non-equilibrium (NS2T) .................................................. 27
2.15 Variation of translational and rotational temperature with Knudsen number for an axisymmetric blunt body in Mach 7.1 flow in thermal non-equilibrium 28
3.1 Geometry of the bicone - measurements (mm) ............................... 31
3.2 Mesh around the bicone .............................................................. 33
3.3 Mach contours for Mach 15.6 flow about an axisymmetric bicone ........ 36
3.4 Pressure contours for Mach 15.6 flow about an axisymmetric bicone ........ 37
3.5 Translational temperature contours for Mach 15.6 flow about an axisymmetric bicone ......................................................... 38
3.6 Translational and rotational temperature contours for Mach 15.6 flow about an axisymmetric bicone using the thermal non-equilibrium model (NS2T) ... 39
3.7 Comparison of pressure coefficients on the surface of an axisymmetric bicone in Mach 15.6 flow in equilibrium and thermal non-equilibrium ............ 41
3.8 Comparison of heating coefficient on the surface of an axisymmetric bicone in Mach 15.6 flow in equilibrium and thermal non-equilibrium ............ 42
3.9 Comparison of translational and rotational temperature on the surface of an axisymmetric bicone in Mach 15.6 flow in equilibrium and thermal non-equilibrium ................................................................. 43
3.10 Comparison of skin friction on the surface of an axisymmetric bicone in Mach 15.6 flow in equilibrium and thermal non-equilibrium ................. 44
3.11 Variation of drag coefficient with Knudsen number for a bicone in Mach 15.6 flow in equilibrium (NS1T) and thermal non-equilibrium (NS2T) ........ 45
3.12 Variation of heat transfer with Knudsen number for a bicone in Mach 15.6 flow in equilibrium (NS1T) and thermal non-equilibrium (NS2T) ........ 46
3.13 Variation of maximum translational and rotational temperatures with Knudsen number for a bicone in Mach 15.6 flow in thermal non-equilibrium (NS2T) 47
4.1 Schematic of optimization process ........................................... 50
4.2 Randomly generated bicone curves using Bezier control points ............. 51
4.3 Flowchart of Genetic Algorithm process .................................... 52
4.4 Randomly generated Bezier curve with eight control points for blunt body optimization .............................................................. 54
4.5 Optimized blunt body shape at Kn = 0.01 using NS2T model ............... 56
4.6 Optimized blunt body shape (blue) at Kn = 0.01 using NS2T model compared to that in Ref. [3] (red) for Kn = 0 using NS1T model ................. 57
4.7 Optimized blunt body Mach contours compared to the original blunt body . 59
4.8 Optimized blunt body pressure contours compared to the original blunt body 59
4.9 Optimized blunt body translational temperature contours compared to the original blunt body .......................................................... 60
4.10 Optimized blunt body rotational temperature compared to the original blunt body ............................................................... 61
4.11 Translational and rotational temperatures along the stagnation line of optimized blunt body compared to the original blunt body ..................... 62
4.12 Heating coefficient of optimized blunt body geometry compared to the original blunt body ......................................................... 63
4.13 Pressure coefficient of optimized blunt body compared to the original blunt body ............................................................... 64
4.14 Skin friction of optimized blunt body compared to the original blunt body . 65
5.1 Randomly generated bicone individual with two Bezier curves for bicone optimization ......................................................... 68
5.2 Optimized bicone shape at Kn = 0.0013 using NS2T model ........... 70
5.3 Optimized bicone Mach contours compared to the original bicone geometry ................................................................. 72
5.4 Optimized bicone pressure contours compared to the original bicone geometry ............................................................... 73
5.5 Optimized bicone translational temperature contours compared to the original bicone geometry ........................................ 74
5.6 Optimized bicone rotational temperature contours compared to the original bicone geometry ........................................... 75
5.7 Translational and rotational temperatures along the surface of the optimized bicone geometry compared to original geometry .......... 76
5.8 Heating coefficient along the surface of the optimized bicone geometry compared to original geometry ............................... 76
5.9 Pressure coefficient along the surface of the optimized bicone geometry compared to original geometry ............................. 77
5.10 Skin friction along the surface of the optimized bicone geometry compared to original geometry .......................................... 77
Acknowledgments

I would first like to thank my advisor, Dr. Ramesh K. Agarwal. His invaluable experience, patience and guidance were essential for the completion of this work. Without his assistance I would not have been able to accomplish anything in the field of CFD.

I would also like to extend my thanks to Dr. Wenwen Zhao for his work developing ZLOW and his patience while I learned to use it and Chris Seager for his time and energy spent developing and teaching me about genetic algorithms.

A special thanks is extended to the NASA Space Grant Consortium for their funding and the Washington University in St. Louis CFD Lab for providing the equipment and software necessary for this research.

Last, but not least, I would like to thank my family for their unwavering support and motivating me to pursue my dreams. I owe all of my successes to them.

Samuel Taylor Gardner

Washington University in Saint Louis
December 2016

"You miss 100% of the shots you don’t take," Wayne Gretzky
- Michael Scott
Dedicated to my parents, Britt and Maribeth Gardner
ABSTRACT OF THE THESIS

The Effects of Rarefaction and Thermal Non-Equilibrium on a Blunt Body and a Bicone in Hypersonic Flow and Their Shape Optimization for Reducing Both Drag and Heat Transfer

by

Samuel Taylor Gardner

Master of Science in Aerospace Engineering

Washington University in St. Louis, December 2016

Research Advisor: Ramesh Agarwal

Design of space vehicles pose many challenging problems due to their hypersonic speeds since they need to travel through different flow regimes due to changes in the density of the atmosphere with altitude. Some of the key characteristics associated with hypersonic flow are extremely high temperatures and heat transfer to the wall of the spacecraft. At these temperatures, the assumption of thermal equilibrium is no longer valid and the effect of rotational non-equilibrium must be included in the modeling of diatomic gas flow. This thesis employs the Navier-Stokes equations, which are modified to include a rotational non-equilibrium relaxation model to analyze the heat transfer, drag, and shock standoff distance for hypersonic flow past an axisymmetric blunt body and a bicone for various levels of rarefaction – including the rotational non-equilibrium effect. The customized flow solver, ZLOW, is used to calculate the numerical solutions for laminar viscous hypersonic flow past a blunt body and a bicone at Knudsen numbers Kn in continuum-transition regime with and without rotational non-equilibrium. The effects of rarefaction in the continuum-transition
regime are modeled by applying the Maxwellian velocity slip and temperature jump boundary conditions on the surface. The effects of the rotational non-equilibrium terms are discussed in this thesis for both the continuum (Kn ≈ 0) and slip flow regime (Kn ≤ 0.1). In addition, both the blunt body and bicone are optimized in hypersonic, rarefied flow with rotational non-equilibrium by using a multi-objective genetic algorithm (MOGA) for reduction of both drag and heat transfer.
Chapter 1

Introduction

1.1 Motivation

During the first stage of atmospheric reentry, vehicles undergo high heat loads and increased drag due to the high velocity and temperature. At these altitudes, the flow can no longer be considered to be in the continuum regime due to the effects of reduced atmospheric density, referred to as rarefaction. Many commercial flow solvers are not able to accurately predict the flow outside the continuum regime, which makes design and optimization of space vehicle geometries under these conditions challenging. Simulating these flow conditions is difficult; however, creating a flow solver capable of accurately predicting these flows is much more cost effective compared to wind tunnel testing in an actual hypersonic environment.

Rarefaction is characterized by the Knudsen number, a non-dimensional parameter given by:

\[ Kn = \frac{\lambda}{L} \]  

(1.1)

where \( \lambda \) is the mean free path of the molecules in the free stream and \( L \) is the characteristic length of the geometry.

The flow is considered to be within the continuum regime when the Knudsen number is very small, less than 0.01. An accurate solution can be calculated in the continuum regime using the Navier-Stokes (NS) equations without slip conditions. These assumptions and equations break down as the rarefaction effects increase. It is possible to modify the NS equations in
order to obtain accurate results into the continuum-transition slip regime, which is defined as $0.01 < Kn < 0.1$. This is done by implementing the Maxwellian Slip wall boundary conditions.

Another significant effect that needs to be considered when hypersonic vehicles re-enter the atmosphere is the effect of a rotational non-equilibrium.

In this study, an in-house FORTRAN based flow solver, ZLOW, is used to implement the Maxwellian slip boundary conditions as well as the rotational thermal non-equilibrium for the calculation of rarefied flow fields. The effects of a two-temperature thermal non-equilibrium model are analyzed for two geometries, an axisymmetric blunt body and bicone, at various Knudsen numbers. Each geometry was then optimized for the reduction of heat transfer and drag at a chosen Knudsen number with the two temperature model.

1.2 Background

1.2.1 Hypersonic Flow Characteristics

Hypersonic flow has the following properties that must be considered:

- Shock stand off distance
- Real gas effects
- Entropy change across shock
- Low density effects
- Thermal non-equilibrium
- Molecular dissociation

The density of a gas behind a shock wave increases as Mach number increases, which causes the shock-wave standoff distance to decrease with Mach number. The decreased standoff
distance coupled with high temperatures due to the transfer of kinetic energy to internal energy in the fluid from viscous effects cause hypersonic bodies to experience large changes in heat transfer. The large heat transfer can be reduced by increasing the radius of the leading edge of a blunt body; however, as the leading edge radius increases, the drag on the body also increases, making the reduction both drag and heat transfer difficult by shape optimization of the body.

The high temperatures associated with hypersonic flows can also create thermal non-equilibrium in rotational and vibrational modes. For a diatomic gas such as nitrogen, the vibrational modes are activated around 3300 K and become significant at temperatures above 5000 K. The rotational energy modes for a diatomic gas are activated at 3 K.

### 1.2.2 Navier-Stokes Equations Including Rotational Non-equilibrium

The 3D Navier-Stokes equation for a diatomic gas in rotational non-equilibrium in conservation law form in Cartesian coordinates can be written as:

\[
\frac{\partial \mathbf{Q}}{\partial t} + \frac{\partial \mathbf{E}}{\partial x} + \frac{\partial \mathbf{F}}{\partial y} + \frac{\partial \mathbf{G}}{\partial z} + \frac{\partial \mathbf{E}_v}{\partial x} + \frac{\partial \mathbf{F}_v}{\partial y} + \frac{\partial \mathbf{G}_v}{\partial z} = \mathbf{S} \tag{1.2}
\]

where

\[
\begin{align*}
\mathbf{E} &= \begin{bmatrix}
\rho u \\
\rho u^2 + p \\
\rho uv \\
\rho w \\
(\rho E + p)u \\
\rho e, u
\end{bmatrix}, & \quad \mathbf{F} &= \begin{bmatrix}
\rho v \\
\rho v^2 + p \\
\rho vw \\
\rho w \\
(\rho E + p)v \\
\rho e, v
\end{bmatrix}, & \quad \mathbf{G} &= \begin{bmatrix}
\rho w \\
\rho uw \\
\rho vw \\
\rho w^2 + p \\
(\rho E + p)w \\
\rho e, w
\end{bmatrix},
\end{align*}
\]

\[
\begin{align*}
\mathbf{E}_v &= \begin{bmatrix}
0 \\
\tau_{xx} \\
\tau_{yx} \\
\tau_{zx} \\
q_{r tran}^x + q_r^x + u_i \tau_{xi} \\
q_r^x
\end{bmatrix}, & \quad \mathbf{F}_v &= \begin{bmatrix}
0 \\
\tau_{xy} \\
\tau_{yy} \\
\tau_{zy} \\
q_{r tran}^y + q_r^y + u_i \tau_{yj} \\
q_r^y
\end{bmatrix}, & \quad \mathbf{G}_v &= \begin{bmatrix}
0 \\
\tau_{xz} \\
\tau_{yz} \\
\tau_{zz} \\
q_{r tran}^z + q_r^z + u_i \tau_{zk} \\
q_r^z
\end{bmatrix}
\end{align*}
\]
\[
Q = \begin{bmatrix}
\rho \\
\rho u \\
\rho v \\
\rho w \\
\rho E \\
\rho e_r
\end{bmatrix},
S = \begin{bmatrix}
0 \\
0 \\
0 \\
0 \\
0 \\
\frac{\rho R}{Z R_{\tau}} (T_l - T_r)
\end{bmatrix}
\] (1.3)

In Equation 1.3, \(\rho\) is the mass density; \(u, v,\) and \(w\) are the bulk velocity components in the \(x, y,\) and \(z\) directions; \(e\) is the total energy, and \(e_r\) is the rotational energy per unit volume of the gas.

\[e_{\text{tran}} = \frac{2}{3} R T_l, \quad e_r = R T_r, \quad k_{\text{tran}} = \frac{15}{4} \mu R, \quad k_r = \mu R\] (1.4)

The stress tensor, \(\tau_{ij}\), and the heat-flux vector, \(q_{ij}\), are obtained from the Chapman-Enskog expansion.

\[
\tau_{ij} = \tau_{ij}^{(0)} + \tau_{ij}^{(1)} + \tau_{ij}^{(2)} + \cdots + \tau_{ij}^{(n)} + O(Kn^{n+1})
\] (1.5)

\[
q_{ij} = q_{ij}^{(0)} + q_{ij}^{(1)} + q_{ij}^{(2)} + \cdots + q_{ij}^{(n)} + O(Kn^{n+1})
\] (1.6)

The Navier-Stokes equations are obtained by taking the first two terms of the expansions, the zeroth order and first order terms, in the stress tensor and heat-flux vector expansions. These become:

\[
\tau_{ij} = \tau_{ij}^{(0)} + \tau_{ij}^{(1)} = -2\mu \frac{\partial u_i}{\partial x_i}
\] (1.7)

where \(\frac{\partial u_i}{\partial x_i}\) is the none-divergent symmetric tensor given in Equation 1.8.
\[ a_{ij} = \frac{1}{2}(a_{ij} + a_{ji}) - \frac{1}{3} \delta_{ij} a_{kk} \]  

(1.8)

\[ q_{ij} = q_{ij}^{(0)} + q_{ij}^{(1)} = -k \frac{\partial T}{\partial x_i} \]  

(1.9)

Parker’s formulas[6] are used to obtain the relaxation time (\( \tau \)) and average number of collisions (\( Z_R \)) in the energy exchange that occurs between the translational and rotational energies.

\[ \tau = \frac{\mu}{p} \]  

(1.10)

\[ Z_R = \frac{Z_R^\infty}{1 + \frac{3}{2} \left( \frac{T_{ref}}{T} \right)^{1/2} + \left( \frac{\pi^2}{4} + \pi \right) \left( \frac{T_{ref}}{T} \right)} \]  

(1.11)

with constants \( Z_R^\infty = 15.7 \) and \( T_{ref} = 80 \) K.

The local viscosity of the flow is calculated using the Sutherland’s law:

\[ \mu = \mu_{ref} \left( \frac{T}{T_{ref}} \right)^{3/2} \frac{T_{ref} + S}{T + S} \]  

(1.12)

where \( \mu_{ref} \) and \( T_{ref} \) are the reference viscosity and temperature; \( S \) is the Sutherland temperature of the gas; and \( T \) is the local temperature.

In order to close the system of equations an equation of state is used, in this case the ideal gas law.
1.2.3 Rarefaction Effects

As space vehicles travel through the atmosphere and into the lower earth orbit, the density of the atmosphere decreases. This reduced density is called the rarefaction. The degree of rarefaction is characterized by the Knudsen number:

\[ Kn = \frac{\lambda}{L} \] (1.13)

where \( \lambda \) is the mean free path of the flow and \( L \) is the characteristic length associated with the geometry. The mean free path is the average distance traveled by a particle in the flow between collisions. In general, the mean free path of a molecule is given by Equation 1.14.

\[ \lambda = \frac{RT}{\sqrt{2\pi d^2 N_A P}} \] (1.14)

where \( R \) is the universal gas constant, \( T \) is the free stream temperature, \( d \) is the average molecule diameter, \( N_A \) is Avogadro’s number and \( P \) is the free stream pressure.

In the case of standard air, Equation 1.14 can be simplified to Equation 1.15.

\[ \lambda = \frac{\mu \sqrt{\pi}}{\rho \sqrt{2RT}} \] (1.15)

where \( \rho \) is the free stream density and \( \mu \) is the viscosity.

It can be seen in Equation 1.15 that as the free stream density decreases, the mean free path increases thus increasing the effect of rarefaction. At extremely low Knudsen numbers (Kn \( \leq 0.01 \)), the flow is in the continuum regime. Some equations used to model flow, such as the Euler and Navier-Stokes equations, are valid only in the continuum regime. As the Knudsen number increases, flow becomes more rarefied and enters the continuum-transition regime (Kn \( \leq 10 \)) and the Navier-Stokes equations are no longer able to accurately predict the flow. The validity of Navier-Stokes equations can be extended to Kn \( \leq 0.1 \) by using
Maxwell’s slip flow boundary conditions\cite{5} on the wall. The mathematical equations valid for each flow regime are shown in Figure 1.1.

![Figure 1.1: Flow regimes and appropriate equations for modeling the flow at a given Knudsen number \cite{1}](image)

### 1.2.4 Maxwell’s Slip Flow Boundary Conditions

The slip wall boundary conditions proposed by Maxwell consist of a velocity slip and a temperature jump condition. In the continuum regime, it is assumed that the velocity on the wall is zero; however, as rarefaction increases, this is no longer a valid and accurate approximation. In the near continuum regime, the velocity slip on the wall should be accounted for. The velocity slip boundary condition is given in Equation 1.16.

\[
U_s - U_w = \left(\frac{2 - \sigma}{\sigma}\right) \lambda \frac{\partial u_x}{\partial n} \tag{1.16}
\]

where \(U_s\) is the velocity of the flow, \(U_w\) is the velocity at the wall, \(\sigma\) is the momentum accommodation coefficient, \(\lambda\) is the local mean free path, and \(\frac{\partial u_x}{\partial n}\) is the velocity gradient normal to the wall.
The temperature jump boundary condition is given in Equation 1.17.

\[
T_0 - T_W = \left(\frac{2 - \alpha}{\alpha}\right) \frac{2\gamma}{(\gamma + 1)Pr} \lambda \frac{\partial T}{\partial n}
\]  

(1.17)

where \(T_0\) is temperature of the flow, \(T_W\) is the wall temperature, \(\alpha\) is the thermal accommodation coefficient, \(\gamma\) is the adiabatic index, and \(Pr\) is the Prandtl number.

Solving the Navier-Stokes equations with slip flow boundary conditions at the wall allows the flow to be accurately modeled for \(Kn \leq 0.1\).

### 1.3 ZLOW

The simulation results are obtained using ZLOW, a flow solver developed by Dr. Zhao Wen-wen of Zhejiang university. ZLOW is a 2D/3D, parallel, structured finite-volume numerical solver capable of simulating gas flow using both the Euler and Navier-stokes equations. The AUSMPW+ flux-splitting method is used, which is an improvement over the original advection upstream splitting method by employing pressure based weight functions[8]. The code employs an implicit time discretization method and a Lower-Upper Symmetric Gauss-Seidel (LU-SGS) scheme for solving the discretized equations[10]. Second order spatial accuracy is obtained by implementing the Monotonic Upstream-Centered Scheme for Conservation Laws (MUSCL) with the Van Albada limiter to handle the shock discontinuities present in the hypersonic flow[2]. The Landau-Teller-Jeans model is used to predict the non-equilibrium between the translational and rotational energies. In order to extend the validity of the Navier-Stokes equations to the continuum-transition regime (Knudsen number \(\leq 0.1\)), the first-order Maxwell’s slip wall boundary conditions are used.

ZLOW uses the structured grid format PLOT3D with double precision. ESI groups CFD-GEOM is used in this thesis to generate the meshes; the geometry is imported using a Python script. Mesh density is determined on a case-by-case basis to obtain the mesh independent solutions as discussed in later sections. The parallel processing in ZLOW is accomplished by a block decomposition scheme in mesh generation with each block assigned to a processor. Blocks are assigned according to the computational resources available.
Chapter 2

Simulation of an Axisymmetric Blunt Body Flow in Equilibrium Using a One Temperature Model (NS1T) and in Thermal Non-Equilibrium using a Two Temperature Model (NS2T)

In this chapter, the axisymmetric blunt body simulations in hypersonic flow are performed at four Knudsen numbers (0.002, 0.01, 0.05 and 0.1) with and without the effects of thermal non-equilibrium. The Maxwell’s slip boundary conditions for slip velocity and temperature jump at the wall are employed in the simulations. The shock standoff distance, the maximum heat transfer and the drag coefficient are obtained for each simulation.

2.1 Axisymmetric Blunt Body Parameters

2.1.1 Geometry

The geometry and free stream flow conditions used in the simulations correspond to the Lobb’s ballistic range experiments and are as given in Ref. [4]. The leading edge radius of the sphere is 6.35 mm. Since the body shape is axisymmetric, only a quarter of the sphere is
required for simulation of the flow field. The quarter geometry of the sphere used is shown in Figure 2.1. In the simulations, a cylindrical body shape is attached to the rear face of the sphere in Fig. 2.1 to exclude the wake effects.

### 2.1.2 Mesh Generation

A structured mesh around the geometry of Figure 2.1 with a cylindrical body attached is generated using CFD-GEOM. A slice of the final mesh is shown in Figure 2.2. This mesh was chosen based on an extensive mesh validation study conducted by Seager et. al [3], who employed several types of meshes of varying densities shown in Table 2.1 and concluded that the segmented mesh (35-105-10)x400 was the most efficient mesh with acceptable accuracy.

Figure 2.3 shows a detailed slice of the segmented (35-105-10)x400 mesh employed in this thesis for blunt body simulations. The structured mesh was broken into 5 even blocks to utilize the parallel processing capabilities of ZLOW and thus to decrease the total simulation time.
Table 2.1: Blunt body mesh validation study [3]

<table>
<thead>
<tr>
<th>Mesh Type</th>
<th>Mesh Size</th>
<th>Sphere</th>
<th>Random Body Shape</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>$C_D$ error</td>
<td>$HT$ error</td>
</tr>
<tr>
<td>Reference</td>
<td>631x1250</td>
<td>---------</td>
<td>---------</td>
</tr>
<tr>
<td>Uniform</td>
<td>80x200</td>
<td>0.1</td>
<td>29.3</td>
</tr>
<tr>
<td>Uniform</td>
<td>120x200</td>
<td>0.1</td>
<td>18.6</td>
</tr>
<tr>
<td>Uniform</td>
<td>160x400</td>
<td>0.1</td>
<td>9.3</td>
</tr>
<tr>
<td>Uniform</td>
<td>200x400</td>
<td>0</td>
<td>8.8</td>
</tr>
<tr>
<td>Uniform</td>
<td>(390-10)x400</td>
<td>0</td>
<td>4.0</td>
</tr>
<tr>
<td>Uniform</td>
<td>(85-115)x400</td>
<td>0</td>
<td>8.3</td>
</tr>
<tr>
<td>Segmented</td>
<td>(44-146-10)x400</td>
<td>0</td>
<td>7.3</td>
</tr>
<tr>
<td>Segmented</td>
<td>(25-75-10)x400</td>
<td>0</td>
<td>8.3</td>
</tr>
<tr>
<td>Mesh of Choice</td>
<td>(35-105-10)x400</td>
<td>0</td>
<td>7.5</td>
</tr>
</tbody>
</table>
2.1.3 Free Stream Flow Conditions

The free stream flow conditions are obtained from Lobb’s experiment and are given in Ref. [4]. The free stream Mach number is 7.1, the temperature is 293 K and the air species distribution is 23.3 percent $O_2$ and 76.7 percent $N_2$. The wall is kept at a constant temperature of 1000 K. Since the species distribution is that of standard air, the mean free path given by Equation 1.15 is valid for calculating the Knudsen number. The mean free path is calculated from the Knudsen number and the characteristic length of the body based on the diameter of 0.0127 m. The free stream density is calculated using Equation 1.15 and the free stream pressure is obtained using the ideal gas law. The free stream pressures required to get the desired Knudsen numbers are given in Table 2.2.
Table 2.2: Free stream parameters for blunt body simulations

<table>
<thead>
<tr>
<th>Knudsen Number</th>
<th>Pressure (Pa)</th>
<th>Mean Free Path (m)</th>
<th>Characteristic Length</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.002</td>
<td>245.595</td>
<td>2.54E-5</td>
<td>.0127</td>
</tr>
<tr>
<td>0.01</td>
<td>49.854</td>
<td>1.27E-4</td>
<td>.0127</td>
</tr>
<tr>
<td>0.05</td>
<td>9.824</td>
<td>6.35E-4</td>
<td>.0127</td>
</tr>
<tr>
<td>0.1</td>
<td>4.912</td>
<td>1.27E-3</td>
<td>.0127</td>
</tr>
</tbody>
</table>

2.2 Computed Results for Axisymmetric Blunt Body in Equilibrium (NS1T) and Non-Equilibrium (NS2T)

2.2.1 Flow Field Properties

The results for the flow properties of the axisymmetric blunt body using Navier-Stokes equations with NS1T and NS2T model are described below. In all the contour plots shown in this chapter, NS1T simulation results are shown on the top half of the plot and the NS2T simulations results are shown on the bottom half. The contours are scaled to the highest value between the NS1T and NS2T simulations; 24 discrete values are used.

Figure 2.4 shows the comparison of NS1T and NS2T Mach contours for four Knudsen numbers. At low Knudsen numbers, the shock standoff distance predicted by both the NS1T and NS2T models is very close to each other as expected. But as the rarefaction increases, the NS2T model begins to predict a more diffuse shock with a much larger standoff distance. The difference in shock standoff distance computed with NS1T and NS2T models increases with Knudsen number, with the NS2T model always predicting a larger shock standoff distance.

Figure 2.5 shows the pressure contours for both the NS1T and NS2T models at four Knudsen numbers. The pressure contours show much less sensitivity to the effects of non-equilibrium compared to the Mach contours. The NS2T model pressure contours exhibit the same shock standoff distance that was shown in Figure 2.4; however, the difference in magnitude of the high pressure region at the standoff point is very small. Notably, the NS2T model predicts
Figure 2.4: Mach contours for Mach 7.1 flow about an axisymmetric blunt body

(a) $Kn = 0.002$
(b) $Kn = 0.01$
(c) $Kn = 0.05$
(d) $Kn = 0.1$
Figure 2.5: Pressure contours for Mach 7.1 flow about an axisymmetric blunt body

(a) Kn = 0.002
(b) Kn = 0.01
(c) Kn = 0.05
(d) Kn = 0.1

a smaller high pressure region in the shock standoff region than the NS1T model. There is no significant difference between the NS1T and NS2T models for pressure in the far field. In all cases the pressure ratio across the shock is approximately 64, which is higher than the ratio of 59 predicted by the inviscid normal shock relations.

As was the case with the Mach and pressure contours, Figure 2.6 shows no significant difference between the NS1T and NS2T results for Knudsen numbers within the continuum regime. As rarefaction increases, the NS2T model predicts an increasingly higher temperature in the post-shock region than the NS1T model. This temperature increase helps explain the increase in predicted shock standoff distance by the NS2T model.
Figure 2.6: Translational temperature contours for Mach 7.1 flow about an axisymmetric blunt body

(a) $\text{Kn} = 0.002$

(b) $\text{Kn} = 0.01$

(c) $\text{Kn} = 0.05$

(d) $\text{Kn} = 0.1$
Figure 2.7 shows the temperature contours of the translational and rotational temperatures predicted by the NS2T model. For each Knudsen number simulation the translational temperature is shown on the top and the rotational temperature is shown on the bottom part of the figure. It is clear that the shock provides the biggest source of non-equilibrium between the translational and rotational temperatures. This is shown further in Figure 2.8 where the translational and rotational temperatures are plotted along the stagnation line.

In case of continuum regime, Kn = 0.002, there are hardly any non-equilibrium effects except in the shock region. Within the shock region, the translational temperature has a slightly higher peak temperature before reaching quickly to the equilibrium value after the shock. As the Knudsen number (rarefaction) increases, the magnitude of the translational/rotational non-equilibrium increases. In case of Kn = 0.01 the temperatures do not reach the equilibrium until close to the boundary layer region. In case of Kn = 0.05 and Kn = 0.1 the temperatures do not reach equilibrium until they hit the wall, which by definition is at a constant temperature. These cases correspond to the more diffused shock with a larger standoff distance. The non-equilibrium effect is imparted by the shock and the flow relies on collisions to equilibrate. As rarefaction increases the density, and therefore the number of collisions decreases. This increases the duration of the thermal non-equilibrium with increase in flow rarefaction.

### 2.2.2 Surface Properties

The surface properties of the blunt body in hypersonic rarefied air are described in this section. Various flow parameters are plotted along the leading edge and the aft-cylinder of the blunt body to analyze the effects of non-equilibrium. The surface pressure predictions in Figure 2.9 show that the pressure coefficients have little dependence on rarefaction or thermal non-equilibrium. As rarefaction increases the NS2T model predicts a slightly lower pressure coefficient than the NS1T model, particularly in the leading edge region of the blunt body.

The heating coefficient, $C_H$, is most sensitive to the effects of non-equilibrium along the leading edge. Figure 2.10 shows that as rarefaction increases, $C_H$ decreases and the difference in $C_H$ between the NS1T and NS2T models increases.
Figure 2.7: Translational and rotational temperature contours for Mach 7.1 flow about an axisymmetric blunt body with non-equilibrium temperature model.
Figure 2.8: Translational and rotational temperatures along the stagnation line for Mach 7.1 flow about an axisymmetric blunt body in thermal non-equilibrium
Figure 2.9: Comparison of pressure coefficients on the surface of an axisymmetric blunt body in Mach 7.1 flow in equilibrium and thermal non-equilibrium

The skin friction, $\tau$, is the only parameter that shows the effect of non-equilibrium after the leading edge of the blunt body. There is excellent agreement between the NS1T and NS2T models along the leading edge; however, as rarefaction increases the NS2T model predicts lower skin friction along the aft-cylinder of the blunt body, as shown in Figure 2.11.
Figure 2.10: Comparison of heat transfer on the surface of an axisymmetric blunt body in Mach 7.1 flow in equilibrium and thermal non-equilibrium.
Figure 2.11: Comparison of skin friction $\tau \left( \frac{N}{m^2} \right)$ on the surface of an axisymmetric blunt body in Mach 7.1 flow in equilibrium and thermal non-equilibrium
2.3 Axisymmetric Blunt Body in Equilibrium (NS1T) vs Non-Equilibrium (NS2T): Discussion of Results

Comparisons of results for the NS1T and NS2T models for the hypersonic rarefied flow of air about an axisymmetric blunt body show that translational/rotational non-equilibrium has a significant effect on the shock standoff distance and maximum heat transfer as rarefaction increases. Tables 2.3 and 2.4 present the results for $C_d$, Drag, maximum heat transfer and shock standoff distance at all Knudsen numbers considered. In the two cases of low Knudsen number, where the flow is primarily in the continuum regime, all computed flow properties are in close agreement as expected; however, as the Knudsen number increases the flow properties begin to change. Figure 2.12 shows the maximum heat transfer for each simulation. As rarefaction increases the simulations predict a large decrease in maximum heat transfer. Figure 2.13 shows that the drag coefficient increases with rarefaction. The behavior of both the drag coefficient and heating coefficient can be explained by their dependence on the free-stream density, $\rho_\infty$. The density decreases faster than the dynamic pressure, $q$ and $D$, which results in value of the drag coefficient.

The shock standoff distance is computed from the stagnation point and plotted as a function of the Knudsen number in Figure 2.14. This plot confirms what was seen in the Mach contours in Figure 2.4 where the NS2T model predicted a larger standoff distance than the NS1T model. In case of $Kn = 0.002$ the shock standoff distance is predicted to be 5.92% larger in the NS2T case compared to NS1T case. This difference increases to 33.82% difference at $Kn = 0.1$.

Table 2.5 shows the relative effect of translational/rotational thermal non-equilibrium as percentage difference between the NS1T and NS2T values given by Equation 2.1:

$$Percent \ Difference = 100 \times \frac{NS1T - NS2T}{NS1T} \quad (2.1)$$

where NS1T corresponds to the value of a flow property of interest obtained by using the NS1T model and NS2T is the value of the flow property of interest obtained using the NS2T model.
This table shows that the NS2T model predicts a lower drag and maximum value of heat transfer in all cases considered. The predicted drag at \( Kn = 0.002 \) is in close agreement between the NS1T and NS2T models as expected. In the slip regime also, the predicted drag by the two models is still in close agreement with the largest difference being at \( Kn = 0.05 \) with a 4.61% difference.

Heat transfer is most sensitive to the effect of non-equilibrium as Knudsen number increases. At \( Kn = 0.002 \) the maximum heat transfer is 2.29% lower using the NS2T model compared to NS1T model, while it is lower by 7.63% at \( Kn = 0.1 \).

The largest difference between the results of NS1T and NS2T models is in the shock standoff distance predictions. In the continuum regime (\( Kn = 0.002 \)) and at the lower end of the slip regime (\( Kn = 0.01 \)) the predicted standoff distances is approximately 8% greater for the NS2T model compared to the NS1T model. However, as the rarefaction increases the shock standoff distance using the NS2T model increases to 24.45% greater for \( Kn = 0.05 \) and 33.82% greater for \( Kn = 0.1 \) compared to that obtained using the NS1T model.

Table 2.3: Blunt body results with thermal equilibrium

<table>
<thead>
<tr>
<th>Kn</th>
<th>( P_{\infty} ) (Pa)</th>
<th>Density (( \frac{kg}{m^3} ))</th>
<th>Cd</th>
<th>Drag (N)</th>
<th>Qmax (( \frac{W}{m^2} ))</th>
<th>Standoff (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.002</td>
<td>245.595</td>
<td>2.92E-3</td>
<td>0.9622</td>
<td>1.0563</td>
<td>1.48E+06</td>
<td>8.67E-4</td>
</tr>
<tr>
<td>0.01</td>
<td>49.85</td>
<td>5.84E-4</td>
<td>1.161</td>
<td>0.2549</td>
<td>6.10E+05</td>
<td>1.004E-3</td>
</tr>
<tr>
<td>0.05</td>
<td>9.824</td>
<td>1.17E-4</td>
<td>1.517</td>
<td>0.06662</td>
<td>2.69E+05</td>
<td>1.244E-3</td>
</tr>
<tr>
<td>0.1</td>
<td>4.912</td>
<td>5.839E-05</td>
<td>1.722</td>
<td>0.037809</td>
<td>1.79E+05</td>
<td>1.39E-3</td>
</tr>
</tbody>
</table>

Table 2.4: Blunt body results with thermal non-equilibrium

<table>
<thead>
<tr>
<th>Kn</th>
<th>( P_{\infty} ) (Pa)</th>
<th>Density (( \frac{kg}{m^3} ))</th>
<th>Cd</th>
<th>Drag (N)</th>
<th>Qmax (( \frac{W}{m^2} ))</th>
<th>Standoff (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.002</td>
<td>245.595</td>
<td>2.9196E-3</td>
<td>0.9579</td>
<td>1.0516</td>
<td>1.44E+06</td>
<td>9.183E-4</td>
</tr>
<tr>
<td>0.01</td>
<td>49.85</td>
<td>5.839E-3</td>
<td>1.134</td>
<td>0.2494</td>
<td>5.96E+05</td>
<td>1.08E-3</td>
</tr>
<tr>
<td>0.05</td>
<td>9.824</td>
<td>1.168E-4</td>
<td>1.447</td>
<td>0.06354</td>
<td>2.59E+05</td>
<td>1.549E-3</td>
</tr>
<tr>
<td>0.1</td>
<td>4.912</td>
<td>5.839E-05</td>
<td>1.667</td>
<td>0.03660</td>
<td>1.65E+05</td>
<td>1.86E-3</td>
</tr>
</tbody>
</table>
Figure 2.12: Variation of maximum heat transfer with Knudsen number for an axisymmetric blunt body in Mach 7.1 flow in equilibrium (NS1T) and thermal non-equilibrium (NS2T)
Figure 2.13: Variation of drag coefficient with Knudsen number for an axisymmetric blunt body in Mach 7.1 flow in equilibrium (NS1T) and thermal non-equilibrium (NS2T)
Figure 2.14: Variation of shock standoff distance with Knudsen number for an axisymmetric blunt body in Mach 7.1 flow in equilibrium (NS1T) and thermal non-equilibrium (NS2T)
Figure 2.15: Variation of translational and rotational temperature with Knudsen number for an axisymmetric blunt body in Mach 7.1 flow in thermal non-equilibrium
Table 2.5: Percent difference between NS1T and NS2T simulations in various flow properties at various Knudsen numbers

<table>
<thead>
<tr>
<th>Knudsen Number</th>
<th>Drag</th>
<th>Qmax</th>
<th>Shock Standoff Distance</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.002</td>
<td>0.45</td>
<td>2.29</td>
<td>-5.92</td>
</tr>
<tr>
<td>0.01</td>
<td>2.15</td>
<td>2.33</td>
<td>-7.57</td>
</tr>
<tr>
<td>0.05</td>
<td>4.61</td>
<td>3.69</td>
<td>-24.45</td>
</tr>
<tr>
<td>0.1</td>
<td>3.19</td>
<td>7.63</td>
<td>-33.82</td>
</tr>
</tbody>
</table>
Chapter 3

Simulation of an Axisymmetric Bicone Flow in Equilibrium Using a One Temperature Model (NS1T) and in Thermal Non-Equilibrium using a Two Temperature Model (NS2T)

In this chapter the axisymmetric bicone simulations in hypersonic flow are performed at four Knudsen numbers (0.0013, 0.01, 0.057 and 0.1) with and without the effects of thermal non-equilibrium. The Maxwell’s slip boundary conditions for slip velocity and temperature jump at the wall are employed in the simulations. A sharp leading edge contrasts from the blunt body in that the shock stays attached to the sharp leading edge of the geometry. The maximum heat transfer and the drag coefficient are obtained for each simulation.

3.1 Axisymmetric Bicone Parameters

3.1.1 Geometry

The geometry for the bicone correspond to the experiments done by Harvey et. al [7]. The leading edge of the bicone is 0.09208 m in the x-direction at an angle of 25° from the...
horizontal. The aft-cone is .0616 m in the x-direction at 55° from the horizontal. Since the body shape is axisymmetric, only a quarter of the sphere is required for the simulation of the flow field. The quarter geometry of the bicone used is shown in Figure 3.1.

Figure 3.1: Geometry of the bicone - measurements (mm)
3.1.2 Mesh Generation

A structured mesh around the bicone geometry of Figure 3.1 is generated using ESI groups CFD-GEOM. The mesh was created based on an extensive mesh validation study performed in the literature [11]. A detailed slice of the axisymmetric bicone mesh can be seen in Figure 3.2. It was found that the direction normal to the wall was most sensitive so 80 cells were placed with an exponential distribution in order to properly resolve all flow conditions near the wall. The structured mesh was broken into 5 blocks in order to take advantage of the parallel processing capabilities of ZLOW and thus decrease total simulation time.
3.1.3 Free Stream Flow Conditions

The free stream flow conditions are obtained from the experiment given in [7]. The free stream Mach number is 15.6, the free stream temperature is 42.6 K, and the flow is pure nitrogen gas ($N_2$). Since the flow is diatomic nitrogen gas as opposed to standard air, the free stream density is calculated from the definition provided in Equation 1.14. The mean free path is calculated from the Knudsen number and the characteristic length of 0.09208 m.
The free stream pressure is then calculated from the density using the ideal gas law. The viscosity is calculated using Sutherland’s Law with a reference viscosity, \( \mu_{ref} = 2.662 \times 10^{-6} \frac{kg}{s \ m} \) and reference temperature 42.6 K. The bicone wall is kept at a constant temperature of 297 K. The free stream pressures required for each Knudsen number are given in Table 3.1.

Table 3.1: Free stream parameters for bicone simulations

<table>
<thead>
<tr>
<th>Knudsen Number</th>
<th>Pressure (Pa)</th>
<th>Mean Free Path (m)</th>
<th>Characteristic Length</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0013</td>
<td>2.227</td>
<td>1.20E-4</td>
<td>.09208</td>
</tr>
<tr>
<td>0.01</td>
<td>0.8148</td>
<td>9.21E-4</td>
<td>.09208</td>
</tr>
<tr>
<td>0.057</td>
<td>0.143</td>
<td>5.25E-3</td>
<td>.09208</td>
</tr>
<tr>
<td>0.1</td>
<td>0.07985</td>
<td>9.21E-3</td>
<td>.09208</td>
</tr>
</tbody>
</table>

**3.2 Computed Results for Axisymmetric Bicone in Equilibrium (NS1T) and Thermal Non-Equilibrium (NS2T)**

**3.2.1 Flow Field Properties**

The results for the flow properties of the axisymmetric bicone using Navier-Stokes equations with NS1T and NS2T models are described below. In all the contour plots shown in this chapter, the NS1T simulations results are shown on the top half of the plot and the NS2T simulations are shown on the bottom half. The contours are scaled to the highest value between the NS1T and NS2T simulations using 24 discrete values.

The Mach contour plots for the comparison of NS1T and NS2T cases are shown in Figure 3.3. Figure 3.3 shows that the shock remains attached on the fore cone, as expected from a sharp nosed hypersonic body. The shock detaches at the aft cone, creating a bow shock. Similarly to the blunt body, the shock become more diffused as Knudsen number increases. For all Knudsen numbers the shock is predicted to be more diffused for the NS2T case in comparison to the NS1T case.
Figure 3.4 shows the pressure contours for both the NS1T and NS2T models at four Knudsen numbers. The highest pressure region occurs where the fore and aft cones meet. The pressures are in close agreement when comparing the NS1T and NS2T cases for the four Knudsen numbers along the wall of the bicone in all cases but $Kn = 0.1$. For the case of $Kn = 0.1$ the NS2T model predicts a lower pressure where the cones meet when compared to the NS1T model.

Figure 3.5 shows the translational temperature contours for both the NS1T and NS2T models. Figure 3.5 shows that there are two high temperature regions of flow around the axisymmetric bicone. The leading point of the fore cone has the highest temperature for all cases; however, the post-shock region of the detached flow on the aft cone have temperatures that are in close agreement with the temperatures at the leading point of the bicone. The highest temperature of the post-shock region occurs at the detachment point, and decreases as the flow moves away from the bend. The effects of thermal non-equilibrium become more significant as Knudsen number increases. The NS2T model predicts a higher maximum translational temperature for all Knudsen numbers.

Figure 3.6 plots the translational and rotational temperature contours from the NS2T results with the translational temperature shown on the top part of the figure and rotational temperature shown on the bottom part of the figure. The rotational temperature remains at the free stream temperature of 42.6 K until the post-shock region. Within the continuum regime the temperatures remain in equilibrium throughout the flow, but as Knudsen number increases the thermal non-equilibrium increases, particularly along the fore cone and in the post-shock region.
(a) $Kn = 0.0013$

(b) $Kn = 0.01$

(c) $Kn = 0.057$

(d) $Kn = 0.1$

Figure 3.3: Mach contours for Mach 15.6 flow about an axisymmetric bicone
Figure 3.4: Pressure contours for Mach 15.6 flow about an axisymmetric bicone.
Figure 3.5: Translational temperature contours for Mach 15.6 flow about an axisymmetric bicone
Figure 3.6: Translational and rotational temperature contours for Mach 15.6 flow about an axisymmetric bicone using the thermal non-equilibrium model (NS2T)

3.2.2 Surface Properties

For each Knudsen number case the following surface properties of the axisymmetric bicone in hypersonic flow are examined: pressure, heat flux, translational/rotational temperature and shear stress. These parameters are plotted along the fore cone and aft cone of the bicone to analyze the effects of thermal non-equilibrium. Figure 3.7 shows in the case of low Knudsen numbers (Kn = 0.0013 and 0.01) the pressure coefficients have little dependence on thermal non-equilibrium when comparing the NS1T model and NS2T model. In case of Kn = 0.01 the NS1T model predicts a slightly lower pressure coefficient on the tip of the bicone and shortly after the high pressure post shock detachment region when compared to the NS2T model. At higher levels of rarefaction the pressure coefficients from the NS1T and...
NS2T simulations start to change. The peak pressure coefficient that occurs shortly after the fore-aft cone connection point in the post shock region is predicted to be much smaller by the NS2T model when compared to the NS1T model. The NS2T model also predicts a less rapid increase in pressure coefficient in the post-shock region. Notably there is a deviation in how the pressure is predicted on the fore cone between cases Kn = 0.057 and 0.1. In case Kn = 0.1 a higher pressure coefficient is predicted in all regions except the post shock spike region. Case Kn = 0.057 predicts a higher pressure coefficient at the leading point of the fore cone, but a lower pressure coefficient along the fore cone until the post shock region.

The heating coefficient is plotted for each of the four Knudsen cases in Figure 3.8. There is evidence of non-equilibrium even in the Kn = 0.0013 case. For the case within the continuum regime the heating coefficient is lower at every point along the bicone wall in the NS2T model when compared to the NS1T model. The case of Kn = 0.01 heating coefficient follows the same trend as Kn = 0.0013. The value of the heating coefficient is lower at every point along the wall for the NS2T model compared to the NS1T model in case Kn = 0.01. As rarefaction increases to Kn = 0.057 and 0.1 the heating coefficient is influenced more by the thermal non-equilibrium. The leading point of the bicone shows close agreement between the two models; however, along the wall the NS2T model predicts a lower heating coefficient than the NS1T model. The NS2T model also predicts a less steep increase in heating coefficient in the post shock region for both Kn = 0.057 and 0.1 cases.

How the thermal non-equilibrium is related to the heating coefficient is investigated in Figure 3.6 by plotting the temperature from the NS1T model against the translational and rotational temperatures from the NS2T model. The primary source of non-equilibrium is at the point of the fore cone and the temperatures converge as they approach the fore cone/aft cone juncture. As rarefaction increases, the magnitude of thermal non-equilibrium increases at the point of the fore cone.

Figure 3.10 shows the skin friction along the wall of the bicone. The skin friction behaves as expected for the fore cone in cases in or near the continuum regime, with a sharp spike at the leading point followed by an exponential decrease. The point where the shock becomes detached shows an unpredictable behavior with a sharp decrease in skin friction after the stagnation point. As the flow becomes more rarefied the spike at the point of the cone flips and becomes a zero point with a sharp increase. The decrease after the shock detachment
becomes much closer to a discontinuity. The NS2T model also predicts a lower amount of skin friction across all points in the first 80% of the bicone and a higher amount across the last 20% of the bicone when compared to the NS1T model for cases $Kn = 0.057$ and $Kn = 0.1$.

Figure 3.7: Comparison of pressure coefficients on the surface of an axisymmetric bicone in Mach 15.6 flow in equilibrium and thermal non-equilibrium

(a) $Kn = 0.0013$

(b) $Kn = 0.01$

(c) $Kn = 0.057$

(d) $Kn = 0.1$
Figure 3.8: Comparison of heating coefficient on the surface of an axisymmetric bicone in Mach 15.6 flow in equilibrium and thermal non-equilibrium
Figure 3.9: Comparison of translational and rotational temperature on the surface of an axisymmetric bicone in Mach 15.6 flow in equilibrium and thermal non-equilibrium.
Comparisons of the results for NS1T and NS2T models for the hypersonic rarefied flow of nitrogen about an axisymmetric bicone show that the translational/rotational non-equilibrium has a significant effect on the maximum values of heat transfer, with little effect on the overall amount of drag. The largest difference in value between the NS1T and NS2T models
occur at $Kn = 0.1$ with a percent difference of 1.71% for drag and 13.61% for maximum heat flux. The NS1T model predicted a lower drag in all cases but $Kn = 0.0013$, which was in agreement with the NS2T.

The maximum heat transfer showed a significant dependence on the thermal non-equilibrium as rarefaction increased. In case of both $Kn = 0.0013$ and 0.01 shows the heat transfer had small dependence on thermal non-equilibrium with the largest difference being 2.12%. There is a large spike in difference between the models as the rarefaction is increased to $Kn = 0.057$, with a 12.47% lower value for maximum heat flux when using the NS2T model and 13.61% lower for Knudsen number 0.1 with the NS2T model.

All results of objective values are listed in Tables 3.2 and 3.3 for NS1T and NS2T models respectively. The percentage difference values used to show the relative effect of translational/rotational thermal non-equilibrium given by Equation 2.1 are listed in Table 3.4.

Figure 3.11: Variation of drag coefficient with Knudsen number for a bicone in Mach 15.6 flow in equilibrium (NS1T) and thermal non-equilibrium (NS2T)
Figure 3.12: Variation of heat transfer with Knudsen number for a bicone in Mach 15.6 flow in equilibrium (NS1T) and thermal non-equilibrium (NS2T)
Figure 3.13: Variation of maximum translational and rotational temperatures with Knudsen number for a bicone in Mach 15.6 flow in thermal non-equilibrium (NS2T)

Table 3.2: Bicone results with thermal equilibrium

<table>
<thead>
<tr>
<th>Kn</th>
<th>$P_{\infty}$ (Pa)</th>
<th>Density ($\frac{kg}{m^3}$)</th>
<th>Cd</th>
<th>Drag (N)</th>
<th>$Q_{\text{max}}$ ($\frac{W}{m^2}$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0013</td>
<td>2.227</td>
<td>1.76E-4</td>
<td>2.209</td>
<td>30.942</td>
<td>1.28E5</td>
</tr>
<tr>
<td>0.01</td>
<td>0.815</td>
<td>6.45E-5</td>
<td>2.246</td>
<td>11.511</td>
<td>6.25E4</td>
</tr>
<tr>
<td>0.057</td>
<td>0.143</td>
<td>1.13E-5</td>
<td>2.321</td>
<td>2.0876</td>
<td>1.55E4</td>
</tr>
<tr>
<td>0.1</td>
<td>0.0799</td>
<td>6.32E-6</td>
<td>2.392</td>
<td>1.2014</td>
<td>9.44E3</td>
</tr>
</tbody>
</table>

Table 3.3: Bicone results with thermal non-equilibrium

<table>
<thead>
<tr>
<th>Kn</th>
<th>$P_{\infty}$ (Pa)</th>
<th>Density ($\frac{kg}{m^3}$)</th>
<th>Cd</th>
<th>Drag (N)</th>
<th>$Q_{\text{max}}$ ($\frac{W}{m^2}$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0013</td>
<td>2.227</td>
<td>1.76E-4</td>
<td>2.209</td>
<td>30.942</td>
<td>1.30E5</td>
</tr>
<tr>
<td>0.01</td>
<td>0.815</td>
<td>6.45E-5</td>
<td>2.236</td>
<td>11.460</td>
<td>6.25E4</td>
</tr>
<tr>
<td>0.057</td>
<td>0.143</td>
<td>1.13E-5</td>
<td>2.314</td>
<td>2.0813</td>
<td>1.36E4</td>
</tr>
<tr>
<td>0.1</td>
<td>0.0799</td>
<td>6.32E-6</td>
<td>2.351</td>
<td>1.1808</td>
<td>8.16E3</td>
</tr>
</tbody>
</table>
Table 3.4: Percent difference in various flow properties between NS1T and NS2T simulations at various Knudsen numbers

<table>
<thead>
<tr>
<th>Knudsen Number</th>
<th>Drag</th>
<th>Qmax</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0013</td>
<td>0</td>
<td>-2.12</td>
</tr>
<tr>
<td>0.01</td>
<td>0.445</td>
<td>0.077</td>
</tr>
<tr>
<td>0.057</td>
<td>0.302</td>
<td>12.47</td>
</tr>
<tr>
<td>0.1</td>
<td>1.71</td>
<td>13.61</td>
</tr>
</tbody>
</table>
Chapter 4

Shape Optimization of a Blunt Body for Reduction of Both Drag and Heat Transfer Using a Genetic Algorithm

Reducing both heat transfer and drag on a geometry in hypersonic flow is difficult since increasing the leading edge radius decreases the heat transfer and increases drag. To optimize geometries for reduction of maximum heat transfer and drag, a multi-objective genetic algorithm is implemented.

4.1 Genetic Algorithm Overview

Genetic algorithms are inspired by biological evolution where the strongest individuals survive and mate to produce more fit offspring based on the desired characteristics. Multi-objective genetic algorithms (MOGA) are more complex than the single-objective genetic algorithms because the fitness of each individual is determined by two or more objective values. These objective values are often in conflict with each other, where by improving a geometry for one objective has the opposite impact on the other objective/s. The unique optimized solution using a MOGA depends on relative weights assigned to each objective value. There is only one optimized solution for the MOGA for each set of relative weights and the set of these unique solutions gives a Pareto-optimal front. A genetic algorithm consists of the following five steps:
In the initialization step, a population of \( N \) random individuals is generated and simulated. Each individual is then evaluated and assigned a fitness based on the desired optimization characteristics. The steps that each individual follows are shown in Figure 4.1.

The shape of an individual is generated using Bezier curves. Bezier curves use of control points make them favorable for generating random shapes or curves for a genetic algorithm. They allow the creation of a smooth curve by using a finite number of control points. This drastically reduces the degrees of freedom in the geometry and reduces the number of iterations required for the algorithm to find the optimized shape. The control points are also easily switched or averaged between the individuals, making them easily adapted for the reproduction step of the genetic algorithm.

The initialization step chooses a random set of control points for each individual. Figure 4.2 shows an example of four randomly generated individuals for the bicone optimization. The bicone was modeled by joining two Bezier curves at a defined point. The red line is the leading edge of the bicone with the yellow circles representing the Bezier control points for that line. The green line is the second curve of the bicone and the blue circles represent its control points.
These control points are then parameterized and loaded into a python script that generates meshes in CFD-GEOM. Once the geometry is properly meshed, the individual is simulated in ZLOW. After the simulation achieves convergence, the values of interest are saved along with the Bezier control points for evaluation. These steps are repeated for each individual in each generation. Once the last individual in a generation has been simulated in ZLOW, the evaluation and natural selection steps in MOGA are executed.

The evaluation step analyzes the objective values of each individual and assigns it a ranking relative to the other individuals in its generation. This is done by sorting the individuals from best to worst and recording its position in the sorting. This is done for each optimization parameter. The fitness of each individual is determined by the weight functions assigned in MOGA. In the case of evenly weighted objective functions in MOGA, the fitness is defined by the sum of its rankings. After each individual has been evaluated and assigned a fitness value, the natural selection occurs. In the natural selection step, the individuals are sorted by their fitness values and the top half of the generation are selected as "survivors", while the bottom half are discarded.
The survivors are then given a chance to influence the next generation of individuals by a reproduction method. Two survivors are chosen using a skewed random draw and undergo crossover. In this step, the individuals share Bezier control point information. There are two possibilities for information exchange: a complete swap of Bezier points or a weighted average of points. The method of information exchange is chosen at random. This process is repeated until a new generation of individuals has been populated. Before the new generation is meshed and simulated, it must also go through a mutation stage. This step chooses a random number of curves from the new generation and randomly alters the points of the chosen individuals in order to increase the diversity of the new generation.

The evaluation, reproduction, and mutation steps are repeated until convergence. The MOGA checks for convergence by checking the difference between objective values between generations. If the changes in the objective values are within 1% in three successive generations then the solution is considered converged. Once convergence is reached, the MOGA produces the "fittest" individual from the final generation as the optimal geometry. Figure 4.3 shows a flowchart for the genetic algorithm process.
4.2 Shape Optimization of a Blunt Body using a Multi-Objective Genetic Algorithm

The case of the axisymmetric blunt body has been studied and optimized previously in the literature [3] for the reduction of both drag and heat transfer; however, the flow conditions have been in the continuum regime and did not account for thermal non-equilibrium or rarefaction. This project aims to optimize the same axisymmetric geometry with the same flow conditions but takes into account the effects of rarefaction, Maxwell’s slip wall boundary conditions and thermal non-equilibrium. In order to properly analyze all of these effects it is important to choose a Knudsen number well into the slip flow regime. The Knudsen number of 0.01 was therefore chosen for the shape optimization of the blunt body so that the effects of the slip flow regime and thermal non-equilibrium on the optimization results could be studied.

4.2.1 MOGA Parameters for Axisymmetric Blunt Body Shape Optimization

The free stream flow conditions and mesh for the blunt body at Kn = 0.01 are the same as given in Chapter 2. The MOGA was written using MATLAB and the custom flow solver ZLOW was used in all simulations. The blunt body shape was generated using a single Bezier curve with 8 control points. Figure 4.2 illustrates an example of a randomly generated individual for the blunt body optimization. In order to maintain a blunt leading edge with appropriate size, the first two and last two control points are fixed for all cases. These are shown in Figure 4.4 as the black control points. The green control points were allowed to vary randomly and were used to control the optimization shape. These four control points were used for the crossover and mutation functions in the MOGA.

The multi-objective genetic algorithm parameters are as follows:

- Objective values: Drag coefficient and maximum heat transfer
- Evenly weighted objective values
Figure 4.4: Randomly generated Bezier curve with eight control points for blunt body optimization
• Initial generation population: 20
• Daughter generation population: 20 (10 survivors/parents and 10 offspring)
• Survivors per generation: 10
• Maximum number of generations: 41
• Crossover types: y-coordinate swap and control point average
• Mutation Probability: 0.5
• Convergence Criteria: ≤1% improvement in generation averaged objective values over three successive generations

The program was required to run a minimum of 7 generations to allow proper development from the initial random population.

### 4.2.2 Computational Resources

Table 4.1 shows the computational resources used in this optimization. The workstation used an Intel i5-6600k (4 cores) overclocked to 4.4GHz with a Hyper 212 Evo cooler and 16 GB of RAM. All computational times are listed in terms of one processor core at 4.4Ghz. In this case, 4.8 CPU hours were completed in 1.2 hours by utilizing all 4 available CPU cores.

<table>
<thead>
<tr>
<th>Mesh Size (cells)</th>
<th>Generations</th>
<th>Total Individuals</th>
<th>Time Per Individual (hours)</th>
<th>Computational Cost (hours)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.4E6</td>
<td>27</td>
<td>280</td>
<td>4.8</td>
<td>1344</td>
</tr>
</tbody>
</table>

### 4.2.3 Blunt Body MOGA Results

The shape optimization results were obtained for the blunt body in hypersonic rarefied flow with rotational non-equilibrium with evenly weighted importance on the reduction of drag
Figure 4.5: Optimized blunt body shape at Kn = 0.01 using NS2T model and heat transfer. The optimized geometry presented in Figure 4.5 shows a large increase in leading edge radius with a streamlined body following the leading edge. The primary focus of this study was to analyze how including the effects of rarefaction and thermal non-equilibrium change the body shape. Figure 4.6 shows the results from the optimization process against the results presented by Seager and Agarwal [3]. In this figure, the blue line is the optimized geometry from this study, the red line is from Ref. [3] and the green line is the original geometry. Including the effects of rarefaction and translational/rotational thermal non-equilibrium gives an optimized geometry very similar to that obtained in Ref. [3]. The new geometry from this study has a slightly larger leading edge radius and a shallower body line.

The flow field properties of the optimized geometry are given in Figures 4.7 - 4.10. Figure 4.7 shows that the optimized blunt body has a larger shock standoff distance than the original blunt body. This is expected due to the larger leading edge radius and the benefit of larger shock standoff distance reducing the wall temperature in hypersonic flow. This relationship is shown in Figure 4.9 where it is clear that the high temperature area of the shock standoff
Figure 4.6: Optimized blunt body shape (blue) at Kn = 0.01 using NS2T model compared to that in Ref. [3] (red) for Kn = 0 using NS1T model
region is further away from the wall stagnation point for the optimized geometry. The pressure contours in Figure 4.8 show a larger high pressure region near the leading edge of the optimized blunt body; however, this high pressure region goes away rapidly compared to that for the original geometry as a function of wall location.

The effects of non-equilibrium are demonstrated by plotting the rotational and translational temperatures from each case along the stagnation line in Figure 4.11. The temperatures for the optimized case vary only slightly from the non optimized shape. The translational temperatures increase significantly at the beginning of the shock and the rotational temperature begins to move towards equilibrium only in the boundary layer region. The two temperatures reach equilibrium in both cases in the post shock boundary layer region. Both the maximum translational and rotational temperatures are slightly reduced in the optimized case.

Figures 4.12 - 4.14 show the plots of various surface properties of the optimized and original geometries. The surface properties provide the best insight into the improvements gained by the optimization. The heating coefficient on the surface of the optimized geometry is significantly reduced along the leading edge of the blunt body, where the heating coefficient is the highest. The heating coefficient along the portion of the blunt body that meets the aft-cylinder is higher than that in the original case; however, the leading edge is the driving force behind the heat flux, therefore this increase is acceptable.

The pressure coefficient follows the same trend as the heating coefficient for the optimized blunt body. The skin friction is shown in Figure 4.14. There is much higher skin friction on the leading edge and on rear portion of the optimized blunt body. The improvements come from the center 50% of the blunt body curve where there is a significant reduction in skin friction.

Table 4.2: Blunt body optimization: improvements in objective values

<table>
<thead>
<tr>
<th>Objective Value</th>
<th>Original</th>
<th>Optimized</th>
<th>Improvement (percent)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$C_D$</td>
<td>1.136</td>
<td>0.952</td>
<td>16.22</td>
</tr>
<tr>
<td>Qmax</td>
<td>5.96E5</td>
<td>5.50E5</td>
<td>7.78</td>
</tr>
</tbody>
</table>

58
Figure 4.7: Optimized blunt body Mach contours compared to the original blunt body

Figure 4.8: Optimized blunt body pressure contours compared to the original blunt body
Figure 4.9: Optimized blunt body translational temperature contours compared to the original blunt body
Figure 4.10: Optimized blunt body rotational temperature compared to the original blunt body
Figure 4.11: Translational and rotational temperatures along the stagnation line of optimized blunt body compared to the original blunt body.
Figure 4.12: Heating coefficient of optimized blunt body geometry compared to the original blunt body
Figure 4.13: Pressure coefficient of optimized blunt body compared to the original blunt body.
Figure 4.14: Skin friction of optimized blunt body compared to the original blunt body
The blunt body optimization results presented in this study demonstrates approximately 16% reduction in drag coefficient and 8% reduction in maximum heat flux. These results contrast a 22% and 13% reduction, respectively, given in the literature for the same geometry[3]. There are three main differences in the flow conditions that could have led to these results: thermal non-equilibrium, rarefaction, and turbulence. The study in the literature[3] included a turbulence model in the simulations while this study assumes laminar flow throughout. It is also possible that the effects of rarefaction and thermal non-equilibrium are less sensitive to the body shape, thereby reducing the optimization potential. Further work needs to be done in optimizing shapes at various levels of rarefaction to determine how the blunt body optimization is affected.
Chapter 5

Shape Optimization of a Bicone for Reduction of Both Drag and Heat Transfer Using a Genetic Algorithm

The optimization of a 3D axisymmetric bicone in hypersonic flow is a novel problem that has not been investigated in literature prior to this study. Due to the relative complexity of the geometry and the increased computational resources required for the simulation and optimization of the bicone in comparison to the blunt body, the bicone is only optimized for the case of evenly weighted objective functions for reducing both drag and heat transfer in the continuum regime with thermal non-equilibrium.

5.1 MOGA Parameters for Axisymmetric Bicone Shape Optimization

The free stream flow conditions for the bicone optimization are the same as the flow conditions given for the NS2T case of $Kn = 0.0013$ in Chapter 3. The bicone geometry was generated by joining two Bezier curves at the fore cone/aft cone juncture point of the original bicone geometry. An example of a randomly generated geometry for the bicone is shown in Figure 5.1.
In Figure 5.1 the red curve represents the fore cone and the green line represents the aft cone. In order to preserve the characteristics of the bicone geometry, the first two and last two points of each curve are fixed to guarantee a sharp leading edge and the characteristic angle change at the cone junction. These points are shown in Figure 5.1 as the black dots. Each curve is represented by 7 control points, so the optimization is done using 3 variable control points for each curve. In order to reduce the computational expense the bicone is run on the same mesh given in Chapter 3, with 40 exponentially spaced cells instead of 80 until the MOGA achieves convergence. After the optimization has converged with the coarse mesh, it is restarted from the converged solution with the refined 80 point mesh to achieve a mesh independent solution.

The multi-objective genetic algorithm parameters are as follows:

- Objective values: drag coefficient and maximum heat transfer
- Evenly weighted objective values
- Population size: 20 (10 survivors/parents and 10 offspring)
• Survivors per generation: 10
• Maximum number of generations: 60
• Crossover types: y-coordinate swap and control point average
• Mutation probability: 0.5
• Convergence criteria: ≤1% improvement in generation averaged objective values over three successive generations

Note the increased maximum number of generations. It was anticipated that the bicone optimization process would take more generations to achieve convergence due to the increased number of degrees of freedom required (6 control points and 2 separate curves).

5.2 Bicone Optimization Computational Resources

Table 5.1 shows the computational resources used in this optimization. The workstation used an Intel i7-4790 (4 cores + 4 with hyper threading) at 3.60GHz with 32 GB of RAM. All computational times are listed in terms of one processor core at 3.60GHz. In this case, 19.2 CPU hours were completed in 2.4 hours by utilizing all 8 available CPU cores. This table illustrates why using the coarse mesh then refining after convergence is so vital to the optimization of the bicone. To run all individuals with the coarse mesh would take an estimated 27648 CPU hours to converge, assuming that the program would achieve convergence in the same number of generations, which would equal 144 days of continuous computing on the i7-4790 available for this study.

Table 5.1: Computational resources for bicone optimization

<table>
<thead>
<tr>
<th>Mesh Size (cells)</th>
<th>Generations</th>
<th>Total Individuals</th>
<th>Time Per Individual (hours)</th>
<th>Computational Cost (hours)</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.0E5</td>
<td>44</td>
<td>450</td>
<td>19.2</td>
<td>8640</td>
</tr>
<tr>
<td>8.0E5</td>
<td>3</td>
<td>30</td>
<td>57.6</td>
<td>1728</td>
</tr>
</tbody>
</table>
5.3 Bicone MOGA Results

The shape optimization results were obtained for the bicone in hypersonic, thermal non-equilibrium flow with evenly weighted objective functions for reduction of both drag and heat transfer. Figure 5.2 shows the optimized bicone geometry and the original bicone geometry. The optimized shape shows a rounding of the leading edge of the fore cone, with an immediate turn to concavity. The fore cone then takes a curved approach to the constrained junction between the fore and aft cones. The aft cone mimics the fore cones optimized shape, featuring rounded boundaries and a concave center.

![Figure 5.2: Optimized bicone shape at Kn = 0.0013 using NS2T model](image)

The flow field properties are shown in Figures 5.3-5.6. Figure 5.3 shows a shock attached to the fore cone, with shock detachment occurring at the fore-aft cone juncture. The shock structure in the optimized case is smaller than than that of the original bicone geometry. The optimized shape also has a lower velocity in the post shock region.
The pressure contours in Figure 5.4 show a reduced high pressure region in the optimized geometry. This reduced high pressure region has also been moved farther back on the aft cone in comparison to the original bicone geometry.

The temperature contours in Figure 5.5 show a larger post shock temperature than the original geometry; however, there is a lower temperature along the wall for the fore cone and first half of the aft cone. The high temperature region of the flow only attaches to the body of the optimized bicone geometry for a short duration on the aft cone.

The translational and rotational temperatures along the wall are shown in Figure 5.7. The optimized shape shows a reduction in translational temperature at the leading point of the bicone. The optimized bicone has a higher translational temperature than the original geometry for much of the fore cone, when the wall temperature is at its highest. Although the optimized shape has a higher translational temperature than the original geometry, there is a reduction in the peak wall temperature at the leading point of the optimized bicone shape, which corresponds to the optimization parameter of minimizing the maximum heat flux.

The heating coefficient is shown in Figure 5.8 for both the optimized bicone and original bicone. The heating coefficient on the leading edge of the optimized bicone showed a large reduction in comparison to the original bicone shape. The large peak on the aft cone corresponds to the high temperature region in the post shock region of the optimized bicone geometry. This confirms the reduction that was seen in the temperature contours previously.

The reduction of drag in the bicone optimization was small in comparison to the heat transfer. This is due to the nature of a sharp nosed geometry’s performance in hypersonic flow. Since a sharp leading edge corresponds to a larger heat transfer and lower drag, a bicone would present greater opportunity for optimizing the heat transfer than the drag. Nonetheless, a reduction in peak pressure coefficient and skin friction is shown in Figures 5.9 and 5.10.
Figure 5.3: Optimized bicone Mach contours compared to the original bicone geometry
Figure 5.4: Optimized bicone pressure contours compared to the original bicone geometry
Figure 5.5: Optimized bicone translational temperature contours compared to the original bicone geometry
Figure 5.6: Optimized bicone rotational temperature contours compared to the original bicone geometry
Figure 5.7: Translational and rotational temperatures along the surface of the optimized bicone geometry compared to original geometry

Figure 5.8: Heating coefficient along the surface of the optimized bicone geometry compared to original geometry
Figure 5.9: Pressure coefficient along the surface of the optimized bicone geometry compared to original geometry

Figure 5.10: Skin friction along the surface of the optimized bicone geometry compared to original geometry
The optimization of the 3D axisymmetric bicone was able to obtain a 4.21% reduction in total drag and a 48% reduction in the maximum heat flux. These results provide a meaningful contribution to the design of hypersonic vehicle design and the potential of using modified bicone shapes. Further work can be done on the subject by creating the pareto-optimal front for the bicone, adding the effects of rarefaction or including a chemical non-equilibrium model to the optimization process.

<table>
<thead>
<tr>
<th>Objective Value</th>
<th>Original</th>
<th>Optimized</th>
<th>Improvement (percent)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Drag</td>
<td>30.94</td>
<td>29.65</td>
<td>4.21</td>
</tr>
<tr>
<td>Qmax</td>
<td>1.30E5</td>
<td>6.76E4</td>
<td>48.18</td>
</tr>
</tbody>
</table>

Table 5.2: Bicone optimization: improvements in objective values
Chapter 6

Conclusions

This study had three primary objectives:

- Quantify the effect of both rarefaction and thermal translational/rotational non-equilibrium on a blunt body and bicone in hypersonic flow
- Analyze the effect of rarefaction and thermal non-equilibrium on the shape optimization of a blunt body in hypersonic flow for reduction in both drag and heat transfer
- Shape optimize bicone in hypersonic flow including the effects of thermal non-equilibrium for reduction in both drag and heat transfer

This study employed the Navier-Stokes equations with a two temperature model and Maxwell’s slip wall boundary conditions to simulate the blunt body and bicone in slip flow regime with and without thermal non-equilibrium. Each case was computed in both the continuum and slip flow regimes. The blunt body flow was computed at Mach 7.1 and Knudsen numbers of 0.002, 0.01, 0.05 and 0.1 while the bicone flow was computed at Mach 15.6 and Knudsen numbers of 0.0013, 0.01, 0.057 and 0.1. To ensure that the effects of rotational non-equilibrium effects were present in the flow, a diatomic gas model was used: standard air for the blunt body and nitrogen for the bicone.

Comparisons of the one temperature (NS1T) and two temperature (NS2T) models showed that for both the blunt body and bicone the effect of translational/rotational thermal non-equilibrium increased with increase in rarefaction. It was shown that the shock region introduced the thermal non-equilibrium by imparting a sharp spike in the translational temperature while not providing enough collisions for the rotational temperature to equilibrate.
For cases within the continuum regime, the flow was able to reach an adequate number of collisions immediately after the shock for thermal equilibrium. As rarefaction increased the shock wave became more diffuse and the equilibrium point was pushed closer to wall. In the most rarefied case of Kn = 0.1 the flow never returned to a thermal equilibrium.

The surface properties of the blunt body simulations showed that the NS2T model predicted a lower pressure coefficient, skin friction and heating coefficient for all Knudsen numbers. The NS2T simulations also showed an increase in peak thermal temperature as non-equilibrium and rarefaction increased. The largest effect of NS2T model was seen in the detached shock standoff distance for the blunt body. The NS2T model predicted a 24% and 34% greater shock standoff distance compared to that obtained by the the NS1T model for the most rarefied blunt body cases at Kn = 0.05 and 0.1 respectively. In addition to the increased shock standoff distance, the shock was much more diffused as rarefaction increased.

The bicone flow properties exhibited special features that were not seen in the case of the blunt body. The sharp leading edge of the bicone was the primary source of thermal non-equilibrium. As Knudsen number increased, the magnitude of the translational/rotational temperature in non-equilibrium increased. This in turn had an effect on the predicted rates of heat transfer for the bicone. For low Knudsen number flow at Kn 0.0013 and 0.01, the NS1T and NS2T models predicted very similar surface heat flux and overall maximum heat flux values. As rarefaction increased both the shape and magnitude of the heat transfer begin to change. The NS2T model predicted 12.5% and 13.61% lower maximum heat flux for Knudsen numbers of 0.057 and 0.1 respectively. The surface properties of pressure and shear stress showed little dependence on thermal non-equilibrium with maximum predicted difference of only 1.7% in the most rarefied case.

From these results it can be concluded that the inclusion of thermal translational/rotational non-equilibrium in the flow computations reduces the values for heat transfer and drag compared to that in thermal equilibrium.

The shape optimization of a blunt body including the effects of rarefaction and thermal non-equilibrium showed that these effects have very limited influence on the optimization. The optimized shape obtained in this study was nearly identical to that presented in literature [3]; however, rarefaction and thermal non-equilibrium negatively influence the effectiveness of optimization in reducing drag and heat transfer. In Ref. [3] it was demonstrated that a 22%
reduction in drag and 13% reduction in maximum heat transfer can be achieved; however, only a 16% reduction in drag and 8% reduction in maximum heat transfer has been achieved in this study. These results suggest that rarefaction and thermal non-equilibrium reduce the ability of the optimized body to reduce drag and heat transfer compared to that of an optimized body in continuum flow and thermal equilibrium.

The primary contribution of this study to the field is the optimization of the 3D axisymmetric bicone in hypersonic thermal non-equilibrium for reduction of both drag and heat transfer. This shape optimization of a bicone has not been conducted in the literature before for any flow regime. The optimization was run in the continuum regime, $Kn = 0.0013$, with evenly weighted importance on the reduction of drag and heat transfer. Due to the sharp leading edge of the bicone it was expected that large improvements could be made in the area of maximum heat flux on the surface. The optimization process was able to reduce the maximum heat flux on the axisymmetric bicone by 48% while also reducing the drag by 4%. Future work on this topic should include the creation of a pareto-optimal front for heat transfer and drag on the axisymmetric bicone. The effects of rarefaction on the bicone optimization should also be investigated.
References


Vita

Samuel Taylor Gardner

Degrees

M.S. Aerospace Engineering, December 2016
Department of Mechanical Engineering & Materials Science, Washington University in St. Louis

B.S. Mechanical Engineering, December 2016
Department of Mechanical Engineering & Materials Science, Washington University in St. Louis

B.A. Magna Cum Laude, Physics, May 2014
Department of Physics,
Drury University

Professional Societies

American Institute of Aeronautics and Astronautics

Publications


December 2016
, Gardner, M.S. 2016