ANALYSIS OF HYPersonic FLOW USING THREE DIMENSIONAL NAVIER-STOKES EQUATIONS

A THESIS SUBMITTED TO THE GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES OF MIDDLE EAST TECHNICAL UNIVERSITY

BY

MUHARREM ÖZGÜN

IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE OF MASTER OF SCIENCE IN AEROSPACE ENGINEERING

SEPTEMBER 2016
Approval of the thesis:

ANALYSIS OF HYPersonic FLOW USING THREE DIMENSIONAL NAVIER-STOKES EQUATIONS

submitted by MUHARREM ÖZGÜN in partial fulfillment of the requirements for the degree of Master of Science in Aerospace Engineering Department, Middle East Technical University by,

Prof. Dr. Gülbin Dural Ünver
Dean, Graduate School of Natural and Applied Sciences

Prof. Dr. Ozan Tekinalp
Head of Department, Aerospace Engineering

Assoc. Prof. Dr. Sinan Eyi
Supervisor, Aerospace Engineering Department, METU

Examinning Committee Members:

Prof. Dr. Serkan Özgen
Aerospace Engineering Department, METU

Assoc. Prof. Dr. Sinan Eyi
Aerospace Engineering Department, METU

Prof. Dr. Ünver Kaynak
Mechanical Engineering Department, TOBB ETU

Assist. Prof. Dr. Nilay Sezer Uzol
Aerospace Engineering Department, METU

Assist. Prof. Dr. Durmuş Sinan Körpe
Aeronautical Engineering Department, THK University

Date:
I hereby declare that all information in this document has been obtained and presented in accordance with academic rules and ethical conduct. I also declare that, as required by these rules and conduct, I have fully cited and referenced all material and results that are not original to this work.

Name, Last Name: MUHARREM ÖZGÜN

Signature: 

iv
ABSTRACT

ANALYSIS OF HYPersonic FLOW USING THREE DIMENSIONAL NAVIER-STOKES EQUATIONS

ÖZGÜN, MUHARREM
M.S., Department of Aerospace Engineering
Supervisor : Assoc. Prof. Dr. Sinan Eyi

September 2016, 58 pages

Hypersonic flow is a popular topic and using CFD is a necessity in hypersonic flow problems because hypersonic experiments cost too much. Hypersonic flow occurs at high Mach numbers, and extreme conditions such as high temperatures, low densities, strong shocks can be seen at this type of flow. Atmospheric reentry vehicles do their tasks at hypersonic flow conditions and very high temperatures can be seen within hypersonic boundary layer on nose surface of the vehicles due to the viscous dissipation. Also, implementing turbulence model is essential since it is effective on aerodynamic forces.

The purpose of this study is to develop an accurate and efficient CFD solver that can be used in hypersonic flows. The flow analysis is based on three dimensional Navier-Stokes equations. These equations are solved by simple Euler integration. The analytical method is used to calculate the Jacobian matrix. Different upwind flux splitting schemes are used for the computational process. Grid refinement study is done to remove the dependency of the solutions on resolution of grids. Flow parameters are analyzed on Apollo AS-202 Command Module and Atmospheric Reentry Demonstrator (ARD). Also, algebraic Baldwin-Lomax turbulence model is used to analyze hypersonic turbulent flow and difference between results of laminar and turbulent flows are observed.
ÖZ

NAVIER-STOKES DENKLEMLERİ KULLANILARAK ÜÇ BOYUTLU HİPERSONİK AKIŞ ANALİZİNİN YAPILMASI

ÖZGÜN, MUHARREM

Yüksek Lisans, Havacılık ve Uzay Mühendisliği Bölümü

Tez Yöneticisi : Doç. Dr. Sinan Eyi

Eylül 2016, 58 sayfa


Anahtar Kelimeler: Hipersonik akış, taşınmala ısı transferi, türbülans modellemesi, hesaplamalı akışkanlar dinamiği, reentry araçları, navier-stokes denklemleri, newton yöntemi
To my family and people who are reading this page
ACKNOWLEDGMENTS

First of all, I would like to express my sincere gratitude to my advisor Assoc. Dr. Sinan Eyi for its valuable guidance, support and advice throughout my research and education.

I would like to specially thank my father and mother for their invaluable support and love especially at hard times of my life.

Special thanks to my dear friend and colleague Hilmi Berk Gür for his support and sincere friendship. I would also specially thank to Emrecan Suiçmez, Özcan Yırtıcı, Oğuz Kaan Onay, Tuğba Pişkin, Gülay Şenol, Ömer Ataş, Derya Kaya, Özgür Yalçın, Özgür Harputlu, Yunus Tansu Aksoy, Pınar Eneren, Eren Turanoğuz, Emre Oktay, Engin Leblebici, Ali Yıldırım, Ezgi Orbay and all of my other colleagues and friends at METU for their help and fellowship. I also thank to my dear friend Armağan Temiz for his supports. I would also thank to my special friends Can, Gökben, Faruk, Umur, and all my friends that I really spent great time at some stages of my life.

I also thank God to give me the opportunity to meet my sweetheart Yudum who helped me at each stage of my life with her endless love and support.

Finally, I would like to thank The Scientific and Technological Research Council of Turkey (TÜBİTAK) for providing financial support during my studies within the project ’112M129’.
# TABLE OF CONTENTS

**ABSTRACT** ................................................................. v

**ÖZ** ................................................................. vii

**ACKNOWLEDGMENTS** ...................................................... x

**TABLE OF CONTENTS** ....................................................... xi

**LIST OF TABLES** ............................................................. xii

**LIST OF FIGURES** .......................................................... xiii

**LIST OF ABBREVIATIONS** ................................................ xiv

**CHAPTERS**

1 INTRODUCTION ....................................................... 1

1.1 Motivation ....................................................... 1

1.2 Objective ....................................................... 2

1.3 Hypersonic Flow ................................................... 2

1.3.1 Flow Characteristics ........................................ 3

1.3.2 Reentry Vehicles ............................................. 6

1.4 Modeling of Turbulent Flow ................................. 7

1.4.1 Algebraic Models ............................................ 7
LIST OF TABLES

TABLES

Table 1.1 Features of different hypersonic vehicles [8] . . . . . . . . . . . . . 7

Table 3.1 Generated Meshes for Grid Independence Study . . . . . . . . . . . 33

Table 3.2 Generated Meshes for Grid Independence Study . . . . . . . . . . . 34

Table 3.3 CPU Time and Residual Values for Flux Splitting Schemes . . . . . 36

Table 3.4 Freestream flow values . . . . . . . . . . . . . . . . . . . . . . . . 39

Table 3.5 Freestream flow values . . . . . . . . . . . . . . . . . . . . . . . . 46
LIST OF FIGURES

FIGURES

Figure 1.1 Shock Waves at Mach 2 and Mach 20 [5] . . . . . . . . . . . . . . 3
Figure 1.2 Entropy Layer [5] . . . . . . . . . . . . . . . . . . . . . . . . . . 4
Figure 1.3 Temperature Profile [5] . . . . . . . . . . . . . . . . . . . . . . . 4
Figure 1.4 Shock Layer [5] . . . . . . . . . . . . . . . . . . . . . . . . . . . 5
Figure 1.5 Changing of Shock Layer Temperature with Reentry Velocity [7] . 6

Figure 2.1 Generalized Transformation from the Physical Domain to the Com-
putational Domain . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 15
Figure 2.2 Flux Vectors and Control Volume . . . . . . . . . . . . . . . . . . 22

Figure 3.1 Dimensions of Apollo AS-202 Capsule (meter) . . . . . . . . . . 31
Figure 3.2 3-D model of Apollo AS-202 Capsule . . . . . . . . . . . . . . . 32
Figure 3.3 Grid Independence Study for Apollo AS-202 Capsule . . . . . . . 32
Figure 3.4 S and Rb parameters . . . . . . . . . . . . . . . . . . . . . . . . . 33
Figure 3.5 Normalized pressure values on the nose surface of Apollo for dif-
ferent grid resolutions at AOA= 0° . . . . . . . . . . . . . . . . . . . . . 34
Figure 3.6 IJK planes in 3D flow domain . . . . . . . . . . . . . . . . . . . 34
Figure 3.7 Residual History of Flux Splitting Methods . . . . . . . . . . . . . 35
Figure 3.8 Comparison of Van Leer and AUSM Schemes by Flow Variables . 36
Figure 3.9 Comparison of Van Leer and AUSM Schemes by Pressure and U-
velocity Values . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 37
Figure 3.10 Comparison of Present Temperature Result and Representative Re-
sult of a Study . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 38
Figure 3.11 Comparison of Present Density Result and a Shadowgraph Image . 38
Figure 3.12 Comparison of the Computational and Experimental Normalized Pressure Values along the Body Nose for the Apollo Command Module (AOA = 0°) . 39
Figure 3.13 Comparison of the Computational and Experimental Normalized Pressure Values along the Body Nose for the Apollo Command Module (AOA = 33°) . 40
Figure 3.14 Laminar(left) and Turbulent(right) Flow Results Around Apollo Capsule (AOA =0°) . 42
Figure 3.15 Turbulent Flow Results at 20° and 33° angle of attack . 43
Figure 3.16 Comparison of Laminar and Turbulent Flows by Pressure and U-velocity Values along the Stagnation Line (AOA =0°) . 45
Figure 3.17 Comparison of Laminar and Turbulent Flows by Temperature Values along the Stagnation Line (AOA =0°) . 45
Figure 3.18 Velocity Vectors Around Apollo AS-202 Capsule (AOA =0°) . . 46
Figure 3.19 Geometry of ARD and 2-D View of its Grid . . . . . . . . 47
Figure 3.20 Grid Distribution and 3-D View of ARD Grid (72x48x18) . . . 47
Figure 3.21 Comparison of the Computational and Experimental Normalized Pressure Values along the Body Nose for the ARD (AOA = 20°) . . . 48
Figure 3.22 Laminar(left) and Turbulent(right) Flow Results Around ARD (AOA =0°) . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 49
Figure 3.23 Velocity Flux Vectors and Streamlines at the Back Region of ARD (AOA = 20°) . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 50
Figure 3.24 Streamlines and Vorticity Region around ARD (AOA = 0°) . . . 50
LIST OF ABBREVIATIONS

CFD Computational Fluid Dynamics
AUSM Advection Upwind Splitting Method
DNS Direct Numerical Simulation
LES Large Eddy Simulation
FVM Finite Volume Method
NASA National Aeronautics and Space Administration
RANS Reynolds Averaged Navier Stokes
TÜBİTAK The Scientific and Technological Research Council of Turkey
METU Middle East Technical University
CAV Cruise and Acceleration Vehicle
RV-W Winged Reentry Vehicle
RV-NW Non-winged Reentry Vehicle
SGS Sub-grid Scale
ARD Atmospheric Reentry Demonstrator
CPU Central Processing Unit

LIST OF SYMBOLS

\( a \) Speed of sound
\( \hat{F}_{c}, \hat{G}_{c}, \hat{H}_{c} \) Convective (Inviscid) flux vectors in generalized coordinates
\( \hat{F}_{v}, \hat{G}_{v}, \hat{H}_{v} \) Viscous flux vectors in generalized coordinates
\( J \) Jacobian of transformation matrix
\( M \) Mach number
\( \hat{W} \) Conservative vector of flow variables in generalized coordinates
\( \hat{R} \) Residual vector
\( Re \) Reynolds number
\( U, V, W \) Contravariant velocity components
\( u, v, w \) Velocity vector components
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \rho )</td>
<td>Density</td>
</tr>
<tr>
<td>( T )</td>
<td>Temperature</td>
</tr>
<tr>
<td>( p )</td>
<td>Pressure</td>
</tr>
<tr>
<td>( \xi, \eta, \zeta )</td>
<td>Generalized coordinates</td>
</tr>
<tr>
<td>( \xi_x, \xi_y, \xi_z )</td>
<td>Transformation metrics in ( \xi ) direction</td>
</tr>
<tr>
<td>( \eta_x, \eta_y, \eta_z )</td>
<td>Transformation metrics in ( \eta ) direction</td>
</tr>
<tr>
<td>( \zeta_x, \zeta_y, \zeta_z )</td>
<td>Transformation metrics in ( \zeta ) direction</td>
</tr>
<tr>
<td>( S )</td>
<td>Source term</td>
</tr>
<tr>
<td>( \tau )</td>
<td>Shear stress</td>
</tr>
</tbody>
</table>
CHAPTER 1

INTRODUCTION

1.1 Motivation

Computational tools are improved everyday with the development of computer technology. These advanced tools are used for the many engineering studies. Computational Fluid Dynamics (CFD) is a field that advanced computational tools are used. CFD is beneficial for reducing the number of experiments and it accelerates the design process. It is also used in hypersonic flow problems since hypersonic experiments cost too much. Because hypersonic flow conditions include huge amount of energy and producing them is difficult. Additionally, there is not many experimental setup that perform such experiments. Thus, success of the spacecraft vehicle design directly depends on the simulation results.

Hypersonic flow is a popular research topic and it is an important field especially for aerospace and military industries [1]. Reentry vehicles are also area of interest since they do their tasks at hypersonic speeds. High convective heat fluxes and aerodynamic heating are the critical problems that is come across at designing reentry vehicles. The numerical Navier-Stokes equations can be used for the analysis of flow parameters for reentry vehicles in a atmosphere, where the continuity assumption is true [2].

Turbulence modeling is one of the research fields that its importance increases day by day with the complex flow problems. The turbulence is a complex issue and it is also significant for hypersonic vehicles because it is effective on aerodynamic forces. Turbulent flows are seen in many hypersonic applications. Therefore, implementation a turbulence model is often essential while modeling the flow [1, 3, 4].
1.2 Objective

Developing a reliable and robust CFD solver for the analysis of the viscous hypersonic flow and investigating turbulence effects are the main objectives of the present study.

The flow analysis depends on three dimensional Navier-Stokes equations. These equations are solved simultaneously by using Euler integration. Analytical differentiation is used to evaluate the Jacobian matrix. The sparse Jacobian matrix is LU factorized and UMFPACK sparse matrix solver is used to execute the solution.

In the present study, Apollo AS-202 Command Module is chosen as a geometrical model. Viscous terms are implemented to the code and Navier-Stokes solver is generated. Also, various flux splitting schemes are reviewed and used for the computational process. Some flow parameters are analyzed with the help of numerical integration of the Navier-Stokes equations. Lastly, one-equation Baldwin-Lomax turbulence model is implemented and turbulence effects are observed in hypersonic flow field.

1.3 Hypersonic Flow

Human being has always wanted to fly faster and higher, and an exponential growth can be observed in speed and altitude with the advancements in aircraft in the twentieth century [5]. Some results of these advancements were supersonic aircrafts, experimental hypersonic airplanes and manned reentry vehicles which operate at conditions Mach number changes between 25 and 36.

Hypersonic flow conditions can be generally seen when Mach number is higher than 5. However 5 is not a magic number. Flow does not dramatically change when Mach number exceeds 5. Also, flowfields may begin to exhibit many hypersonic flow characteristics at smaller Mach number conditions. So basic assumption for hypersonic flows is;

$$M_\infty \equiv \frac{U_\infty}{a_\infty} \gg 1$$

(1.1)

Thus, the internal thermodynamic energy of the freestream fluid particles is smaller than kinetic energy of the freestream fluid particles [6]. Since there is not a distinct
number for hypersonics phenomena, brief description of some characteristics of hypersonic flow is beneficial.

Thin shock layers, entropy layers, viscous interaction, high temperature and low density effects are some characteristics of hypersonic flows [5].

1.3.1 Flow Characteristics

Increase of density across the shock wave becomes larger when the Mach number is increased. This means that the distance between the shock wave and the hypersonic body becomes smaller. The shock layer is defined as the flowfield between the body and the shock wave. This shock layer is very thin for hypersonic speeds and lies close to the body [5]. This change can be seen below.

![Figure 1.1: Shock Waves at Mach 2 and Mach 20](image)

In Figure 1.2 it can be seen that shock wave is curved in the nose region. Strong shock means large entropy increase across a shock wave. Hence, strong entropy gradients can be generated in the nose region. This flowfield with a shock-detachment distance (d) is called entropy layer. This entropy layer affects the boundary layer inside it along the surface [5]. It is a region of strong vorticity.

Hypersonic flow contains a huge amount of kinetic energy. When a boundary layer on a flat plate is considered as seen in Figure 1.3, it is known that hypersonic flow is slowed by viscous effects. Hence, some amount of kinetic energy is transformed into internal energy of the fluid and the temperature increases in the boundary layer. This is called viscous dissipation [5]. The temperature profile within the boundary layer can be seen in Figure 1.3.
The viscosity coefficient increases with temperature. Additionally, the increment in temperature causes the decrement in density through the boundary layer because the pressure is constant in the normal direction. Thus, the thickness of the boundary layer must be larger to satisfy the required mass flow rate through the boundary layer. Both of these effects combine to grow hypersonic boundary layers more rapidly compared to slower speeds. The outer inviscid flow greatly changes when the boundary layer is extremely thickened. This change also affects the growth of the boundary layer. This is called viscous interaction between the boundary layer and the outer inviscid flow. These interactions have significant effects on the surface pressure distribution, skin friction and the heat transfer.

Very high temperatures can be seen within hypersonic boundary layer due to the viscous dissipation. Dissociation and ionization are possible because these high temperatures cause to excite vibrational energy within molecules. The nose region of a blunt
body is sketched in Figure 1.4. The temperature of the gas behind the shock wave can be extremely large at hypersonic speeds. In Apollo reentry mission, the temperature reaches to approximately 11000 K at a Mach number of 36 [
5].

Figure 1.4: Shock Layer [
5]

Figure 1.5 shows the changing temperature behind a normal shock wave with the free-stream velocity. The curves represent respectively the tendency of nonreacting calorically perfect gas which its specific heat ratio $\gamma = 1.4$ and equilibrium chemically reacting gas. The difference between the curves reveals the vitality of chemically reacting effects. The gas doesn’t behave ideally when the temperature of the gas increased to high values [
5]. All of these are called high temperature or real gas effects.

High temperature and chemically reacting flows have impact on high heat transfer rates, lift, drag and moments on hypersonic vehicle. Especially aerodynamic heating plays an important part on all aspects of the hypersonic vehicle design. Communications blackout is another consequence of high temperature flows because of free electrons which is caused by ionization. Free electrons absorb radiation of radio frequency and transmission of radio waves becomes impossible [
5].

The flight at very high altitudes involves low density flow. Thus, it is a physical aspect of some hypersonic applications. Mean free path changes at low density flows and this changes the flow physics. For instance, continuity of the air begin to break down
at an altitude of 300,000 ft (90km) which is Space Shuttle operation condition and individual molecular effects become more important [5].

1.3.2 Reentry Vehicles

Reentry vehicles are the vehicles entering to the atmosphere of earth from space. There are some types of reentry vehicles. Some aerothermodynamic features of hypersonic vehicles can be seen in Table 1.1.

It might be expressed that laminar-turbulent transition, turbulence and viscous thermal surface effects play a major role for CAV’s, while high temperature real gas effects and thermo-chemical thermal surface effects are very significant for RV-W’s and RV-NW’s. The CAV is slender and flies at small angles of attack. However RV-W and RV-NW have a blunt shape and RV-W flies at large angles of attack to increase the effective bluntness [8].

Non-winged reentry vehicles (RV-NW) became important after the designers decided that vehicle shapes should be simple and compact. Thus, capsules and probes were made as examples of RV-NW’s. The lunar mission of Apollo capsule was a pioneer and successful exploration to direct the future of humanity.
Table 1.1: Features of different hypersonic vehicles [8]

<table>
<thead>
<tr>
<th></th>
<th>Winged Reentry Vehicles (RV-W)</th>
<th>Cruise and Acceleration Vehicles (CAV)</th>
<th>Non-winged Reentry Vehicles (RV-NW)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mach number range</td>
<td>30-0</td>
<td>0-7 (12)</td>
<td>30-0</td>
</tr>
<tr>
<td>Configuration</td>
<td>blunt</td>
<td>slender</td>
<td>very blunt, blunt</td>
</tr>
<tr>
<td>Flight time</td>
<td>short</td>
<td>long</td>
<td>short</td>
</tr>
<tr>
<td>Angle of attack</td>
<td>large</td>
<td>small</td>
<td>head on</td>
</tr>
<tr>
<td>Drag</td>
<td>large</td>
<td>small</td>
<td>large</td>
</tr>
<tr>
<td>Aerodynamic lift/drag ratio</td>
<td>small</td>
<td>large</td>
<td>small, zero</td>
</tr>
<tr>
<td>Flow field</td>
<td>compressibility effects</td>
<td>viscosity effects</td>
<td>compressibility effects</td>
</tr>
<tr>
<td></td>
<td>dominated</td>
<td>dominated</td>
<td>dominated</td>
</tr>
<tr>
<td>Viscous surface effects</td>
<td>locally important</td>
<td>very important</td>
<td>not important</td>
</tr>
<tr>
<td>Thermo-chemical surface effects</td>
<td>very important</td>
<td>important</td>
<td>very important</td>
</tr>
</tbody>
</table>

1.4 Modeling of Turbulent Flow

Turbulence is one of the most difficult problems in nature and calculating the properties of turbulent flows is being more realistic object with increasing power of computer everyday. Turbulence modeling includes mathematical and numerical modeling in CFD problems to examine turbulence effects in flows. Accuracy, ease of use and cost are criteria in the selection of turbulence model.

1.4.1 Algebraic Models

Algebraic turbulence models are based on Prandtl’s mixing length theory and finding a general definition for the mixing length is important. Boundary layer is divided into inner and outer regions in most of the algebraic models. Smagorinsky [9], Cebeci and Smith [10] and Baldwin-Lomax [11] are algebraic models proposes different relationship between mixing length and eddy viscosity. Mixing length is a geometry dependent property.
1.4.2 One-Equation Models

One-equation turbulence models are based on modeling eddy viscosity. Several one-equation transport models for turbulence have appeared over the years. Baldwin-Barth [12] and Spalart-Allmaras [13] models are the well-known one-equation models. Spalart-Allmaras turbulence model is widely used in time. It is especially useful for aerodynamics flows.

1.4.3 Two-Equation Models

Two-equation turbulence models generally solve two transport equations, one is for turbulent kinetic energy (k) and other is for defining turbulent length scale. The second transport equation is mostly solved for turbulent dissipation (\( \epsilon \)) or turbulent dissipation rate (\( \omega \)). Some of these models is only good for near-wall regions and some of them is good for high Reynolds number flows. All two equation models have their own strong and weak sides. Jones and Launder introduced the k-\( \epsilon \) model in 1972 [14], and it is still very popular in engineering world. Also, different versions of it are applied to many flow problems.

1.4.4 Large Eddy Simulation

The equations represent the large-scale turbulent eddies are directly solved and small-scale eddies are modelled in large eddy simulation (LES). Thus, the computational cost for the small scale motions is very low, compared with DNS. It is more accurate than Reynolds stress models and less accurate than direct numerical simulation (DNS). It can be expected that LES is more successful for flows which involve vortex and unsteady separation. There are four steps in LES. Firstly, the velocity is decomposed into two components: Filtered and subgrid scale (SGS) components. Secondly, Navier-Stokes equations are used to derive the equations for the filtered velocity component. In addition, SGS stress tensor is modeled. It is possible to use an eddy viscosity model to do this. Finally, the filtered equations are solved numerically and one realization of the turbulent flow occurs [15].
1.4.5 Direct Numerical Simulation

Navier-Stokes equations are solved for all time and length scales of flow in direct numerical simulation (DNS). Thus, modeling the turbulence is unnecessary. It is a highly expensive method although it is the most accurate one for the turbulence modelling.

1.5 Literature Survey

Most of turbulence models are used for modeling hypersonic turbulent flow. In this study, Baldwin-Lomax turbulence model is used with three dimensional Navier-Stokes hypersonic flow solver. This turbulence model has some advantages and disadvantages. Baldwin-Lomax model is a zero equation algebraic turbulence model. It yields reasonable results for such applications as boundary layer flow and fully developed pipe or channel flow. However it generally performs poorly for separated flows due to their inability to account for flow history effects. Thus, the modified versions of the Baldwin-Lomax model are also used as well as the original one. The original and modified versions of the Baldwin-Lomax model are also used for the hypersonic applications.

In a study which presents the accuracy of turbulent heating and prediction techniques of skin friction for hypersonic applications, the original and the modified Baldwin-Lomax turbulence model are used with a space marching code. The wall damping formulation is used in the original model and the local damping formulation is used in the modified model. Then, some turbulent predictions are compared with experimental data for flat plates. They give similar results for hot wall conditions, but differ for cold walls. The wall damping heating and skin friction results are 10-30 % above the local damping results and predictions of the local damping have good agreement with the experimental heating data. Also, grid requirements are studied for accurate turbulent heating predictions. The results indicate that grids with a $y^+$ less than 2 are suitable for design of hypersonic vehicles [16].

In the another study examining $k-\omega$ turbulence models in hypersonic boundary layer
applications, it is said that uncorrected two equation models perform progressively worse especially for cold wall cases when Mach number is increased in the hypersonic regime. Baldwin-Lomax model performs better in these circumstances. In addition, after some corrections were made on the \( k-\omega \) model for use with jet or mixing-layer free shear flows and boundary layer flows in hypersonic regime, results of the Baldwin-Lomax model are used as a reference at cold-wall conditions to compare with the corrected \( k-\omega \) model results. It is well-known that Baldwin-Lomax turbulence model can perform reasonably well for attached hypersonic boundary layer flows as other simple algebraic turbulence models do \[17\].

In reference \[18\], some of problems in the design of hypersonic missiles are given. These problems are generally connected with the high levels of heat transfer rate. In reference \[19\], a laminar separated flow around of a cone-flare model is studied when Mach number is 6. Velocity profiles in the region of shock wave-boundary layer interaction are investigated. In both of these studies, laminar-turbulent transition is examined and the Baldwin-Lomax model is used for the turbulent boundary layer calculations. Reasonable results are obtained.

There are also some studies that the Baldwin-Lomax turbulence model is modified for hypersonic flow conditions such as reference \[20\]. The modified Baldwin-Lomax model is used in wedge flow problem at Mach 18 and nozzle flow problem at Mach 20. The Navier-Stokes equations with thin-layer approximation are solved for the given hypersonic flow conditions and the results are compared with the experimental data. The agreement between the numerical solutions and the experimental data are good. Although modified version of the Baldwin-Lomax model is used to compensate weak sides of the original model and it was originally developed for 2D transonic flow with flow separations, it is sometimes used to describe 3D hypersonic flows with massive flow separations \[21\], \[22\].

Overall, the Baldwin-Lomax turbulence model can be generally used for the solution of hypersonic turbulent problems with in a range of reasonable error. According to the feature of the problem, some modifications may be needed on the model or another turbulence model may be used. In this study, only the original Baldwin-Lomax turbulence model is implemented to the hypersonic CFD solver.
Chapter 2 describes flow model including Navier-Stokes equations in cartesian and
generalized coordinates, non-dimensionalization of the equations, how to equations
are discretized numerically, flux vector splitting schemes, boundary conditions and
turbulence modeling. It is mentioned from how Navier-Stokes equations are trans-
formed from cartesian coordinates to generalized coordinates in this part. Also, three
different flux vector splitting schemes and boundary conditions are explained in de-
tail. Finally, Baldwin-Lomax turbulence model is reviewed. Results are explained
and discussed in Chapter 3. Two different reentry geometries are used to obtain the
solutions: Apollo and Atmospheric Reentry Demonstrator(ARD) capsules. It is ex-
plained that how to grid independence study is done for Apollo capsule and compari-
son of the different flux splitting schemes for the computer performance. In addition,
three dimensional flow results are analyzed and discussed for both geometries. Fi-
nally, conclusion and future works are summarized in Chapter 4.
CHAPTER 2

FLOW MODEL

2.1 Introduction

Simulating the fluid motion precisely in the computer environment is a hard process. Complex flow problems can only be solved with complex equations. Solving these equations to model the fluid motion may require high computation time. Therefore, using simplified equations and getting accurate results with reduced CPU times are the driving force of many research that new methods and algorithms are introduced.

In this study, 3-D Navier-Stokes equations are solved. Computational space is transformed from cartesian coordinates to generalized coordinates. In addition, different flux splitting schemes are used.

2.2 Navier-Stokes Equations in Cartesian Coordinates

Three dimensional, steady-state Navier-Stokes equations can be described in the cartesian coordinate system as the following equation:

\[
\frac{\partial (F_c - F_v)}{\partial x} + \frac{\partial (G_c - G_v)}{\partial x} + \frac{\partial (H_c - H_v)}{\partial x} - S = 0
\] (2.1)

\(F_c, G_c\) and \(H_c\) are convective (inviscid) flux vectors and \(F_v, G_v\) and \(H_v\) are viscous flux vectors. In addition, \(S\) is the source vector. Convective flux vectors are defined as:
\[ F_c = \begin{bmatrix} \rho u \\ \rho u^2 + p \\ \rho u v \\ \rho u w \\ (\rho e_t + p)u \end{bmatrix}, \quad G_c = \begin{bmatrix} \rho v \\ \rho v^2 + p \\ \rho v w \\ \rho v w \\ (\rho e_t + p)v \end{bmatrix}, \quad H_c = \begin{bmatrix} \rho w \\ \rho w^2 + p \\ \rho w v \\ \rho w v \\ (\rho e_t + p)v \end{bmatrix} \] (2.2)

where \( u, v \) and \( w \) are velocity components. Viscous flux vectors can be written as follows:

\[ F_v = \begin{bmatrix} 0 \\ \tau_{xx} \\ \tau_{xy} \\ \tau_{xz} \\ u\tau_{xx} + v\tau_{xy} + w\tau_{xz} \end{bmatrix}, \quad G_v = \begin{bmatrix} 0 \\ \tau_{xy} \\ \tau_{yy} \\ \tau_{yz} \\ u\tau_{xy} + v\tau_{yy} + w\tau_{yz} \end{bmatrix} \] (2.3)

\[ H_v = \begin{bmatrix} 0 \\ \tau_{xz} \\ \tau_{yz} \\ \tau_{zz} \\ u\tau_{xz} + v\tau_{yz} + w\tau_{zz} \end{bmatrix}, \quad S = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} \] (2.4)

\( \tau \) indicates the shear stress which is defined as:

\[ \tau_{xx} = \frac{2}{3} \mu \left( 2 \frac{\partial u}{\partial x} - \frac{\partial v}{\partial y} - \frac{\partial w}{\partial z} \right), \quad \tau_{xy} = \mu \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \]

\[ \tau_{yy} = \frac{2}{3} \mu \left( 2 \frac{\partial v}{\partial y} - \frac{\partial u}{\partial x} - \frac{\partial w}{\partial z} \right), \quad \tau_{yz} = \mu \left( \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right) \] (2.5)

\[ \tau_{zz} = \frac{2}{3} \mu \left( 2 \frac{\partial w}{\partial z} - \frac{\partial u}{\partial x} - \frac{\partial v}{\partial y} \right), \quad \tau_{xz} = \mu \left( \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z} \right) \]

\( \mu \) is the dynamic viscosity of the fluid. Remaining equation comes from the state equation for pressure as below:
\[ p = \rho(\gamma - 1) \left[ e - \frac{\rho}{2} \left( u^2 + v^2 + w^2 \right) \right] \quad (2.6) \]

2.3 Transformation

Transformation from physical domain \((x, y, z)\) to computational domain \((\xi, \eta, \zeta)\) increases the efficiency of the computations. Also, boundary conditions are implemented easier. Generalized coordinate transformation is applied to solve the governing equations in computational domain.

The figure 2.1 shows 3-D coordinate transformation between physical and computational domains.

Figure 2.1: Generalized Transformation from the Physical Domain to the Computational Domain

Transformation occurs using the following equations:

\[ \xi = \xi(x, y, z) \]
\[ \eta = \eta(x, y, z) \]
\[ \zeta = \zeta(x, y, z) \quad (2.7) \]

Relation between partial derivatives in computational domain and physical domain is derived by using the chain rule:
\[
\frac{\partial}{\partial x} = \xi_x \frac{\partial}{\partial \xi} + \eta_x \frac{\partial}{\partial \eta} + \zeta_x \frac{\partial}{\partial \zeta}
\]
\[
\frac{\partial}{\partial y} = \xi_y \frac{\partial}{\partial \xi} + \eta_y \frac{\partial}{\partial \eta} + \zeta_y \frac{\partial}{\partial \zeta}
\]
\[
\frac{\partial}{\partial z} = \xi_z \frac{\partial}{\partial \xi} + \eta_z \frac{\partial}{\partial \eta} + \zeta_z \frac{\partial}{\partial \zeta}
\]

(2.8)

\[
\xi, \eta, \zeta, \xi_y, \eta_y, \zeta_y, \ldots, \zeta_z \text{ are the transformation metrics. Differential terms in generalized}
\]

coordinates are defined as:

\[
\frac{\partial \xi}{\partial x} = \xi_x dx + \xi_y dy + \xi_z dz
\]

(2.9)

\[
\frac{\partial \eta}{\partial y} = \eta_x dx + \eta_y dy + \eta_z dz
\]

\[
\frac{\partial \zeta}{\partial z} = \zeta_x dx + \zeta_y dy + \zeta_z dz
\]

The transformation of the partial derivatives between physical space coordinates and

computational domain coordinates are defined with the following equations:

\[
\begin{bmatrix}
\frac{\partial \xi}{\partial \eta} \\
\frac{\partial \eta}{\partial \xi} \\
\frac{\partial \zeta}{\partial \xi}
\end{bmatrix} = \begin{bmatrix}
\xi_x & \xi_y & \xi_z \\
\eta_x & \eta_y & \eta_z \\
\zeta_x & \zeta_y & \zeta_z
\end{bmatrix} \begin{bmatrix}
\frac{\partial x}{\partial \xi} \\
\frac{\partial y}{\partial \eta} \\
\frac{\partial z}{\partial \zeta}
\end{bmatrix} = J \begin{bmatrix}
\frac{\partial x}{\partial \xi} \\
\frac{\partial y}{\partial \eta} \\
\frac{\partial z}{\partial \zeta}
\end{bmatrix}
\]

\[
J = \begin{bmatrix}
x_\xi & x_\eta & x_\zeta \\
y_\xi & y_\eta & y_\zeta \\
z_\xi & z_\eta & z_\zeta
\end{bmatrix} \begin{bmatrix}
\frac{\partial \xi}{\partial \eta} \\
\frac{\partial \eta}{\partial \xi} \\
\frac{\partial \zeta}{\partial \xi}
\end{bmatrix} = \frac{1}{J} \begin{bmatrix}
\frac{\partial \xi}{\partial \eta} \\
\frac{\partial \eta}{\partial \xi} \\
\frac{\partial \zeta}{\partial \xi}
\end{bmatrix}
\]

(2.10)

\[
J = \frac{\partial (\xi, \eta, \zeta)}{\partial (x, y, z)}
\]

\(J\) is the Jacobian matrix. The equation for Jacobian matrix can be rewritten as:
\[
J = \frac{1}{x_\xi(y_\eta z_\zeta - y_\zeta z_\eta) - x_\eta(y_\xi z_\zeta - y_\zeta z_\xi) + x_\zeta(y_\xi z_\eta - y_\eta z_\xi)} \tag{2.11}
\]

Overall, metrics for the transformation from physical space to computational space are found as:

\[
\begin{align*}
\xi_x &= J(y_\eta z_\zeta - y_\zeta z_\eta) \\
\xi_y &= J(x_\xi z_\eta - x_\eta z_\xi) \\
\xi_z &= J(x_\eta y_\zeta - x_\zeta y_\eta) \\
\eta_x &= J(y_\zeta z_\xi - y_\xi z_\zeta) \\
\eta_y &= J(x_\zeta z_\eta - x_\eta z_\zeta) \\
\eta_z &= J(x_\xi y_\zeta - x_\zeta y_\xi) \\
\zeta_x &= J(y_\xi z_\eta - y_\eta z_\xi) \\
\zeta_y &= J(x_\eta z_\xi - x_\xi z_\eta) \\
\zeta_z &= J(x_\xi y_\eta - x_\eta y_\xi) 
\end{align*}
\tag{2.12}
\]

2.4 Non-dimensionalization

Governing equations of fluid flow can be non-dimensionalized and simplified. Units of parameters become unimportant after non-dimensionalization. Thus, flow analysis at different conditions can be implemented efficiently. Reference parameters are defined to non-dimensionalize the Navier-Stokes equations and they can be shown as:

\[
\begin{align*}
x^* &= \frac{x}{L} & y^* &= \frac{y}{L} & z^* &= \frac{z}{L} \\
U_{ref} &= \left(\frac{p_\infty}{\rho_\infty}\right)^{1/2} \\
u^* &= \frac{u}{U_{ref}} & v^* &= \frac{v}{U_{ref}} & w^* &= \frac{w}{U_{ref}} \\
\rho^* &= \frac{\rho}{\rho_\infty} & p^* &= \frac{p}{p_\infty} & \mu^* &= \frac{\mu}{\mu_\infty} & e_t^* &= \frac{e_t}{(p_\infty/\rho_\infty)}
\end{align*}
\tag{2.13}
\]

where L is the characteristic length (length of the vehicle) and subscript \(\infty\) represents free stream values. Reynolds number is defined with the non-dimensionalized parameters and it can be shown as:
\[ Re = \frac{\rho_{\text{ref}} U_{\text{ref}} L}{\mu_{\text{ref}}} = \frac{\rho_{\infty} \left( \frac{p_{\infty}}{\rho_{\infty}} \right)^{1/2}}{\mu_{\infty}} L \]  

(2.14)

Free stream speed of sound is defined in the following equation:

\[ a_{\infty} = \frac{U_{\infty}}{M_{\infty}} = (\gamma R T_{\infty})^{1/2} = \left( \gamma \frac{p_{\infty}}{\rho_{\infty}} \right)^{1/2} \]  

(2.15)

where \( \gamma \) is the specific heat ratio. Using (2.15), the equation (2.14) becomes as:

\[ Re = \frac{\rho_{\infty} \left( \frac{U_{\infty}}{\sqrt{\gamma} M_{\infty}} \right)}{\mu_{\infty}} L = 1 \frac{1}{\sqrt{\gamma} M_{\infty}} \left( \rho_{\infty} U_{\infty} L \right) \]  

(2.16)

Reynolds number in final form can be written as:

\[ Re = \frac{1}{\sqrt{\gamma} M_{\infty}} Re_{\infty} \]  

(2.17)

The equations of motion are non-dimensionalized in the following equations.

**Conversation of mass:**

\[ \frac{\partial}{\partial x^*} \left( \rho^* u^* \right) + \frac{\partial}{\partial y^*} \left( \rho^* v^* \right) + \frac{\partial}{\partial z^*} \left( \rho^* w^* \right) = 0 \]  

(2.18)

**Conversation of momentum (x-direction):**

\[ \frac{\partial}{\partial x^*} \left( \rho^* (u^*)^2 + p^* \right) + \frac{\partial}{\partial y^*} \left( \rho^* u^* v^* \right) + \frac{\partial}{\partial z^*} \left( \rho^* u^* w^* \right) \]

\[ = \frac{\partial}{\partial x^*} (\tau^*_{xx}) + \frac{\partial}{\partial y^*} (\tau^*_{xy}) + \frac{\partial}{\partial z^*} (\tau^*_{xz}) \]  

(2.19)

**Conversation of momentum (y-direction):**

\[ \frac{\partial}{\partial x^*} \left( \rho^* u^* v^* \right) + \frac{\partial}{\partial y^*} \left( \rho^* (v^*)^2 + p^* \right) + \frac{\partial}{\partial z^*} \left( \rho^* v^* w^* \right) \]

\[ = \frac{\partial}{\partial x^*} (\tau^*_{xy}) + \frac{\partial}{\partial y^*} (\tau^*_{yy}) + \frac{\partial}{\partial z^*} (\tau^*_{yz}) \]  

(2.20)
Conversation of momentum (z-direction):

\[
\frac{\partial}{\partial x^*}(\rho^* u^* w^*) + \frac{\partial}{\partial y^*}(\rho^* v^* w^*) + \frac{\partial}{\partial z^*}(\rho^* (w^*)^2 + p^*) = \frac{\partial}{\partial x^*}(\tau^*_{xx}) + \frac{\partial}{\partial y^*}(\tau^*_{yz}) + \frac{\partial}{\partial z^*}(\tau^*_{zz})
\]

(2.21)

Conversation of energy:

\[
\frac{\partial}{\partial x^*}(\rho^* u^* e^* t + p^* u^*) + \frac{\partial}{\partial y^*}(\rho^* v^* e^* t + p^* v^*) + \frac{\partial}{\partial z^*}(\rho^* w^* e^* t + p^* w^*) = \frac{\partial}{\partial x^*}(u^* \tau^*_{xx} + v^* \tau^*_{xy} + w^* \tau^*_{xz}) + \frac{\partial}{\partial y^*}(u^* \tau^*_{xy} + v^* \tau^*_{yy} + w^* \tau^*_{yz}) + \frac{\partial}{\partial z^*}(u^* \tau^*_{xz} + v^* \tau^*_{yz} + w^* \tau^*_{zz})
\]

(2.22)

where shear stresses are non-dimensionalized below:

\[
\begin{align*}
\tau^*_{xx} &= \frac{\sqrt{\gamma M_\infty}}{Re_\infty} \left(2\mu^* \frac{\partial u^*}{\partial x^*} - \lambda^* \nabla^* U^* \right) \\
\tau^*_{xy} &= \frac{\sqrt{\gamma M_\infty}}{Re_\infty} \left(2\mu^* \frac{\partial v^*}{\partial y^*} - \lambda^* \nabla^* U^* \right) \\
\tau^*_{yy} &= \frac{\sqrt{\gamma M_\infty}}{Re_\infty} \left(2\mu^* \frac{\partial v^*}{\partial z^*} - \lambda^* \nabla^* U^* \right) \\
\tau^*_{yz} &= \frac{\sqrt{\gamma M_\infty}}{Re_\infty} \left(2\mu^* \frac{\partial w^*}{\partial y^*} - \lambda^* \nabla^* U^* \right) \\
\tau^*_{zz} &= \frac{\sqrt{\gamma M_\infty}}{Re_\infty} \left(2\mu^* \frac{\partial w^*}{\partial z^*} - \lambda^* \nabla^* U^* \right)
\end{align*}
\]

(2.23)

From now on, the Navier-Stokes equations are represented in non-dimensional form.

### 2.5 Navier-Stokes Equations in Generalized Coordinates

Navier-Stokes equations are derived in generalized coordinates and in conservative form applying equation (2.8) into (2.1):

\[
\frac{\partial (\hat{F}_c - \hat{F}_v)}{\partial \xi} + \frac{\partial (\hat{G}_c - \hat{G}_v)}{\partial \eta} + \frac{\partial (\hat{H}_c - \hat{H}_v)}{\partial \zeta} - \hat{S} = 0
\]

(2.24)

Convective and viscous flux vectors are defined as:
\[
\hat{F}_c = \frac{1}{J} (\xi_x F_c + \xi_y G_c + \xi_z H_c) \quad \hat{F}_v = \frac{1}{J} (\xi_x F_v + \xi_y G_v + \xi_z H_v)
\]
\[
\hat{G}_c = \frac{1}{J} (\eta_x F_c + \eta_y G_c + \eta_z H_c) \quad \hat{G}_v = \frac{1}{J} (\eta_x F_v + \eta_y G_v + \eta_z H_v)
\]
\[
\hat{H}_c = \frac{1}{J} (\zeta_x F_c + \zeta_y G_c + \zeta_z H_c) \quad \hat{H}_v = \frac{1}{J} (\zeta_x F_v + \zeta_y G_v + \zeta_z H_v)
\]

(2.25)

Convective flux vectors are written as:

\[
\hat{F}_c = \frac{1}{J} \begin{bmatrix}
\rho U \\
\rho U u + \xi_x p \\
\rho U v + \xi_y p \\
\rho U w + \xi_z p \\
(\rho e_t + p)U
\end{bmatrix} \quad \hat{G}_c = \frac{1}{J} \begin{bmatrix}
\rho V \\
\rho V u + \eta_x p \\
\rho V v + \eta_y p \\
\rho V w + \eta_z p \\
(\rho e_t + p)V
\end{bmatrix} \quad H_c = \begin{bmatrix}
\rho W \\
\rho W u + \zeta_x p \\
\rho W v + \zeta_y p \\
\rho W w + \zeta_z p \\
(\rho e_t + p)W
\end{bmatrix}
\]

(2.26)

where \( U, V \) and \( W \) are contravariant velocity components:

\[
U = \xi_x u + \xi_y v + \xi_z w
\]
\[
V = \eta_x u + \eta_y v + \eta_z w
\]
\[
W = \zeta_x u + \zeta_y v + \zeta_z w
\]

(2.27)

Viscous flux vectors can be written as:

\[
\hat{F}_v = \frac{1}{J} \begin{bmatrix}
0 \\
\xi_x \tau_{xx} + \xi_y \tau_{xy} + \xi_z \tau_{xz} \\
\xi_x \tau_{xy} + \xi_y \tau_{yy} + \xi_z \tau_{yz} \\
\xi_x \tau_{xz} + \xi_y \tau_{yz} + \xi_z \tau_{zz} \\
\xi_x b_x + \xi_y b_y + \xi_z b_z
\end{bmatrix} \quad \hat{G}_v = \frac{1}{J} \begin{bmatrix}
0 \\
\eta_x \tau_{xx} + \eta_y \tau_{xy} + \eta_z \tau_{xz} \\
\eta_x \tau_{xy} + \eta_y \tau_{yy} + \eta_z \tau_{yz} \\
\eta_x \tau_{xz} + \eta_y \tau_{yz} + \eta_z \tau_{zz} \\
\eta_x b_x + \eta_y b_y + \eta_z b_z
\end{bmatrix}
\]

(2.28)
\[
\hat{H}_v = \frac{1}{J} \begin{bmatrix}
0 \\
\zeta_x \tau_{xx} + \zeta_y \tau_{xy} + \zeta_z \tau_{xz} \\
\zeta_x \tau_{xy} + \zeta_y \tau_{yy} + \zeta_z \tau_{yz} \\
\zeta_x \tau_{xz} + \zeta_y \tau_{yz} + \zeta_z \tau_{zz} \\
\zeta_x b_x + \zeta_y b_y + \zeta_z b_z
\end{bmatrix}
\]  

(2.29)

where,

\[
b_x = u \tau_{xx} + v \tau_{xy} + w \tau_{xz} \\
b_y = u \tau_{xy} + v \tau_{yy} + w \tau_{yz} \\
b_z = u \tau_{xz} + v \tau_{yz} + w \tau_{zz}
\]

(2.30)

and shear stresses can be written as follows:

\[
\tau_{xi,xj} = \frac{\sqrt{\gamma} M_\infty}{Re_\infty} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + \lambda \frac{u_k}{x_k} \delta_{ij} \right]
\]

(2.31)

where \( \lambda \) is the bulk viscosity and it is defined by the stokes hypothesis as in the following equation:

\[
\lambda = -\frac{2}{3} \mu
\]

(2.32)

2.6 Spatial Discretization

CFD codes use a flow domain which solver algorithms work in, and the solver requires grid generation in this domain to do computations. There are two types of grids: structured and unstructured. The unstructured grid generation is usually faster than generation of the structured grid although structured grid calculations usually require less computation time. Because topology of the unstructured grid is irregular and it is have to be stored for each single node. Moreover, the structured grid gives more accurate results for simple single element problems such as airfoil, wing etc.
However, for more complex flows, the structured grid gives more accurate results because it is more flexible. In the present study, structured grid is generated to reduce the computation time and complexity of the numerical algorithm.

Finite volume method is one of the most common approaches for spatial discretization, and it is also used in present study. Computational spaces are divided into discrete control volumes in this method. Flow variables are defined at the cell centers and the fluxes are defined at the cell faces. The fluxes are calculated by using the flow variables according to the spatial discretization.

Three dimensional control volume and flux vectors are shown below.

![Figure 2.2: Flux Vectors and Control Volume](image)

According to the conversation law, generated and leaving flux rates must be equal in the control volume. The fluxes are obtained from flow variable interpolation of left and right cells.

### 2.7 Flux Vector Splitting Schemes

Flux values at cell faces are needed to solve the semi-discrete equation. Different methods are available to do flux calculations. The aim is acquiring the flow variables from the neighboring cells and improving approximation of solution. Flux calcula-
tions are done using upwind scheme in this study. Calculation of inviscid flux is done by applying the flux balance for each cell of the solution domain as:

\[(\delta_\xi \hat{F}_c)_{i,j,k} = (\hat{F}_c)_{i+\frac{1}{2},j,k} - (\hat{F}_c)_{i-\frac{1}{2},j,k} \]  

(2.33)

\(\delta_\xi\) denotes the difference operator. The evaluation of flux values at cell faces is enhanced by using the flow variables from the neighboring cells:

\[
(\hat{F}_c)_{i+\frac{1}{2},j,k} = \hat{F}_c^+ (\hat{W}^-)_{i+\frac{1}{2},j,k} + \hat{F}_c^- (\hat{W}^+)_{i+\frac{1}{2},j,k} \\
(\hat{F}_c)_{i-\frac{1}{2},j,k} = \hat{F}_c^+ (\hat{W}^-)_{i-\frac{1}{2},j,k} + \hat{F}_c^- (\hat{W}^+)_{i-\frac{1}{2},j,k} 
\]

(2.34)

\(\hat{F}_c^+\) and \(\hat{F}_c^-\) are transfer informations respectively from the cell which is to the left (information is spreading from left to right) and right of the interface (information is spreading from right to left). \(\hat{W}^-\) and \(\hat{W}^+\) are the flow variables at the cell center respectively in the left and right side of the interface. The other flux vectors \(\hat{G}_c\) and \(\hat{H}_c\) is discretized similarly. Upwind flux vector splitting can be written as:

\[
[\hat{F}_c^+ (\hat{W}^-)_{i+\frac{1}{2},j,k} + \hat{F}_c^- (\hat{W}^+)_{i+\frac{1}{2},j,k}] - [\hat{F}_c^+ (\hat{W}^-)_{i-\frac{1}{2},j,k} + \hat{F}_c^- (\hat{W}^+)_{i-\frac{1}{2},j,k}] \\
+ [\hat{G}_c^+ (\hat{W}^-)_{i+\frac{1}{2},j,k} + \hat{G}_c^- (\hat{W}^+)_{i+\frac{1}{2},j,k}] - [\hat{G}_c^+ (\hat{W}^-)_{i-\frac{1}{2},j,k} + \hat{G}_c^- (\hat{W}^+)_{i-\frac{1}{2},j,k}] \\
+ [\hat{H}_c^+ (\hat{W}^-)_{i+\frac{1}{2},j,k} + \hat{H}_c^- (\hat{W}^+)_{i+\frac{1}{2},j,k}] - [\hat{H}_c^+ (\hat{W}^-)_{i-\frac{1}{2},j,k} + \hat{H}_c^- (\hat{W}^+)_{i-\frac{1}{2},j,k}] \\
- \hat{S}_{i,j,k} = 0 
\]

(2.35)

There are several flux splitting schemes to split flux vectors. In this study, Van Leer and Steger-Warming schemes are used.

### 2.7.1 Steger-Warming Scheme

Steger-Warming flux splitting scheme was introduced by Steger and Warming [23]. The flux vector is splitted by splitting the eigenvalues of the Jacobian matrix of the flux vector.

\[
\lambda_1 = u \quad \lambda_2 = u + a \quad \lambda_3 = u - a 
\]

(2.36)

The splitted eigenvalues of the Euler equations can be written as:
These eigenvalues are used to obtain splitted flux vectors and the flux vector can be written as:

\[
\hat{F}^\pm_c = \frac{\rho}{2\gamma} \begin{bmatrix}
\beta \\
\beta u + a(\lambda_2^\pm - \lambda_3^\pm)\xi_x \\
\beta v + a(\lambda_2^\pm - \lambda_3^\pm)\xi_y \\
\beta w + a(\lambda_2^\pm - \lambda_3^\pm)\xi_z \\
\beta \left(\frac{u^2 + v^2 + w^2}{2}\right) + aU(\lambda_2^\pm - \lambda_3^\pm) + \frac{a^2(\lambda_2^\pm - \lambda_3^\pm)}{\gamma - 1}
\end{bmatrix}
\]  

(2.38)

where

\[
\beta = 2(\gamma - 1)\lambda_1^\pm + \lambda_2^\pm + \lambda_3^\pm
\]  

(2.39)

Flux vector splitting of Steger-Warming scheme in other directions (\(\hat{G}_c^\pm\) and \(\hat{H}_c^\pm\)) can be found in Appendix A.

2.7.2 Van Leer Scheme

Steger-Warming scheme splits the flux vector with the eigenvalues of the Jacobian. Although it is a commonly used method, using eigenvalues for splitting cause discontinuities at stagnation points and sonic regions because generated fluxes are non-differentiable. A new upwind scheme using Mach number splitting was introduced by Bram Van Leer [24]. Detailed discretization can be seen below:
\[ M = M^+ + M^- \]

\[
M^+ = \begin{cases} 
0 & M \leq -1 \\
\left(\frac{M + 1}{2}\right)^2 & -1 < M < 1 \\
M & M \geq 1 
\end{cases}
\]  \hspace{1cm} (2.40)

\[
M^- = \begin{cases} 
M & M \leq -1 \\
-\left(\frac{M + 1}{2}\right)^2 & -1 < M < 1 \\
0 & M \geq 1 
\end{cases}
\]

All scalar value of Mach number is directed to downstream for the supersonic flow while it is directed to both upstream and downstream for the subsonic flow. Flux vectors can be splitted in supersonic regions as below:

\[
\hat{F}^+ = \hat{F}, \quad \hat{F}^- = 0 \quad M \geq 1 \\
\hat{F}^+ = 0, \quad \hat{F}^- = \hat{F} \quad M \leq -1
\]  \hspace{1cm} (2.41)

Splitted flux vectors can be written for the subsonic region in \( \xi \) direction as follows:

\[
\hat{F}^\pm = \rho a \frac{(M \pm 1)^2}{\gamma} \begin{bmatrix} 
1 \\
\frac{1}{\gamma}(-\hat{U}_\xi \pm 2a)\hat{\xi}_x + u \\
\frac{1}{\gamma}(-\hat{U}_\xi \pm 2a)\hat{\xi}_y + u \\
\frac{1}{\gamma}(-\hat{U}_\xi \pm 2a)\hat{\xi}_z + u \\
\hat{U}_\xi(-\hat{U}_\xi \pm 2a) + 2a \frac{u^2 + v^2 + w^2}{\gamma + 1} + \frac{\gamma u^2 - 1}{\gamma^2 - 1}
\end{bmatrix}
\]  \hspace{1cm} (2.42)

\( a \) is the speed of sound and \( \gamma \) is the specific heat ratio in the above equations. Contravariant velocity \( \hat{U} \) is defined with the directional cosine vectors of \( \hat{\xi}_x, \hat{\xi}_y, \hat{\xi}_z, \hat{\eta}_x, ..., \hat{\zeta}_z \) as the following equation.
\[ \hat{\xi}_i = \frac{\xi_i}{\sqrt{\xi_x^2 + \xi_y^2 + \xi_z^2}} \quad \hat{\eta}_i = \frac{\eta_i}{\sqrt{\eta_x^2 + \eta_y^2 + \eta_z^2}} \]

\[ \hat{\zeta}_i = \frac{\zeta_i}{\sqrt{\zeta_x^2 + \zeta_y^2 + \zeta_z^2}} \]

\[ \hat{U}_\xi = u\hat{\xi}_x + v\hat{\xi}_y + w\hat{\xi}_z \quad \hat{U}_\eta = u\hat{\eta}_x + v\hat{\eta}_y + w\hat{\eta}_z \]

\[ \hat{U}_\zeta = u\hat{\zeta}_x + v\hat{\zeta}_y + w\hat{\zeta}_z \]

Flux vector splitting of Van Leer scheme in other directions ($\hat{G}_{\xi}^\pm$ and $\hat{H}_{\xi}^\pm$) can be found in Appendix A.

### 2.7.3 AUSM Scheme

AUSM (Advection Upstream Splitting Method) is another flux vector splitting method and introduced by Liou and Steffen [25]. It is similar to the Van Leer splitting method. The flux vector is splitted into convective and pressure fluxes.

\[
F_i = \begin{bmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ \rho uw \\ \rho uH \end{bmatrix} = \begin{bmatrix} \rho u \\ \rho u^2 \\ \rho uv \\ \rho uw \\ \rho uH \end{bmatrix} \begin{bmatrix} 0 \\ p \\ 0 \\ 0 \end{bmatrix} \quad (2.44)
\]

The convective flux (the first part) is splitted according to the Mach number value and the pressure flux is splitted according to the pressure value. Right and left Mach number specifications are done similar to Mach number definition of Van Leer as in the equation 2.26. Additionally, the right and left pressure values can be split by two different definitions. The first definition is:
\[ p^\pm = \begin{cases} 
\frac{1}{2} p (1 \pm M) & |M| \leq 1 \\
\frac{1}{2} p \left( \frac{M \pm |M|}{M} \right) & |M| > 1 
\end{cases} \] (2.45)

And the second definition is:

\[ p^\pm = \begin{cases} 
\frac{1}{4} p (M \pm 1)(2 \pm M) & |M| \leq 1 \\
\frac{1}{2} p \left( \frac{M \pm |M|}{M} \right) & |M| > 1 
\end{cases} \] (2.46)

The split flux vector in \( \xi \) direction can be written as:

\[
\hat{F}_c^\pm = M^\pm 
\begin{bmatrix}
\rho a \\
\rho a + u\hat{\xi}_x p^\pm \\
\rho a + v\hat{\xi}_y p^\pm \\
\rho a + w\hat{\xi}_z p^\pm \\
\rho e + (\gamma - 1)(\rho e - \frac{u^2 + v^2 + w^2}{2})
\end{bmatrix}
\] (2.47)

Flux vector splitting of AUSM in other directions (\( \hat{G}_c^\pm \) and \( \hat{H}_c^\pm \)) can be found in Appendix A.

### 2.8 Boundary Conditions

The direction of the propagation of information is have to be taken into consideration when boundary conditions are defined. Also ghost cells are generated to employ proper boundary conditions at the outer part of the grid. One applied ghost cell layer at the outside of computational boundaries enables first order discretization to be implemented accurately. In this study wall, far field and symmetry boundary conditions are used.
2.8.1 Far Field Boundary Condition

Flow information propagates to leave the computational domain for this type boundary conditions. The values of variables in/at ghost cells are defined as free stream values. It can be used when density is calculated using the ideal gas law.

2.8.2 Wall Boundary Condition

Through the wall of the geometry, any fluxes should not pass. In viscous flows, no-slip boundary condition is enforced at the wall. Tangential wall velocities are extrapolated from the interior cells, thus they are equal to tangential fluid velocities. However, normal velocities at the wall surface are set to be zero.

2.8.3 Symmetry Boundary Condition

The symmetry boundary conditions are used to reduce computational effort in problems. The flow field and the geometry is symmetric. Thus, hypersonic flow is solved on/in half of the computational domain. Normal velocity components should be in opposite direction with the same magnitude and tangential velocity components should be equal.

2.9 Turbulence Modeling

One-equation and algebraic Baldwin-Lomax turbulence model is implemented to the hypersonic CFD solver to analyze the turbulent flow.

2.9.1 Baldwin-Lomax Turbulence Model

Baldwin-Lomax turbulence model \(^{[11]}\) is an algebraic model. It is also a form of the outer eddy viscosity and this form do not need knowledge of the conditions at the edge of the boundary layer was developed. It is formulated to be used in applications where the boundary layer thickness, \(\delta\), and displacement thickness, \(\delta_v^*\), are not easily
determined. This model uses two eddy viscosities: inner and an outer eddy viscosity. The inner viscosity is given by

\[ \nu_t = (l_{mix})^2|\omega| \]  

(2.48)

where \( \omega \) is the magnitude of the vorticity vector for three dimensional flows. The mixing length is calculated from the Van-Driest equation as seen below:

\[ l_{mix} = \kappa y[1 - exp(-y^+ / A_0^+)] \]  

(2.49)

The outer viscosity is calculated with the following equation:

\[ \nu_t = \rho \alpha C_{cp} F_{wake} F_{kleb}(y, y_{max}/C_{kleb}) \]  

(2.50)

where

\[ F_{wake} = \min[y_{max} F_{max}, C_{wk} y_{max} U_{dif}^2 / F_{max}] \]  

(2.51)

and

\[ F_{max} = \frac{1}{\kappa} \left[ y_{max} (l_{mix} |\omega|) \right] \]  

(2.52)

This model calculates the outer layer length scale with vorticity. It does not use the displacement or thickness. Thus, it doesn’t need to locate the boundary layer edge.
CHAPTER 3

RESULTS AND DISCUSSION

3.1 Apollo Reentry Vehicle Results

The Apollo mission was designed for landing humans on the Moon and bring them safely back to Earth. Apollo 11 was the first mission that achieved this goal in 1969. The Command Module was spacecraft, along with the Lunar Module, used for the Apollo program. In the present study Apollo AS-202 Command Module is used as reentry vehicle model.

3.1.1 Geometry Modeling and Grid Independence Study

The geometry profile of Apollo command module is shown in Figure 3.1.

Figure 3.1: Dimensions of Apollo AS-202 Capsule (meter)

Three dimensional solid model of the vehicle is shown in Figure 3.2.

O-type structured grid topology is used in construction of the grid. This type of
Grid refinement study is done to remove the mesh dependency on the flow solutions. Three different grids are generated for this purpose. Since $y^+$ parameter is important for the region of boundary layer, it is below 0.5 in all grids and this value is small enough. It is a non-dimensional distance and this is important to determine the proper size of the cells near domain walls in turbulence modeling. Generated grids for grid independence study are seen in Figure 3.3 and dimensions of the generated grids are seen in Table 3.1. The main goal is to use the lowest grid resolution which provides enough accuracy.
Table 3.1: Generated Meshes for Grid Independence Study

<table>
<thead>
<tr>
<th>Grid Resolution</th>
<th>Number of Nodes (i x j x k)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>64 x 48 x 18</td>
</tr>
<tr>
<td>Medium</td>
<td>96 x 72 x 27</td>
</tr>
<tr>
<td>Fine</td>
<td>128 x 96 x 36</td>
</tr>
</tbody>
</table>

The computations are performed on the half solution domain to reduce the computational cost. The present CFD solver is axisymmetrical.

Pressure values are normalized with the maximum pressure value on the nose surface and S/Rb value is the non-dimensionalized value of the nose surface length (S) with respect to body radius (Rb). Normalized pressure values on the nose surface of the Apollo reentry vehicle are used for grid independence study. S and Rb parameters is virtually seen in Figure 3.4. Mach number is 10.18 and Reynolds number is $1.1 \times 10^6$ in freestream flow.

![Figure 3.4: S and Rb parameters](image)

The comparison of normalized pressure values for different grids is seen in Figure 3.5. The value of S/Rb is zero means the center of the nose.

As it can be seen from the Figure 3.5 normalized pressure results of medium and fine grids differ very slightly from each other. Additionally, Table 3.2 shows that CPU time for fine grid is nearly five times longer than the CPU time for medium grid. Therefore using the medium grid is computationally more efficient.
Figure 3.5: Normalized pressure values on the nose surface of Apollo for different grid resolutions at AOA= 0°

Table 3.2: Generated Meshes for Grid Independence Study

<table>
<thead>
<tr>
<th>Grid Resolution</th>
<th>CPU Time (second)</th>
<th>Number of Cells</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>61874</td>
<td>50337</td>
</tr>
<tr>
<td>Medium</td>
<td>308473</td>
<td>175370</td>
</tr>
<tr>
<td>Fine</td>
<td>1402691</td>
<td>422275</td>
</tr>
</tbody>
</table>

Slices through i,j,k directions in three dimensional flow domain which is constructed with the medium grid is seen in Figure 3.6

Figure 3.6: I,J,K planes in 3D flow domain
3.1.2 Comparison of Flux Splitting Schemes

In this part, three different flux splitting schemes are employed for the first order Navier-Stokes solution and residual histories are compared. The aim of this comparison is to determine whether splitting schemes reaches the convergence criteria or not. Also, deciding which scheme converges quicker is another important issue. The residual comparison of Van Leer, AUSM and Steger-Warming splitting schemes is shown in Figure 3.7.

![Figure 3.7: Residual History of Flux Splitting Methods](image)

As it can be seen from the Figure 3.7, fluctuations on the graphs continue until the end of the splines. Also, none of the flux splitting schemes reaches to $10^{-3}$ residual value although computations are done with huge number of iterations, means that none of the schemes reaches the convergence criteria. However, it is clearly seen that the Steger-Warming scheme doesn’t have chance to converge and it shows bad performance. It is also seen that Van Leer scheme has a little lower residual compared to AUSM scheme in same iteration number. The last residual values and CPU times of Van Leer and AUSM schemes are shown in Table 3.3.
Table 3.3: CPU Time and Residual Values for Flux Splitting Schemes

<table>
<thead>
<tr>
<th>Flux Splitting Scheme</th>
<th>CPU Time</th>
<th>Residual Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Van Leer</td>
<td>308473</td>
<td>$0.138 \times 10^{-2}$</td>
</tr>
<tr>
<td>AUSM</td>
<td>384016</td>
<td>$0.159 \times 10^{-2}$</td>
</tr>
</tbody>
</table>

It is understood that from the Table 3.3, Van Leer scheme is more effective than AUSM scheme. Furthermore, the temperature and the density solutions are obtained using Van Leer and AUSM schemes. The comparison of this two schemes is seen in Figure 3.8.

![Figure 3.8: Comparison of Van Leer and AUSM Schemes by Flow Variables](image)

As it can be observed from the Figure 3.8, density of flow is similar for both schemes. However, sudden temperature changes the region at which inside the wake of flow and in front of the nose (shock region) are able to capture by Van Leer scheme accurately. Also, Van Leer method provides more details about the flow compared to AUSM scheme. Therefore, Van Leer flux splitting scheme is employed for 3-D flow results.
In Figure 3.9, non-dimensionalized pressure and u-velocity values along stagnation line are shown at 0 degree angle of attack for Van Leer and AUSM schemes. It is seen that these values are similar for both schemes and don’t make difference to decide which scheme is better.

![Figure 3.9: Comparison of Van Leer and AUSM Schemes by Pressure and U-velocity Values](image)

(a) Pressure  
(b) U-velocity

3.1.3 3D Flow Results

In this part, three dimensional flow results are presented and flow parameters are analyzed on Apollo Capsule. Van Leer scheme is used for flux splitting and Baldwin-Lomax turbulence model is used for the hypersonic turbulent flow analyze. Then turbulent flow results are compared with laminar flow results. Additionally, real gas effects can not be shown in results since ideal gas assumptions are implemented in the present study. In Figure 3.10, present normalized temperature result of hypersonic turbulent flow is compared with another temperature result of simulation of a Mach 16 reentry configuration. In the current study, detached eddy simulation (DES) methodology is used and results of flow field are more realistic. Also, relevant physical phenomena is summarized in the figure.
As it can be seen from the Figure 3.10, almost every feature of the hypersonic physical phenomena including shear layer, wake, expansion and recompression shock is observed in the result of the present study. In Figure 3.11, present normalized density result of hypersonic turbulent flow is compared with a shadowgraph image of a Manned Capsule Concept which was made by NASA.

Description of shadowgraph is given as "A shadowgraph is a process that makes visible the disturbances that occur in a fluid flow at high velocity, in which light passing
through a flowing fluid is refracted by the density gradients in the fluid resulting in bright and dark areas on a screen placed behind the fluid" in reference [28]. In Figure 3.11 it is seen that bow shock and wake regions are caught by the present solver.

Freestream conditions for flows, the results of which take place in this part are seen in Table 3.4.

Table 3.4: Freestream flow values

<table>
<thead>
<tr>
<th>Flow variable</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mach number</td>
<td>10.18</td>
</tr>
<tr>
<td>Reynolds number</td>
<td>$1.1 \times 10^6$</td>
</tr>
</tbody>
</table>

In Figure 3.12 and 3.13 computational normalized pressure values for laminar and turbulent flows along half nose of the body are compared with experimental values respectively at $0^\circ$ and $33^\circ$ angle of attack.

![Normalized Pressure Values](image)

Figure 3.12: Comparison of the Computational and Experimental Normalized Pressure Values along the Body Nose for the Apollo Command Module (AOA = $0^\circ$)

Experimental data is taken from Bertin’s study for the Figure 3.12 and 3.13 [29]. It is
seen that there is a noticeable difference between laminar and turbulent flow results. Pressure values of turbulent flow are higher than laminar flow results as expected. Turbulent flow results fit well with the experimental results at both angle of attack. Additionally, pressure difference between laminar and turbulent flows is maximum at a point that is close to the center (normally at the center) of the nose for 0° angle of attack and a point that is close to shoulder of the nose for 33° angle of attack. In reference [29], it is specified that maximum pressure occurs nearly at \((S/Rb) = 0.95\) point for 33° angle of attack. If the Figure 3.13 is examined, it is seen that maximum pressure difference between laminar and turbulent flow results is nearly at \((S/Rb) = 0.95\) point. This shows that effect of turbulence is encountered more apparent while it is got closer to stagnation point. It can be observed from the both figures that, pressure values decrease suddenly at the center of the body. This sudden change occurs because of singular grid point which is at the center. Normally flux vectors need area or volume to pass the interface for communication between cells and obtained information is used for other flow calculations. However, there is no area or volume along the singular grid point and communication is not healthy there.
This is not an unsolvable problem. Creating a small patch grid cell that fluxes can pass instead of point in that region may be a solution. In addition, the decrease of the values of variables at the nose center is not observed when real gas assumptions are implemented. Also, higher order discretizations and flux limiters can provide another solution for the problem if ideal gas equations are used. For more information, reference [30] can be reviewed.
Figure 3.14: Laminar(left) and Turbulent(right) Flow Results Around Apollo Capsule ( AOA =0° )
Figure 3.15: Turbulent Flow Results at 20° and 33° angle of attack
In Figure 3.14, the distributions of variables are shown. As it can be seen, the differences between the results of laminar and turbulent flows are not clearly identified for mach number, pressure and density variables. However, temperature differences can be observed only at rear part of the vehicle.

Since distribution differences of variables are not understood well; pressure, u-velocity and temperature distributions along the stagnation line are drawn. In Figure 3.16 and 3.17, it is seen that values of variables are nearly same for laminar and turbulent flows along the stagnation line. Thus, turbulence model doesn’t create big difference. However, the reason can of it can be singular point problem at the center mentioned before. In addition, rapid changes are observed in these graphs because front of the shock and rear of the shock are physically different. Pressure and temperature increases through the shock wave. Normally, temperature should reach to peak value at the first point of the shock wave in downstream direction. After that point, it should be decrease a little bit. In contrast, the peak value before the wall can not be seen in Figure 3.17. Reference [31] can be reviewed for a similar type of results.

In Figure 3.15, hypersonic turbulent flow solutions at $20^\circ$ and $33^\circ$ angle of attack are compared. It is seen that stagnation point comes close to shoulder region when AOA increases. Also, the separated flow region and area that vorticities are seen at the rear of the geometry increases.

Numerical values in all figures are the normalized values. Also, real values are very different from what they should be because of using ideal gas equations and not to being included chemical reaction effects in the solver.
In figure 3.18, velocity vectors around the vehicle are shown. Development of the velocity vectors can be noticed and does not contrast with boundary layer physics.
3.2 Atmospheric Reentry Demonstrator (ARD) Results

ARD is an experimental vehicle made by European Space Agency (ESA). Freestream conditions for flows, the results of which take place in this part are seen in Table

<table>
<thead>
<tr>
<th>Flow variable</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mach number</td>
<td>10</td>
</tr>
<tr>
<td>Reynolds number</td>
<td>$1.2 \times 10^6$</td>
</tr>
</tbody>
</table>

3.2.1 3D Flow Results

In this part, limited three dimensional flow results are presented and flow parameters are analyzed on ARD. Van Leer scheme is used for flux splitting and Baldwin-Lomax turbulence model is used for the hypersonic turbulent flow analyze. Then turbulent flow results are compared with laminar flow results.

In Figure 3.19 and 3.20, the geometry of ARD and constructed grid are seen. Since the ARD is also blunt body which is similar to Apollo, critical changes of distribution of the variables are not expected at the surface of the nose. But rear geometry of ARD is different than Apollo. Grid independence study is not be done. Dimensions of the grid is (72 x 48 x 18). Only a general results about comparison of laminar and
turbulent flow solutions and the figures that show some physical results about rear part of ARD take place.

Figure 3.19: Geometry of ARD and 2-D View of its Grid

Figure 3.20: Grid Distribution and 3-D View of ARD Grid (72x48x18)
In Figure 3.21 computational normalized pressure values for laminar and turbulent flows along half nose of the body are compared with experimental values at 20° angle of attack. Experimental data is taken from reference [32].

![Graph showing comparison of computational and experimental normalized pressure values](image)

Figure 3.21: Comparison of the Computational and Experimental Normalized Pressure Values along the Body Nose for the ARD (AOA = 20°)

Contrast to results of Apollo vehicle, normalized pressure values are same for laminar and turbulent flows. The reason is probably inadequate grid resolution. However, general trend of pressure distribution on the nose surface is accurate. Also, some unreal values can be seen around the center of the nose because of singular grid point.

In Figure 3.22 the distribution of variables at 20° angle of attack can be seen. Any change is not be seen for Mach number, pressure and density distributions. However, small changes at the rear part of the geometry are observed for temperature. It seems that high temperatures spread to a wider area in laminar flow.
Figure 3.22: Laminar(left) and Turbulent(right) Flow Results Around ARD
( AOA =0° )
Flux vectors and streamlines at behind of ARD when angle of attack is 20° can be seen in Figure 3.23. Vorticity appears at the top region after flow is separated from the geometry.

![Figure 3.23: Velocity Flux Vectors and Streamlines at the Back Region of ARD (AOA = 20°)](image)

Streamlines and vorticity region at behind of ARD when angle of attack is 0 degree is seen in Figure 3.24.

![Figure 3.24: Streamlines and Vorticity Region around ARD (AOA = 0°)](image)

When results are inspected, it is seen that distributions of variables around ARD are generally like they are desired. However, a finer grid must be used and converged solutions also have to be examined to obtain more accurate results.
CHAPTER 4

CONCLUSION AND FUTURE WORK

The goal of this study is to progress an efficient and accurate CFD solver which can be used in hypersonic flows. Three dimensional Navier-Stokes equations and ideal gas equations are used for the flow analysis. N-S equations are solved with Euler integration. Grid refinement study is done and flow variables are examined around Apollo AS-202 Capsule and Atmospheric Reentry Demonstrator (ARD). Baldwin-Lomax turbulence model is implemented to analyze differences between laminar and turbulent flows.

Different flux splitting schemes are used and they are compared according to convergence, performance and flow results. Then, results of laminar and turbulent flows are compared. It is seen that, normalized pressure results of turbulent flows are bigger than results of laminar flows as expected. However, u-velocity, pressure and temperature results of laminar and turbulent flows are nearly same along the stagnation line. This means that algebraic Baldwin-Lomax turbulence model shows its effect more within the boundary layer. In addition, rapid changes at u-velocity, pressure and temperature values along the stagnation line are investigated. Although u-velocity and pressure values follow the correct trend, temperature don’t reach to pick value at point in front of the shock. Also, ARD is used as another and different geometrical model for validation but, generally distributions of flow variable results are studied due to the limited experimental result.

For future work, some modifications can be done on the solver to show how real gas effects make difference. Additionally, more turbulence model such as Spalart-Allmaras can be implemented to the code and they can be validated. Also, obtaining
some healthier heat flux results provides to observe difference between laminar and turbulent flows better. Also some skin friction calculations can be done if usable experimental and computational data can be found. Using Newton-Gmres method can double performance of the code. Design optimization and parallelization may be other subjects that are worth to be studied on.
REFERENCES


APPENDIX A

FLUX VECTOR SPLITTING METHODS

A.1 Steger-Warming Method

Obtained flux vectors in $\eta$ and $\zeta$ directions are given as:

$$
\hat{G}_c^\pm = \frac{\rho}{2\gamma} \begin{bmatrix}
\beta \\
\beta u + a(\lambda_1^\pm - \lambda_2^\pm) \hat{\eta}_x \\
\beta \nu + a(\lambda_1^\pm - \lambda_2^\pm) \hat{\eta}_y \\
\beta w + a(\lambda_1^\pm - \lambda_2^\pm) \hat{\eta}_z \\
\beta \left( \frac{u^2 + \nu^2 + w^2}{2} \right) + aU(\lambda_1^\pm - \lambda_2^\pm) + \frac{a^2(\lambda_1^\pm - \lambda_2^\pm)}{\gamma - 1}
\end{bmatrix}
$$

(A.1)

$$
\hat{H}_c^\pm = \frac{\rho}{2\gamma} \begin{bmatrix}
\beta \\
\beta u + a(\lambda_2^\pm - \lambda_2^\mp) \hat{\zeta}_x \\
\beta \nu + a(\lambda_2^\pm - \lambda_2^\mp) \hat{\zeta}_y \\
\beta w + a(\lambda_2^\pm - \lambda_2^\mp) \hat{\zeta}_z \\
\beta \left( \frac{u^2 + \nu^2 + w^2}{2} \right) + aU(\lambda_2^\pm - \lambda_2^\mp) + \frac{a^2(\lambda_2^\pm - \lambda_2^\mp)}{\gamma - 1}
\end{bmatrix}
$$

A.2 Van Leer Method

Obtained flux vectors in $\eta$ and $\zeta$ directions are given as:
\[ \hat{G}_c^\pm = \rho a \frac{(M \pm 1)^2}{\gamma} \begin{bmatrix} 1 \\ \frac{1}{\gamma}(-\hat{U}_\eta \pm 2a)\hat{\eta}_x + u \\ \frac{1}{\gamma}(-\hat{U}_\eta \pm 2a)\hat{\eta}_y + u \\ \frac{1}{\gamma}(-\hat{U}_\eta \pm 2a)\hat{\eta}_z + u \\ \frac{\hat{U}_\eta(-\hat{U}_\eta \pm 2a)}{\gamma + 1} + \frac{2a}{\gamma^2 - 1} + \frac{u^2 + v^2 + w^2}{2} \end{bmatrix} \] (A.2)

\[ \hat{H}_c^\pm = \rho a \frac{(M \pm 1)^2}{\gamma} \begin{bmatrix} 1 \\ \frac{1}{\gamma}(-\hat{U}_\zeta \pm 2a)\hat{\zeta}_x + u \\ \frac{1}{\gamma}(-\hat{U}_\zeta \pm 2a)\hat{\zeta}_y + u \\ \frac{1}{\gamma}(-\hat{U}_\zeta \pm 2a)\hat{\zeta}_z + u \\ \frac{\hat{U}_\zeta(-\hat{U}_\zeta \pm 2a)}{\gamma + 1} + \frac{2a}{\gamma^2 - 1} + \frac{u^2 + v^2 + w^2}{2} \end{bmatrix} \] (A.3)

### A.3 AUSM Method

Obtained flux vectors in \( \eta \) and \( \zeta \) directions are given as :

\[ \hat{G}_c^\pm = M^\pm \begin{bmatrix} \rho a \\ \rho a + u\hat{\eta}_x p^\pm \\ \rho a + v\hat{\eta}_y p^\pm \\ \rho a + w\hat{\eta}_z p^\pm \\ a[pe_t + (\gamma - 1)(pe - \frac{u^2 + v^2 + w^2}{2})] \end{bmatrix} \] (A.3)

\[ \hat{H}_c^\pm = M^\pm \begin{bmatrix} \rho a \\ \rho a + u\hat{\zeta}_x p^\pm \\ \rho a + v\hat{\zeta}_y p^\pm \\ \rho a + w\hat{\zeta}_z p^\pm \\ a[pe_t + (\gamma - 1)(pe - \frac{u^2 + v^2 + w^2}{2})] \end{bmatrix} \]