HEAT TRANSFER ACCURACY STUDY FOR HYPERSONIC FLOW USING OVERSET MESH

by

ERIK GUNAWAN WINARDI

ROY P. KOOMULLIL, COMMITTEE CHAIR

GARY CHENG

ROBERT NICHOLS

A THESIS

Submitted to the graduate faculty of The University of Alabama at Birmingham,
in partial fulfillment of the requirements for the degree of

Master of Science

BIRMINGHAM, ALABAMA

2010
EVALUATION OF DIFFERENT FLUX SCHEMES FOR HYPersonic FLOWS
USING GENERALIZED OVERSET MESH
ERIK GUNAWAN WINARDI
MASTER OF SCIENCE IN MECHANICAL ENGINEERING

ABSTRACT

An accurate prediction of aerodynamic heating is critical to the design of a Thermal Protection System (TPS) for hypersonic vehicles. It however remains a challenging task as the majority of Riemann solver based shock capturing schemes suffer from the “carbuncle” phenomenon. A new adaptive scheme approach based on the Riemann solver called HLLC+ was implemented in the in-house computational fluid dynamics solver for generalized meshes to improve the accuracy of solution in hypersonic simulation. Study of different convective fluxes was performed with generalized single mesh approach. In addition, generalized meshes with an overset framework were used to study the effect of numerical flux calculations, mesh topology, overset interpolation approach, and mesh resolution on heat transfer predictions for high speed flows. A library based approach was used for the calculation of domain connectivity information and data communication between overlapping meshes. The results indicated that a high dissipative Riemann solver could prevent the ‘carbuncle’ phenomenon at the expense of solution accuracy at the boundary layer region. The adaptive scheme HLLC+ proved to be an effective approach for preventing the occurrence of carbuncle. Furthermore, the use of a small resolution non-dimensional wall distance ($y^+$) resulted in more accurate heat transfer prediction. It is concluded that both
accurate numerical scheme and non-dimensional wall distance are the most important factors that affect the heat transfer prediction accuracy.
ACKNOWLEDGEMENT

I would like to thank you my advisor Dr. Roy Koomullil for guiding me in learning the fundamental of computational fluid dynamics and giving me the opportunity in doing research in the field. I would like also to thank you my committee members Dr. Gary Cheng and Dr. Robert Nichols for their incredible guidance in finishing my research thesis. I would also want to thank to all the professors and staffs at mechanical engineering department for their patient to assist me in my learning process at the university.

I would like to thank all my friends that I encountered during my study at University of Alabama at Birmingham. Special thanks to Balaji, Nitin, and Sandeep for their invaluable help and guidance during my study. I would also thank to my roommate Il Hwan Kim for his supports and accompany for more than two years in Birmingham.

Finally, I would like to give a special thank to my family. My father, dr. Tjahjanegara Winardi and my mother, dr Lenny Gunawan for all their support and love. I also appreciate the support from my brother, Andreas Winardi, my sister in law Amelia Irawan, and also their daughter Allie for entertaining me since the day she was born last year.
LIST OF TABLES

Table 1.1. Advantages and Disadvantages of Four Meshing Methods. ........................................4
Table 4.1. Initial Condition for Shock Tube Problems. .................................................................30
Table 4.2. Mesh Information for Overset Meshes with Different Topology. .............................63
Table 4.3. Structure-Structure Overset Mesh with Different y+ to the Wall Information. 
..........................................................................................................................................................64
LIST OF FIGURES

Figure 2.1. Types of Flux Splitting Schemes. .................................................................8
Figure 2.2. Shock Dissipation at Cells Differs at Tangential Face (A-B, C-D) and at
Normal Face (A-C, B-D). .............................................................................................9
Figure 2.3. Effect of Cell Aspect Ratio and Carbuncle. ..................................................11
Figure 2.4. Overset Mesh Method Assembly. ..................................................................13
Figure 3.1. Shock-tube Problem ....................................................................................20
Figure 3.2. Riemann Problems at Each Interface ..........................................................21
Figure 3.3. One Intermediate State of HLL and HLLE Riemann Fan............................22
Figure 3.4. Two Intermediate States of The HLLC Riemann Fan .................................24
Figure 4.1. Carbuncle Phenomenon for Flow over a Hypersonic Cylinder. ...............29
Figure 4.2. Comparison of Computed and Exact Solution for Test 1 at Time 0.2. .......32
Figure 4.3. Comparison of Computed and Exact Solution for Test 2 at Time 0.15. ......34
Figure 4.4. Comparison of Computed and Exact Solution for Test 3 at Time 0.012. ......35
Figure 4.5. Comparison of Computed and Exact Solution for Test 4 at Time 0.035. ......37
Figure 4.6. Comparison of Computed and Exact Solution for Test 5 at Time = 0.012....38
Figure 4.7. Results from Mesh Resolution Study using HLLC scheme.......................40
Figure 4.8. Results from Mesh Resolution Study using HLLE scheme. .........................41
Figure 4.9. Results from Mesh Resolution Study using HLLC+ scheme. .......................42
Figure 4.10. Shock Tube Test 4 with 100 Mesh Points in x Direction..............................44
Figure 4.11. Shock Tube Test 4 with 200 Mesh Points in x Direction..............................45
Figure 4.12. Shock Tube Test 4 with 400 Mesh Points in x Direction..............................47
Figure 4.13. Mesh used for Ramp Testcase and the Location of the Line of Interest. .....48
Figure 4.14. Density Distribution for Ramp Testcase ............................................49
Figure 4.15. Density Contour for Combination HLLC+ with Different Delta Values .....50
Figure 4.16. The Density Values Inside the Shock Region .....................................50
Figure 4.17. Density Plot for Different Delta Values for HLLC+ Scheme ................51
Figure 4.18. Density Plot Comparison for HLLE, Roe-HLLE, and HLLC+ Scheme ......51
Figure 4.19. Comparison of Pressure Contour from Simulations using Different Flux Evaluations .................................................................................................................................................................................................53
Figure 4.20. Comparison of Pressure Distribution on the Cylinder Surface ..........54
Figure 4.21. The Location of Region of Interest for Cylinder case ......................55
Figure 4.22. Pressure Distribution across the Line of Interest at 0 degree ..........56
Figure 4.23. Closer Look of Pressure Distribution across the Line of Interest at 0 degree .............................................................................................................................................................................................................................................56
Figure 4.24. Simulation of Moving Shock on Overset Mesh Assembly ..............58
Figure 4.25. Simulations of Shock Wedge Interaction using Overset Meshes ......58
Figure 4.26. Overset Grid Assembly for Structured Background and Body Fitted Meshes .............................................................................................................................................................................................................................................59
Figure 4.27. Overset Mesh with Structured Background Mesh and Hybrid Body Fitted Mesh and Pressure Distribution from the Simulation .............................................................................................................................................................................................................................................61
Figure 4.28. Coefficient of Pressure across the Cylinder Surface for Different Interpolation Scheme .............................................................................................................................................................................................................................................62
Figure 4.29. Pressure Distribution using Overset Meshes with Different Mesh Topologies .............................................................................................................................................................................................................................................63
Figure 4.30. Comparison of Pressure Distribution from Overset Meshes with Different Mesh.

Figure 4.31. Shock/Shock Interactions Experimental Set Up at ONERA.

Figure 4.32. Pressure Contour for Shock/Shock Interaction Study with 20° Shock Generator.

Figure 4.33. Pressure Contour for Shock/Shock Interaction Study with 22° Shock Generator.

Figure 4.34. Close View at the Shock/Shock Interaction Patterns.

Figure 4.35. Pressure Distribution on a Cylinder Surface.

Figure 4.36. Heat Transfer Distribution on a Cylinder Surface.

Figure 5.1. Comparison of Heat Flux Distribution on the Cylinder Surface.

Figure 5.2. Comparison of Heat Flux Predictions from Different Interpolation Techniques.

Figure 5.3. Comparison of Heat Flux Predictions using Overset Meshes with Different Mesh Topologies.

Figure 5.4. Comparison of Heat Flux Distribution from Overset Meshes with Different Mesh Resolution.

Figure 5.5. Temperature Contour for Cylinder Testcase with Single Mesh.

Figure 5.6. Cp Value at the Cylinder Surface.

Figure 5.7. Heat Transfer at the Cylinder Surface.

Figure 5.8. Heat Transfer Contour at Cylinder Surface for Different Flux Schemes.

Figure 5.9. Density Contour at Cylinder Surface for Different Flux Schemes.

Figure 5.10. Pressure Contour at Cylinder Surface for Different Flux Schemes.
# TABLE OF CONTENT

ABSTRACT ................................................................................................................................. ii

ACKNOWLEDGEMENT ............................................................................................................. iv

LIST OF TABLES ....................................................................................................................... v

LIST OF FIGURES .................................................................................................................... vi

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

2. LITERATURE REVIEW ....................................................................................................... 5

2. 1. Hypersonic Flow ............................................................................................................. 5

2. 2. Hypersonic Simulation Challenges ............................................................................... 6

2. 2. 1. Stability of Numerical Scheme (Carbuncle Phenomenon) ..................................... 7

2. 2. 2. Mesh Design, Quality, and Alignment ................................................................. 10

2. 3. Overset Mesh Benefits .................................................................................................. 11

2. 4. Hypersonic Experimental Tests and Ground Facilities .............................................. 14

3. NUMERICAL METHOD ...................................................................................................... 16

3. 1. Governing Equations .................................................................................................... 16

3. 2. Riemann/Shock Tube Problem .................................................................................... 18

3. 2. 1. Roe Scheme ............................................................................................................ 21

3. 2. 2. HLL and HLLE scheme .......................................................................................... 21

3. 2. 3. HLLC scheme ......................................................................................................... 23
3. 3. Adaptive Riemann Solver .................................................................25
3. 4. Higher Order Implementation and Limiter ..............................................27
4.  NUMERICAL RESULTS.................................................................................29
4. 1. Shock Tube Problem. ............................................................................30
4. 2. Mesh Resolution Study for Shock-Tube Problem ...................................39
4. 3. 30° Supersonic Inviscid Ramp .................................................................47
4. 4. Effect of Numerical Flux Evaluation on Solution Accuracy .................52
4. 5. Overset Mesh Interpolation Study using Conserved and Primitive Variables. ......57
4. 6. Overset Mesh Study on a Hypersonic Cylinder .........................................58
4. 6. 1. Effect of Different Overset Mesh Interpolations on Solution Accuracy ......59
4. 6. 2. Effect of Different Overset Mesh Topologies on Solution Accuracy ........62
4. 6. 3. Effect of Different Overset Non-Dimensional Wall Distance on Solution Accuracy ..............................................................................................................63
4. 7. Shock/Shock Interaction in a Laminar Hypersonic Flow ...........................64
5.  CONCLUSION AND FUTURE WORK..........................................................70
5. 1. Future Work ............................................................................................72
LIST OF REFERENCES.........................................................................................79
1. INTRODUCTION

A recent increase in space exploration has boosted interest in analyzing hypersonic problems. Hypersonic technology is currently focused for developing space vehicles with the purpose of space exploration. However, the future of hypersonic technology is not limited just for space exploration, as globalization demands faster public transportation. Flying with a hypersonic airplane at an average speed of Mach 5 allows people to travel from United States to Australia in just 2.5 hours compared to 14 hours required by the fastest commercial airplane currently available. The current necessity was partially fulfilled by the commercial Concorde airplane which was capable of cruising at a supersonic speed of Mach 2. The first hypersonic airplane intended to be used inside the earth’s atmosphere is the X-15 which has the capability to travel at a maximum speed of Mach 6.7.

Hypersonic flow is characterized as a high temperature flow. The high temperature is caused by extreme viscous dissipation within the boundary layer and also by the temperature rise across the strong bow shock. The temperature at the nose region of the Apollo reentry vehicle traveling at Mach 36 could reach up to 11,000 K\(^{[1]}\). For this reason, aerodynamic heat transfer study in hypersonic flows has become an important subject for the development of reliable space vehicles. A reliable thermal protection system (TPS) is required to prevent the space vehicle from melting.
During the last decade, the computational fluid dynamic (CFD) technique has matured enough to be routinely applied in the aerodynamic analysis of supersonic and transonic problems. It is considered as an essential tool in the development of air vehicles. However, a reliable prediction of aerodynamic heat transfer for hypersonic problems still remains a challenge. Most of the shock capturing methods used for transonic problems produce unrealistic solutions when used to solve hypersonic problems\cite{2}. The unrealistic results are mostly caused by the numerical error which dominates the physical solutions. The phenomenon is famously called as ‘carbuncle’ phenomenon. Many cures have been developed for specific cases, but no universal cures have been developed. The need for a universal cure for carbuncle is high and is still a big challenge in the CFD field. A new adaptive scheme approach called HLLC+ was introduced and believed to give accurate solution for hypersonic problems\cite{3}. More details about the carbuncle behavior and several proposed cures will be elaborated in the Literature Review chapter.

The first step in the CFD simulation process is mesh generation. The earliest method developed for mesh generation is the structured mesh. In a structured mesh, the domain of interest is decomposed into a number of quadrilaterals for two dimensions and into hexahedrons for three dimensions. The next method developed is the unstructured mesh where the domain of interest is divided into triangles for two dimensions and into tetrahedrons for three dimensions. Both methods have their own strengths and limitations. The structured mesh method is better in decomposing the boundary layer region because of its capability to produce a good quality of high aspect ratio mesh. Structured mesh method is computationally more efficient than the unstructured mesh
method as the structured mesh method requires less data storage. The mesh connectivity information in the structured mesh method is implicitly determined by the neighbor, whereas, in an unstructured mesh method a data storage is required for all mesh connectivity information. The unstructured mesh however is superior to the structured mesh for complex geometry problems and mesh adaptation approaches. The time required to generate an unstructured mesh for complex geometry is less than that for a structured mesh.

In an attempt to combine the advantages of both the structured and unstructured mesh method, a new meshing method called the generalized mesh has been developed\textsuperscript{[4]}. Generalized mesh is an unstructured mesh composed of arbitrary polyhedrals. It allows the flexibility to generate various mesh shapes. Therefore, it provides the possibility to create high aspect ratio meshes at boundary layer region. The generalized mesh method however still faces difficulty when used for moving body or large deformation problems\textsuperscript{[5,6,7]}. The method will generate skewed meshes in the moving-body or large deformation region. For moving body and large deformation problems, an overset mesh method is considered to be the best approach. An overset mesh method is a method of using more than one mesh to decompose the domain of interest. The meshes are overlapped with each other and are solved separately. An interpolation approach is used to transfer appropriate flow variables between the meshes. By using the overset mesh approach the domain of interest and the environment could be represented in difference meshes which could avoid the generation of skewed meshes.

Another common difficulty in simulating complex fluid flow problems is that not all geometries can be well represented with a single mesh. In many cases, different
geometrical features are best represented by different mesh types. The overset mesh approach helps to solve these difficulties by constructing a mesh system made of overlapping meshes. The complex fluid problem is decomposed into much simpler overlapping meshes and the time required to generate each mesh is reduced. The summary of benefits and limitations for each mesh type are presented in Table 1.1.

The objectives of this research are: 1) to evaluate different convective fluxes in laminar hypersonic problems; 2) to implement a new adaptive scheme, HLLC+, introduced by Trammel, R. et.al.\textsuperscript{[3]} in our generalized mesh based CFD solver, HYB3D\textsuperscript{[4]}; 3) to study the error estimation of heat transfer using the overset mesh method with three different parameters: different interpolation methods, different mesh topologies, and different non-dimensional wall distance.

The thesis is constructed as follows: Chapter one covers the introduction and background. Chapter two covers a brief summary of hypersonic flows and its challenges. Chapter three describes the numerical schemes for the solution of the governing equations. Chapter four discusses the results. Chapter five covers the conclusion and future work.

| Table 1.1. Advantages and Disadvantages of Four Meshing Methods. |
|----------------------------------|----------------|----------------|----------------|----------------|
| Complex Geometry                | -              | +              | +              | +              |
| Mesh Adaptation                 | -              | +              | +              | +              |
| Mesh Generation Time            | -              | +              | +              | +              |
| Memory Requirement              | +              | -              | -              | -              |
| Solver Time                     | +              | -              | -              | -              |
| Viscous Computation             | +              | -              | +              | +              |
| Moving Body Problems            | -              | -              | -              | +              |
2. LITERATURE REVIEW

This section briefly talks about hypersonic flow and the difficulty in achieving accurate prediction in computer simulation. The attempt to get a better hypersonic prediction and a cure for the carbuncle phenomenon is covered in this chapter. The introduction and the potential benefit of the overset mesh approach are also explained.

2.1. Hypersonic Flow

Hypersonic flow is a flow which travels at a velocity that exceeds Mach 5. However, some flows under Mach 5 could produce a characteristic similar to hypersonic flow. The dividing line between supersonic to hypersonic flows is not as obvious as from subsonic to supersonic flow. According to J.D. Anderson, the best interpretation of hypersonic flow is a flow region where the flow phenomena become gradually more important as the Mach number increases\(^{[1]}\). Another indicator of hypersonic flow is that it is characterized by much smaller internal thermodynamic energy than kinetic energy\(^{[1]}\). Hypersonic flow is also characterized as a high temperature flow. The high temperature is caused by extreme viscous dissipation within the viscous boundary layer of the hypersonic flow. In the early 1950s, H. Julian Allen of Ames Aeronautical Laboratory came up with a blunt body design that allows a shock layer region to be created in front of the nose region, to deflect most of the heat away from the vehicle\(^{[8]}\). Even with the
blunt body design, the temperature at the shock layer region of the Apollo during reentry at Mach 36 actually reached up to 11,000 K\textsuperscript{[1]}. Therefore, in order to design reliable hypersonic vehicles, the study of the heat transfer for a hypersonic flow environment is very crucial. Current re-entry vehicles use ablative materials for thermal protection systems. The ablative materials help in transferring the generated heat away from the vehicle, which results in a decrease in temperature at the surface of the re-entry vehicle.

The cost to run a space vehicle prototype in ground experiments is very expensive. Also, there is difficulty in generating exact outer space environment in ground experiments. Knowing the limitations and high cost of conducting ground experiments, an alternative approach is needed to validate the design of hypersonic vehicles. Computer simulation approach has already been used in many airplane designs and proven to give reliable solutions. Thus, using computer simulation to predict the heat transfer in hypersonic flow has become a viable alternative for designing and developing hypersonic vehicles.

2.2. Hypersonic Simulation Challenges

Even with the maturity of computer simulation for subsonic and supersonic problems, simulating a high-quality hypersonic flow is still a challenging task to accomplish\textsuperscript{[9],[10]}. One of the most challenging tasks in hypersonic simulations is to calculate an accurate aerodynamics heat transfer\textsuperscript{[11],[12]}. The complexity of the chemical reaction and the transport phenomena is difficult to model which leads to inaccurate prediction on the heat generated from chemical reactions. Another common challenge is to select an accurate and stable numerical scheme to calculate the solution in hypersonic flows. Most of the high fidelity shock-capturing methods used for subsonic and
supersonic problems suffer from the well-known phenomena called ‘carbuncle’ when used for hypersonic problems\textsuperscript{[2]}. In order to get high-quality prediction of aerodynamic heating, several parameters need to be carefully selected such as: stability and order of accuracy of the scheme used, and mesh design, quality, and alignment to the shock.

2.2.1. Stability of Numerical Scheme (Carbuncle Phenomenon)

The most commonly used shock capturing method for subsonic and supersonic problems is the flux difference splitting scheme also known as the Riemann solver. However, the scheme faces challenges in producing reliable solutions when used to solve hypersonic problems. The pathological behavior was first reported by Peery and Imlay in 1988, during simulations on high speed flows over a cylinder\textsuperscript{[13]}. The pathological behavior is also known as a carbuncle phenomenon and the cause is still not completely understood. It is believed that carbuncle is a phenomenon that occurs when the numerical error in the stagnation region dominates the real physical solution and leads to spurious unphysical solution\textsuperscript{[14]}.

Pandolfi\textsuperscript{[5]} categorized the Flux Difference Splitting method into three types (Figure 2.1): 1) a strong carbuncle prone scheme where pressure perturbations damp out while creating constant density perturbations. 2) A light carbuncle prone scheme where pressure perturbations remain constant and do not interact with the constant density perturbations. 3) A carbuncle-free scheme where density perturbations damp out with or without mutual interaction with pressure perturbations.
Liou\textsuperscript{[15]} hypothesized that the pressure term of a governing equation causes the instability and initializes the numerical error. He stated that in order to cure the instability, the pressure term has to be separated from the governing equation. However, this method is considered non-physical because the pressure also affects the mass balance of the governing equation\textsuperscript{[16]}. In addition, this scheme still suffers from carbuncle.

Ismail\textsuperscript{[16]} hypothesized that carbuncle is divided into three different states: pimple, bleeding, and carbuncle. At the pimple state, the spurious vorticity starts being produced along the shock. Depending on the Mach number, the pimple stage may or may not last long and will develop into the bleeding stage. In the bleeding stage, the spurious result starts to take effect downstream of the shock. The bleeding stage is most visible in the Mach number, velocity, and vorticity contours. The last stage is the carbuncle where the spurious solution dominates all the upstream and downstream conditions of the shock. The behavior is in agreement with flux analysis performed by Dumbser\textsuperscript{[17]}. In his research, he observed that the instability begins in the upstream region and influences the
downstream solution, which explains why the shock-fitting scheme never produces instability on a blunt body problem. Ismail\textsuperscript{[16]} suggests that in order to cure the carbuncle, a scheme has to prevent the creation of the pimple stage. He proposed the use of vorticity control method to prevent this phenomenon.

Both Loh\textsuperscript{[18],[19]} and Xu\textsuperscript{[20],[21]} agreed that the amount of dissipation at the cell face tangential to a shock is much less or closer to absent than the dissipation at a normal cell face. The tangential cell face at A-B and C-D in Figure 2.2 has a very small pressure gradient between its left and right cells. However, the normal cell face A-C and B-D has very large pressure gradients because of the shock properties. They concluded that the Riemann solver does not produce enough dissipation at the tangential faces to the shock; adding numerical dissipation to these faces will provide greater stability. These results are also agreed by Henderson\textsuperscript{[22]}, who studied different tangential face angles and found that the angle with less than a 90 degree tangential face will produce fewer carbuncles.

```
  shock
     |
   A   B
     |
   C   D
```

*Figure 2.2. Shock Dissipation at Cells Differs at Tangential Face (A-B, C-D) and at Normal Face (A-C, B-D).*

Kitamura\textsuperscript{[23]} investigated several methods that suffer from carbuncle and hypothesized that carbuncle is not only a one-dimensional but a multidimensional problem. He found that some schemes that were not stable in one dimension would also
fail in multidimensional problems. He believed that multidimensional dissipation is needed to cure the instability.

Many fixes for the Riemann Solver have been proposed in the last decade but these fixes only work for specific problems, and no universal cure has been agreed upon. Greater understanding of the carbuncle is still being investigated. Two major approaches to cure Riemann solver from the phenomenon are: 1) Adding a numerical dissipation directly to the convective flux calculation part. 2) Using a combination of high and less dissipative schemes by introducing a switching parameter to switch between those schemes. From these two approaches, it is agreed that adding more dissipation to the convective part of the numerical scheme could suppress the phenomenon. The drawback of using a high dissipation scheme is that it will reduce the order of accuracy of the numerical scheme being used.

2. 2. 2. Mesh Design, Quality, and Alignment

In order to have an accurate heat transfer prediction, only preventing the occurrence of carbuncle is not enough. Mesh design, mesh quality, and mesh alignment are also very important factors for solving accurate heat transfer in hypersonic flow. Detailed mesh study for inviscid and viscous hypersonic flows have been performed by Henderson\textsuperscript{[22]}. The effect of the mesh aspect ratio, mesh clustering at the shock location, mesh alignment with the shock structure, and mesh size were studied. Different mesh aspect ratios were studied and the result indicated that the carbuncle was greatly reduced as the aspect ratio increased, as shown in Figure 2.3. The result is in agreement with the finding by Pandolfi\textsuperscript{[2]}. Mesh clustering at the shock location to increase the mesh aspect
ratio has also proven to reduce the carbuncle effect. In order to do mesh clustering, an adaptive mesh capability is needed. Xu and Hu also believe that the numerical dissipation at a cell face is dependent on the orientation of the shock\cite{24}. The more perpendicular a cell face is to the shock, the less numerical dissipation is being produced. No dissipation is produced if a cell face is laid exactly perpendicular to the shock. The same conclusion was also given by Henderson\cite{22}. Henderson also concluded that the mesh misalignment with the shock will produce a greater magnitude of carbuncle, caused by less dissipation when the mesh is misaligned with the shock location.

The effect of mesh design and topology were studied by Gnoffo\cite{10,11} who concluded that a tetrahedral mesh with mesh alignment to the shock still produces spurious solutions. It is suspected that using a tetrahedral mesh will create uncontrollable mesh misalignment and cause the spurious solution. The study using a tetrahedral mesh was also conducted by Candler\cite{12} who also concluded the same results.

\begin{figure}
\centering
\includegraphics[width=0.5\textwidth]{cell_aspect_ratio.png}
\caption{Effect of Cell Aspect Ratio and Carbuncle.}
\end{figure}

\hspace{1cm} a) Low aspect ratio cell produces more carbuncle.
\hspace{1cm} b) High aspect ratio cell produces less carbuncle.

2. 3. Overset Mesh Benefits

Overset mesh is a method that uses a set of overlapping meshes to discretize the domain of interest. The overset mesh method has already been used for more than twenty years to simplify the mesh generation process. The method is also known as Chimera mesh\cite{7}. One of the benefits of using the overset mesh method is that it significantly
reduces the mesh generation time, especially for complex geometries. In the process of design optimization, design changes occur rapidly. When using the overset mesh method, only the parts that are re-designed need to be re-meshed. Thus, the whole design does not need to be created repeatedly which could save a significant amount of mesh generation time. For example, in the shuttle wing design process, there is no need to create a whole new shuttle mesh every time the wing design is changed. The overset mesh method gives the capability to add only the new wing design mesh to the entire overset mesh. The other benefit of overset mesh is that it can be used effectively to perform large body deformations or moving body simulations. By using the overset mesh method, the need to regenerate the mesh for every time step is not required. One example of a moving body simulation which could use the overset mesh method is the store separation simulation.

The development of overset meshes for structured meshes is already in a mature state and is used for many applications, with the application of overset mesh for unstructured and generalized overset meshes rapidly following\cite{10}. This makes the study using the overset mesh method for unstructured and generalized meshes very promising.

The most important step in using overset mesh method is to connect the solution between each mesh. The overlapping meshes in the overset method are connected by appropriate interpolating points between all the meshes. These points are called fringe points. The fringe points that will provide the information to the other vicinity meshes are called donor points. The points that collect the information from the donor are called receptor points. The domain that wants to be excluded from being calculated in order to save computational time is marked as hole points or blank regions. If the receptor points do not have valid donor members, then the area will be marked as orphan points. All this
important point information is collected in a database called the domain connectivity information. Domain connectivity information also includes the information about the corresponding fringe points for each donor and receptor point. A close look at an overset mesh assembly is shown in Figure 2.4. Two individual meshes are used, a body-fitted cylinder mesh and a background mesh. The recipient fringe points for the body-fitted cylinder mesh are colored with black dots. The values at these fringe points are calculated by interpolating the value from the neighboring cells of the background mesh. The cylinder fringe points act as a boundary value for the cylinder mesh. The similar interpolations also happen with the background recipient fringe points. The green dots represent the hole cutting or blank region of the background mesh, where no computational calculation is executed to reduce the computational cost.

Figure 2.4. Overset Mesh Method Assembly.
Most of the legacy computational fluid dynamic solvers are not developed for the overset mesh method. However, the overset capability can be added by appropriate modifications to the CFD solvers. A software called SUGGAR (Structured, Unstructured and Generalized overset Grid AssembleR) has been created by Noack\textsuperscript{[25]} to assemble an overset mesh from single meshes and also to create all the domain connectivity information. The overset mesh generation using SUGGAR is highly automated. Software called DirtLib (Donor interpolation Receptor Transaction Library) has also been developed by Noack\textsuperscript{[26]}. This software envelops all the routine requirements for the overset mesh method into a general CFD solver and performs the overset mesh interpolation to the appropriate location. With the benefit of these two algorithms, an overset mesh assembly can be performed automatically using a single mesh flow solver.

2.4. Hypersonic Experimental Tests and Ground Facilities

Milner\textsuperscript{[27]} had collected hypersonic experimental data from the past decade. Flight experiment is very costly and not much data is available for hypersonic flight tests. Most of the hypersonic experimental data is taken from ground tests inside a wind tunnel. The experimental data that are often used for comparison with the simulation result is acquired from the ground test data, not from the flight test. The most commonly used experimental data for computational benchmark is the hypersonic flow over a cylinder. The test case is commonly chosen because of the simplicity of the geometry and the accuracy of wind tunnel experimental data.

Several groups had performed experimental benchmark cases for hypersonic heat transfer. Calspan University at Buffalo research center (CUBRC) and Office National
d'Études et de Recherches Aérospatiales (ONERA) are among the leaders in performing hypersonic ground experiments for generating benchmark data in their wind tunnels. At CUBRC, the supersonic and hypersonic flows are tested at Large Energy National Shock Tunnel facilities (LENS). The facilities have been used for solving aerodynamics problems including planetary reentry to Earth and Mars. At ONERA, the fundamental studies are mostly performed in the R5Ch wind tunnel. The R5Ch wind tunnel has also been used as a benchmark for some of the advanced experiments performed in other wind tunnels. The benchmark data used in this study were obtained from experimental test cases performed at CUBRC and ONERA.

NATO Research and Technology Organization Advanced Vehicle Technology (RTO-AVT) collected most of the ground test experimental data. For this purpose, they started a panel called “Working Group 10 (WG 10)”. The collected data is very useful for comparing the accuracy of the numerical scheme.
3. NUMERICAL METHOD

Upwind scheme is one of the most commonly used numerical schemes for subsonic and supersonic problems that deal with shock. However, the scheme suffers from spurious solution when used for hypersonic problems. The purpose of this study is to analyze the adaptive scheme of Roe-HLLE and HLLC+ to prevent this spurious solution. This chapter describes: 1) the discretization of the equations governing fluid flows using a finite volume approach, 2) a brief introduction and explanation about Riemann problems and Riemann solvers, and 3) the evaluation and the implementation of convective fluxes using the HLLC+ scheme.

3.1. Governing Equations

The governing equations for a computational fluid dynamics solver are based on three fundamental physical principles: conservation of mass, conservation of momentum, and conservation of energy. These equations can be written in either differential or integral form. The integral form of the governing equations is preferable for solving a fluid flow problem using unstructured mesh because the form allows the use of any elements with an arbitrary number of faces. The integral form of the governing equation can be written as:
\[
\frac{d}{dt} \int_{\Omega} \dot{Q} dv + \oint_{\partial \Omega} \mathbf{F} \cdot \mathbf{n} ds = \oint_{\partial \Omega} \mathbf{F}^v \cdot \mathbf{n} ds
\]

(3.1)

Where: \( \mathbf{n} \) is the unit vector pointing outward from the control surface, \( \partial \Omega \) is the control volume, and \( ds \) is the control surface area.

The conserved variable vector \( \mathbf{Q} \) is defined as:

\[
\begin{bmatrix}
\rho \\
\rho \mathbf{u} \\
\rho \mathbf{v} \\
\rho \mathbf{w} \\
E
\end{bmatrix}
\]

The inviscid flux vector \( \mathbf{F} \) is defined as:

\[
\begin{bmatrix}
\rho \beta \\
\rho u \beta + p n_x \\
\rho v \beta + p n_y \\
\rho w \beta + p n_z \\
\beta(E + p) + p n_t
\end{bmatrix}
\]

The viscous flux vector \( \mathbf{F}^v \) is defined as:

\[
\begin{bmatrix}
f_1^v \\
f_2^v \\
f_3^v \\
f_4^v \\
f_5^v
\end{bmatrix} =
\begin{bmatrix}
0 \\
\tau_{xx} n_x + \tau_{xy} n_y + \tau_{xz} n_z \\
\tau_{yx} n_x + \tau_{yy} n_y + \tau_{yz} n_z \\
\tau_{zx} n_x + \tau_{zy} n_y + \tau_{zz} n_z \\
u f_2^v + v f_3^v + w f_4^v + k(T_x n_x + T_y n_y + T_z n_z)
\end{bmatrix}
\]

\( n_t \) is the contravariant velocity component of the grid speed and is defined as follows:

\( n_t = x_i n_x + y_i n_y + z_i n_z \); \( \beta \) is the contravariant velocity component of the fluid particle with respect to the grid and is defined as \( \beta = \theta - n_t \); where \( \theta = u n_x + v n_y + w n_z \), and \( E \) is the total energy per unit volume and is defined as \( E = \frac{p}{\gamma - 1} + \frac{1}{2} \rho (u^2 + v^2 + w^2) \).

The discretized form of the governing equation can be written as\(^{[28]}\):
\[
\left[ \frac{(1+\theta_2)V^{n+1}_o}{\Delta t} + \frac{\partial \text{RHS}^{n+1,m}_o}{\partial Q^{n+1}_o} \right] \Delta Q^{n+1,m}_o = \\
- \left[ \frac{(1+\theta_2)V^{n+1}_o}{\Delta t} (Q^{n+1,m}_o - Q^n_i) - \theta_2 V^{n-1}_o \Delta Q^{n-1}_o + Q^n_i \text{RHS}^{n+1}_{o,GCL} + \text{RHS}^{n+1,m}_o \right]
\]

where \( V \) is the volume of the cell, the subscript \( i \) represents all the neighboring cells that surround the reference cell \( o \), where \( \theta_2 \) is the parameter for the selection of the time order of accuracy and \( \theta_2=0 \) for first order time and \( \theta_2=1/2 \) for second order time, superscripts \( n \) and \( n+1 \) represent the times steps, superscript \( m \) represents the Newton iteration level, \( \text{RHS}_o \) is the summation of inviscid and viscous fluxes, and \( \text{RHS}_{o,GCL} \) is the flux resulting from mesh deformation.

The discretization of the governing equation described above, has been implemented in a parallel framework for generalized meshes. A CFD solver based on the generalized meshes framework, known as HYB3D, has been developed. The solver has been validated with different benchmark testcases\textsuperscript{[5],[6]}. The framework will be used for the implementation of new flux evaluation schemes.

3.2. Riemann/Shock Tube Problem

Shock-tube problems are the main foundation of the flux difference scheme. In order to understand the mechanism of flux difference splitting scheme, the shock-tube problems need to be understood. Shock-tube problem is a fundamental tool used to study the interaction between waves. It is very beneficial in understanding the hyperbolic partial differential equation such as the Euler equation. Shock-tube problem also gives an
exact solution to some complex nonlinear equations which are useful as the benchmark test for numerical schemes study.

The initial condition of the shock-tube problems is composed of two uniform states separated by a discontinuity membrane, as shown in Figure 3.1.a. The pressure on one side of the membrane is greater than the other side ($P_L > P_R$). All viscous effects are negligible along the tube walls and the tube is assumed to be infinitely long in order to avoid reflections at the tube end. When the discontinuity membrane is instantaneously removed, the pressure discontinuity propagates to the lower pressure region. Simultaneously as the membrane is removed, the discontinuity breaks into two leftward and rightward moving waves separated by a contact discontinuity wave and creates four different regions in the shock-tube, as shown in Figure 3.1.b. The shock divides region one and two which have the discontinuity for density, pressure, and velocity. Region two and three are separated by a contact discontinuity which have a discontinuity in density, but have constant pressure and velocity. In the rarefaction wave region, the fluid properties smoothly change from region three to region four. The solution of the flow field in the shock tube is sketched in Figure 3.1. The shock-tube problem is also known as the Riemann Problem.

<table>
<thead>
<tr>
<th>$P_L$</th>
<th>$P_R$</th>
</tr>
</thead>
</table>

a) Initial Condition of The Shock Tube Problem.
The nature of the finite volume method can be represented as a series of Riemann problems. The value at mesh point i at time level t can be calculated as an average of the properties created by the waves coming from the left and right sides as shown in Figure 3.2. The philosophy of shock-tube problems initiates the birth of exact and approximate Riemann solvers. The exact Riemann solver, also known as the Godunov Method, finds the exact solution of the Riemann problem by considering the speed, direction, and strength of discrete pressure waves, shock waves, and contact discontinuity waves emerging from the cell interface. The exact Riemann solver is more iterative and computationally more expensive than the approximate Riemann solver. The approximate Riemann solver uses approximate solutions to solve the Riemann problem which significantly reduces computational cost. Because of this benefit, the study will use approximate Riemann solvers. Examples of commonly used Riemann solvers are the Roe, HLLC, and HLLE schemes.
3. 2. 1. Roe Scheme

The Roe scheme is the most popular and most widely used approximate Riemann solver because of its accuracy and robustness. The scheme however suffers from the carbuncle phenomenon when used for solving hypersonic problems due to the fact that it captures the contact discontinuity. The assumptions and theory of the Roe scheme\textsuperscript{[34]} are not included in this work. The Roe flux through a cell face is calculated as:

$$ F_{Roe} = \frac{1}{2} \left[ F(Q_L) + F(Q_R) - |\bar{A}| (Q_L - Q_R) \right] $$  \hspace{1cm} (3.5)

Where $|\bar{A}| = T|\Lambda|T^{-1}$. \(T\) is a matrix whose columns are the right eigenvectors of $\bar{A}$, $T^{-1}$ is a matrix with its rows as the left eigenvectors of $\bar{A}$, and $|\Lambda|$ is a diagonal matrix whose elements are the absolute values of the eigenvalues of $\bar{A}$.

3. 2. 2. HLL and HLLE scheme

Harten, Lax, and van Leer proposed a novel approach for solving a Riemann problem which later became known as the HLL scheme. This approach obtains the approximation for the inter cells numerical flux directly. The scheme assumed that two
waves, with speed of $S_L$ and $S_R$, separate three constant values. $S_L$ and $S_R$ are the fastest signal velocities and are assumed to be known. The conserved variables of the HLL scheme are given as:

$$ Q_{HLL} = \begin{cases} Q_R & \text{if } S_L > 0, \\ Q_M = \frac{(F_L - S_L Q_L) - (F_R - S_R Q_R)}{S_R - S_L} & \text{if } S_L < 0 < S_R, \\ Q_R & \text{if } S_R < 0. \end{cases} $$

(3.6)

The Rankine-Hugoniot jump conditions across the left and right waves are used to find flux variable $F_M$, which are $F_M = F_L + S_L(Q_M - Q_L)$ and $F_M = F_R + S_R(Q_M - Q_R)$ and the final expression of the flux variables is:

$$ F_M = S_R F_L - S_L F_R + \frac{S_R S_L}{S_R - S_L} (Q_R - Q_L) $$

(3.7)

$$ F_{HLL} = \begin{cases} F_L & \text{if } S_L \geq 0 \\ F_M & \text{if } S_L \leq 0 \leq S_R \\ F_R & \text{if } S_R \leq 0 \end{cases} $$

(3.8)

![Figure 3.3. One Intermediate State of HLL and HLLE Riemann Fan.](image)

While the HLL scheme is very efficient and robust, the two waves assumption is only accurate for hyperbolic systems of two equations, as shown in Figure 3.3. Determining the wave speeds is very crucial to have stable and not overly dissipative
solutions when using the HLL scheme. To stabilize the scheme, Einfeldt et al\textsuperscript{36} proposed a way to determine the wave speed as follow:

\[ S_L = \min(0, \bar{u} - \bar{a}, (u - a)_L) \]  
\[ S_R = \max(0, \bar{u} + \bar{a}, (u + a)_R) \]  

Where \( \bar{u} \) and \( \bar{a} \) are the Roe-averaged properties and speed of sound value respectively.

The HLL scheme that uses the wave speed criteria as Einfeldt et al\textsuperscript{36} proposed is better known as the HLLE scheme. The use of the wave speed helps to ensure the method to be positively conservative and prevent the HLLE scheme from generating a vacuum state, which makes it more stable than the original HLL scheme. Because the HLLE scheme does not take the contact discontinuity into account, the scheme does not suffer from the carbuncle phenomenon. However the scheme is known to have excessive dissipation at the boundary layers.

3.2.3. HLLC scheme

The HLLC scheme, where C stands for Contact Wave, tries to restore the missing contact discontinuity and shear waves which the HLL scheme ignore. The contact discontinuity waves speed is represented by \( S_M \) and is calculated as

\[ S_M = \frac{\rho_L (S_L - u_L) - \rho_R u_R (S_R - u_R) + P_R - P_L}{\rho_R (S_R - u_R) - \rho_L (S_L - u_L)} \]  

Where \( S_L = \min((\bar{u} - \bar{a}),(u - a)_L) \) and \( S_R = \max((\bar{u} + \bar{a}),(u + a)_R) \).

In the HLLC scheme the intermediate state is divided into two separate regions, as shown in Figure 34. The integration over the control volume for these regions gives two integral averages \( Q^*_L \) and \( Q^*_R \) and the relation of these values with \( Q_M \) can be found using the HLL scheme.
The Rankine-Hugoniot jump condition is used across each wave speed \((S_L, S_M, S_R)\). With the fact that the pressure and velocity component normal to the contact waves are constant, the intermediate state of conserved variable and fluxes values are solved (equations 3.9 – 3.10). The Rankine-Hugoniot jump conditions are:

\[
F_L^* = F_L + S_L(Q_L^* - Q_L), \quad F_M^* = F_M + S_M(Q_M^* - Q_M), \quad \text{and} \quad F_R^* = F_R + S_R(Q_R^* - Q_R)
\]

\[
Q_M = \left( \frac{S_M - S_L}{S_R - S_L} \right) Q_L + \left( \frac{S_R - S_M}{S_R - S_L} \right) Q_R^* \tag{3.12}
\]

![Figure 3.4. Two Intermediate States of The HLLC Riemann Fan.](image)

\[
Q_K^* = \begin{bmatrix}
\rho_L \frac{S_K - q_K}{S_K - S_M} \\
\frac{u_K + (S_M - q_K) \eta_x}{S_K - S_M} \\
\frac{v_K + (S_M - q_K) \eta_y}{S_K - S_M} \\
\frac{w_K + (S_M - q_K) \eta_z}{S_K - S_M} \\
E_K + (S_M - q_K) \left( S_M + \frac{P_K}{\rho_K(S_K - q_K)} \right)
\end{bmatrix}
\tag{3.13}
\]

Where \(K = L\) or \(K = R\)

\[
F_{HLLC} = \begin{cases}
F_L & \text{if } S_L > 0 \\
F_L^* = F_L + S_L(Q_L^* - Q_L) & \text{if } S_L \leq 0 < S_M \\
F_R^* = F_R + S_R(Q_R^* - Q_R) & \text{if } S_M \leq 0 < S_R \\
F_R & \text{if } S_R < 0
\end{cases}
\tag{3.14}
\]
The HLLC scheme offers more accurate and less dissipative predictions at the boundary layers than the HLL/HLLE scheme. However, the scheme is vulnerable to the carbuncle phenomenon because it captures the contact discontinuity exactly.

3.3. Adaptive Riemann Solver

Many numerical schemes have been proposed in the literature to combine different numerical schemes to avoid carbuncle phenomenon and to get better accurate results. This class of schemes is called adaptive schemes. One of the adaptive schemes that is present in the current numerical framework is the adaptive Roe-HLLE scheme.

One of the methods to cure the carbuncle phenomenon is by using a combination of Riemann solver. A dissipative Riemann solver does not suffer from the phenomenon. However, a Riemann solver has to have just enough dissipation to calculate the prediction of shock location and viscous boundary layer accurately. The idea of using a switching Riemann Solver was proposed by Quirk\cite{14} in 1992. He suggested using a combination of a high and low dissipation Riemann Solver to prevent the shock instability and to have the ability in capturing the shock location exactly. He proposed the idea of using a pressure parameter to specify a strong shock region. The switching parameter is based on a pressure jump between the reference cell and its neighbors and is defined as,

\[
\left| \frac{p_R - p_L}{\min(p_L, p_R)} \right| > \alpha, \text{ where } \alpha = \text{threshold parameter} \tag{3.15}
\]

The threshold parameter is problem dependant and can be adjusted according to how much dissipation is needed. Quirk\cite{14} chose to switch between the Roe and HLLE methods. Roe scheme was used in no strong shock region and HLLE scheme was used when the condition of Equation 3.15 was satisfied. Even though the HLLE scheme is
believed to be very dissipative, the use of this scheme in the vicinity of a strong shock will not reduce the total scheme accuracy. The use of the Roe-HLLE adaptive scheme gives smoother and more realistic conserved variables contour than using Roe or HLLE scheme alone.

Another adaptive scheme method was proposed by Trammel et al.\textsuperscript{[3]} which used the combination of HLLC and HLLE Riemann solvers. The approach is called the HLLC+ approach. The idea of using an adaptive scheme is to have an accurate solution without the need of generating shock-alignment mesh. The performance of the HLLC+ approach gives a better solution than the original HLLC or HLLE scheme and also maintains the time accuracy\textsuperscript{[3]}. The implementation of the HLLC+ flux calculation for generalized mesh is:

\[
F_{HLLC+} = \beta_{HLLC+} F_{HLLC} + (1 - \beta) F_{HLLE}
\]

(3.16)

\[
\beta_{HLLC+} = \max(\beta_{\text{New}}, 0.4)
\]

(3.17)

\[
\beta_{\text{New}}\begin{cases} 0.0 & \text{if } S_w > 1.0/\text{Delta} \\ 1 - \tanh(10\varphi_p^3) & \text{otherwise} \end{cases}
\]

(3.18)

\[
\varphi_p = \max(S_w/\text{Delta}, 0)
\]

(3.19)

\[
S_w = \max(k_L, k_R)
\]

(3.20)

\[
k = \frac{|p_R - p_L|}{\min(p_L, p_R)}
\]

(3.21)

The Delta value of 1 to 10 has been found to be adequate to eliminate the carbuncles and errors induced when strong shocks are present. \(F_{HLLC}\) and \(F_{HLLE}\) are calculated using the flux calculation of HLLC and HLLE scheme respectively. The value of \(\beta_{HLLC+}\) is used to limit the high dissipation produced by HLLE. \(\beta_{\text{NEW}}\) is the switching
parameter. $k$ is the pressure switch sensor and it is in first order accurate. The original HLLC+ formulation is generated for structure mesh solver and the pressure switch is based on the second derivative of pressure, while the current implementation is based on the first derivative of pressure.

The HLLC+ scheme is implemented for a generalized mesh framework and the results from the validation of the scheme is given in the next chapter. Both adaptive scheme Roe-HLLE and HLLC+ will be used in this study to validate the overset mesh methodology.

3.4. Higher Order Implementation and Limiter

In order to have a better solution representation, second order linear reconstruction from the cell averaged values is applied. The piecewise linear reconstruction of the conserved variables from the cell averaged values is calculated using the Taylor’s series expansion from all the neighboring cells. During the solution process, the values at the cell face are extrapolated using the cell averaged values.

The practice of getting a high order solution by adding the gradient of flow variables could produce local minima and local maxima. The creation of local minima or local maxima will induce a high chance of creating spurious oscillations of the flow variables near regions of the high solution gradient. The limiter function is used to prevent the development of the spurious oscillations and also to preserve the monotonicity. The limiter function is applied directly to the gradient of the flow variables in the Taylor’s series expansion. The disadvantage of using the limiter function is that it will reduce the order of accuracy of the scheme used in those regions. The limiter used in
this study is the limiter function introduced by Barth\textsuperscript{[29]}. The limiter condition is shown in equations 3.22 and 3.23. $Q_L$ is the neighboring cell and $Q_C$ is the value at the cell center.

$$\phi_{Barth} = \min(\phi_F)$$  

(3.22)

$$\phi_F = \begin{cases} 
\min \left( 1, \frac{Q_L^{max} - Q_C}{Q_L - Q_C} \right) & \text{if } Q_L - Q_C > 0 \\
\min \left( 1, \frac{Q_L^{min} - Q_C}{Q_L - Q_C} \right) & \text{if } Q_L - Q_C < 0 \\
1 & \text{if } Q_L - Q_C = 0 
\end{cases}$$  

(3.23)

where $Q_{cell}^{\min} = \min(Q_C, Q_n)$ and $Q_{cell}^{\max} = \max(Q_C, Q_n)$
4. NUMERICAL RESULTS

The carbuncle phenomenon occurs when a Riemann solver tries to capture the contact discontinuity exactly. An example of carbuncle phenomenon is shown in Figure 4.1. The numerical error dominates the physical solution which creates an unphysical and spurious solution. In order to prevent the carbuncle phenomenon, two types of adaptive scheme fixes were investigated: the adaptive Roe – HLLE and HLLC+. Laminar hypersonic flows of calorically and thermally perfect gases without chemical reaction were used for investigating the accuracy of these numerical schemes.

![Mesh used for Hypersonic Cylinder testcase.](image1)

![Carbuncle Phenomenon](image2)

Figure 4.1. Carbuncle Phenomenon for Flow over a Hypersonic Cylinder.
4. 1. Shock Tube Problem.

To validate the implementation of HLLC, HLLE, and HLLC+ schemes for the solution of the Riemann problem, shock tube problems with different initial conditions were used. Toro\cite{30} provides several initial value problems to investigate the accuracy and robustness of a Riemann solver. These initial conditions are tabulated in Table 4.1, where $X_o$ is the location of the initial discontinuity. The spatial domain of the shock tube problem is taken as $0 \leq x \leq 1$. The numerical solution is computed with a three dimensional structured mesh of dimension $100 \times 10 \times 10$ in $x$, $y$, and $z$ directions respectively. A CFL number of 0.9 and the specific gas constant of 1.4 are used for these simulations.

<table>
<thead>
<tr>
<th>Test</th>
<th>$\rho_L$</th>
<th>$u_L$</th>
<th>$P_L$</th>
<th>$\rho_R$</th>
<th>$u_R$</th>
<th>$P_R$</th>
<th>$X_o$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.0</td>
<td>0.75</td>
<td>1.0</td>
<td>0.125</td>
<td>0.0</td>
<td>0.1</td>
<td>0.3</td>
</tr>
<tr>
<td>2</td>
<td>1.0</td>
<td>-2.0</td>
<td>0.4</td>
<td>1.0</td>
<td>2.0</td>
<td>0.4</td>
<td>0.5</td>
</tr>
<tr>
<td>3</td>
<td>1.0</td>
<td>0.0</td>
<td>1000.0</td>
<td>1.0</td>
<td>0.0</td>
<td>0.01</td>
<td>0.5</td>
</tr>
<tr>
<td>4</td>
<td>5.99924</td>
<td>19.5975</td>
<td>460.894</td>
<td>5.99242</td>
<td>-6.19633</td>
<td>46.0950</td>
<td>0.4</td>
</tr>
<tr>
<td>5</td>
<td>1.0</td>
<td>-19.59745</td>
<td>1000.0</td>
<td>1.0</td>
<td>-19.5975</td>
<td>0.01</td>
<td>0.8</td>
</tr>
</tbody>
</table>

Test 1 has a shock wave and contact discontinuity traveling to the right and rarefaction waves traveling to the left. The test is useful in assessing the entropy satisfaction property of numerical methods. Test 2 is suitable for assessing the performance of numerical methods for low-density flows. Test 3 has a strong shock and is designed to assess the robustness and accuracy of numerical methods. Test 4 has three different discontinuities moving to the right and is also designed to assess the ability of the numerical methods to resolve slow moving contact discontinuity. Test 5 consists of rarefaction waves traveling to the left and shock traveling to the right with a stationary contact discontinuity. The comparison of the numerical results with the exact results for
these five shock tube problems are shown in figure 4.2 – 4.6. It can be seen from these figures that HLLC, HLLE, and HLLC+ with the value Delta of 5 predicts the results accurately.

As shown in Figure 4.2, HLLC+ and HLLE scheme predict a better velocity profile at the shock discontinuity than HLLC scheme for test 1. In test 2 and test 3, Figure 4.3 and Figure 4.4, all three schemes give similar solutions. For test 4, Figure 4.5 the HLLC+ scheme gives a better shock discontinuity prediction than the other two schemes. Also in test 5, Figure 4.6 the HLLC+ gives more accurate value in locating the shock discontinuity than HLLC or HLLE.
Figure 4.2. Comparison of Computed and Exact Solution for Test 1 at Time 0.2.
a) Density

b) Pressure
Figure 4.3. Comparison of Computed and Exact Solution for Test 2 at Time 0.15.
Figure 4.4. Comparison of Computed and Exact Solution for Test 3 at Time 0.012.
a) Density

b) Pressure
Figure 4.5. Comparison of Computed and Exact Solution for Test 4 at Time 0.035.

c) Velocity

a) Density
Figure 4.6. Comparison of Computed and Exact Solution for Test 5 at Time = 0.012.
4.2. **Mesh Resolution Study for Shock-Tube Problem**

A study of effect of mesh resolution to the solution of shock tube problem has been performed. In this study shock-tube problem 4 has been selected as the benchmark case because it has the strongest shock and a slow moving contact discontinuity. Another reason for selecting this testcase for the mesh resolution study is that the numerical results from all other four cases behaved similarly with the exact solution as compared to test 4. Three different meshes of sizes 100x10x10, 200x10x10, and 400x10x10 are used for this study. The results from the study are presented in Figure 4.7 to Figure 4.12 for HLLC, HLLE, and HLLC+. In general, as the mesh resolution increases the contact discontinuity and the shockwave are predicted with better accuracy. From Figure 4.7 to Figure 4.12, it can be seen that HLLE scheme gives the smoothest density, pressure, and velocity contours while the HLLC scheme gives the most spurious contours. The HLLE scheme also gives the best prediction of shock and contact discontinuity locations with the lowest dissipation.

![Density Contour](image-url)
Figure 4.7. Results from Mesh Resolution Study using HLLC scheme.

a) Density Contour.

b) Pressure Contour

c) Velocity Contour
Figure 4.8. Results from Mesh Resolution Study using HLLE scheme.

a) Density Contour.

b) Pressure Contour.

c) Velocity Contour.
Figure 4.9. Results from Mesh Resolution Study using HLL+C scheme.
a) Density Plot.

b) Pressure Plot.
C) Velocity Plot.

Figure 4.10. Shock Tube Test 4 with 100 Mesh Points in $x$ Direction.

a) Density Plot.
Figure 4.11. Shock Tube Test 4 with 200 Mesh Points in x Direction.
a) Density Plot.

b) Pressure Plot.
Figure 4.12. Shock Tube Test 4 with 400 Mesh Points in $x$ Direction.

4.3. 30° Supersonic Inviscid Ramp

A simple case that can demonstrate the problem with a Riemann solver is a supersonic inviscid flow past a 30° ramp with the freestream Mach number of 4.0. A three-dimensional mesh with dimension of 58 x 120 x 2, as shown in Figure 4.13, is used for the simulation. The ramp will produce an oblique shock which theoretically should give smooth density values behind the oblique shock.
Figure 4.13. Mesh used for Ramp Testcase and the Location of the Line of Interest.

Four different Riemann solvers and four different Delta values for the HLLC+ scheme were used to solve the inviscid ramp testcase. Figure 4.14 and Figure 4.15 show the density contours from the simulations using these different numerical schemes. The density distributions along the line of interest, shown by a red line in Figure 4.13, are depicted in Error! Reference source not found. Figure 4.16 and Figure 4.17. These figures have been focused at the inside of the region between the shock and the wedge to clearly see the variation of density. It can be seen from Figure 4.16 Error! Reference source not found. that the predicted density distributions for HLLC and Roe scheme give the most disturbances behind the shock region. Also, it can be seen that HLLE and adaptive Roe-HLLE give a smoother density oscillation in that region. The results from the HLLC+ scheme different Delta values are shown in Figure 4.17. The lowest value of Delta gives the most density disturbances because of less contribution of HLLE flux for lower value of Delta. The HLLE flux is more dissipative than HLLC scheme and therefore less dampening effects for lower value of Delta. As the value of Delta
increased, the density distribution became smoother. This shows that the more dissipation from HLLE scheme, the error inside the shock region is reduced. Also, it can be seen from Figure 4.17 that increasing Delta more that 10 does not have any effect in dampening out the oscillations.

Figure 4.14. Density Distribution for Ramp Testcase.
Figure 4.15. Density Contour for Combination HLLC+ with Different Delta Values.

Figure 4.16. The Density Values Inside the Shock Region.
Figure 4.17. Density Plot for Different Delta Values for HLLC+ Scheme.

Figure 4.18. Density Plot Comparison for HLLE, Roe-HLLE, and HLLC+ Scheme.
Figure 4.18 shows a comparison between HLLE, adaptive Roe-HLLE, and HLLC+ with a Delta of 20. HLLE scheme gives the smoothest density distribution and also confirms that adding dissipation could remove a spurious solution inside a strong shock.

4.4 Effect of Numerical Flux Evaluation on Solution Accuracy

A laminar hypersonic flow past a cylinder is used as a benchmark testcase to investigate carbuncle phenomenon for different Riemann solvers. The details of the experimental data is available in Holden et al.\textsuperscript{[31]}. The freestream flow Mach number, temperature, Reynolds number per meter, and specific heat for this testcase are 16.01, 43.2 K, 91,100, and 1.4 respectively. The effect of numerical schemes on the solution accuracy is discussed below. A three dimensional single block mesh with 40,542 nodes and 25,704 cells was used for the simulation. The single block mesh was used for the study to avoid any uncertainties associated with overset meshes. The simulations were carried out using second order spatial discretization and the results of the predicted pressure contours for the seven different numerical schemes are presented in Figure 4.19.
Figure 4.19. Comparison of Pressure Contour from Simulations using Different Flux Evaluations.

It can be seen from the figure that the Roe and HLLC scheme produces spurious predictions, while HLL, HLLE, and van Leer giving a more reasonable contour. Both the adaptive Roe-HLLE and HLLC+ predict a smooth variation of the pressure without
carbuncle. Figure 4.20 compares the computed coefficient of pressure ($C_p$) distribution on the cylinder surface using different flux evaluation with the modified Newtonian theory\textsuperscript{[11]}. Most of the numerical schemes used predict a good agreement with the modified Newtonian theory, except for the Roe scheme. The deviation in the Roe scheme is due to the presence of the carbuncle in front of the cylinder. HLL, HLLC, and van Leer schemes show a little disturbances around the stagnation region of the cylinder. The adaptive scheme Roe-HLLE and HLLC$+$ predicted the pressure distribution with the least amount of disturbances at the stagnation region.

![Figure 4.20. Comparison of Pressure Distribution on the Cylinder Surface.](image)

The comparison of pressure distributions along the stagnation streamline are compared below. The location of the line of interest is shown in Figure 4.21. The
pressure distribution results from all the schemes are shown in Figure 4.22 and it can be seen that only the Roe scheme gives the incorrect shock location while the other schemes predict a similar shock location. Closer views of the shock location are given in Figure 4.23. As it is shown in the figure, the order from highest to the lowest dissipation results at representing the shock are: HLLC, Van Leer, Roe-HLLE, HLLC+, HLLE, and HLL.

Figure 4.21. The Location of Region of Interest for Cylinder case.
Figure 4.22. Pressure Distribution across the Line of Interest at 0 degree

Figure 4.23. Closer Look of Pressure Distribution across the Line of Interest at 0 degree.
The heat transfer predictions for these fluxes were also studied. The setting of the boundary condition at the cylinder surface is taken as an isothermal surface with the freestream temperature. Thus, the prediction of heat flux was miscalculated because of the wrong wall temperature used for the simulation as compared to the experimental setup. Because of the implementation of the correct temperature at the cylinder surface needed to be revisited, the preliminary results are given in the future work section in Chapter 5.

4.5. **Overset Mesh Interpolation Study using Conserved and Primitive Variables.**

Before applying the overset mesh method to solve hypersonic problems, the interpolation method for the overset mesh framework needs to be validated. A simple case of a shock moving across a region of overset mesh is used for studying the overset interpolation method. The mesh used for the study is shown in Figure 4.24. The conserved variable interpolation resulted in spurious values at the interface region resulting in a non-physical solution as shown in Figure 4.24b. The result from primitive variable interpolation for the transfer of information across overlapping region is shown in Figure 4.24c. It can be observed that when interpolating with the primitive variables, the shock moves through the overset region without any non-physical disturbances.

Another testcase was studied for the evaluation of interpolation accuracy is a flow over a 45° wedge\[^{37}\]. The mesh used for this simulation is shown in Figure 4.25. The overset mesh interpolation method using conserved variables gave spurious unphysical values at the fringes points as shown in Figure 4.25b. With primitive variable
interpolation, the values at the fringes points become smooth and realistic as shown in Figure 4.25c. Therefore the overset simulations presented in the following sections are carried out using primitive variable interpolation.

4.6. Overset Mesh Study on a Hypersonic Cylinder

From the results of single mesh, the adaptive scheme Roe-HLLE and HLLC+ give promising result in preventing the carbuncle and provide realistic pressure predictions. The Roe-HLLE scheme was used for study the effect of interpolation
schemes, mesh topology, and mesh resolution on solution accuracy. The hypersonic cylinder test case described in Section 4.4 is used for this study also. A detailed view of one of the meshes used for this study together with the fringe points and blanked points is shown in Figure 4.26. In the figure the mesh points tagged as black squares represent the fringe points for the cylinder mesh where the variables are interpolated from the background mesh. The mesh points tagged as red squares represent the fringe points where for the background mesh the variables are interpolated from the cylinder mesh. Mesh points tagged as green squares represent blanked cells in the background mesh, where the computations are not carried out.

![Diagram of overset grid assembly](image)

**a) Overset Assembly**  
**b) Close View of Overset Assembly**

*Figure 4.26. Overset Grid Assembly for Structured Background and Body Fitted Meshes.*

4.6.1. Effect of Different Overset Mesh Interpolations on Solution Accuracy

An accurate interpolation of information between meshes in the overlapping region is the most crucial process in the overset mesh method. The influence of different overset mesh interpolation method on the accuracy of pressure is analyzed using a
structured background mesh and hybrid body fitted mesh. The mesh contains 101,441 cells for the background mesh and 10,692 cells for the body fitted mesh. The overall view of the overset mesh and details of the overset assembly are shown in Figure 4.27a and b respectively. The interpolation approaches used in this study include simple inverse distance weighting, weighted least-square approach, Laplacian weighting, and clipped Laplacian weighting. The inverse distance interpolation scheme employs the weighting factor based on the inverse of the distance between the points where the data is to be interpolated and the data points within the search radius are used for interpolation. The weights for the inverse distance weighting procedure are bounded between (0, 1) and hence are monotone (the interpolated value is between the minimum and maximum values of the donor member values). For the least square based interpolation, the data at each grid point is expressed in terms of the gradient and a value at the reference point. The reference point can be any arbitrary data point in the set used for interpolation. The resulting over-determined system is solved for the gradient using the least square approach and the Graham Schmidt method. The least square approach for data interpolation can have weights that are greater than one and may not be monotone, which can lead to stability problems in the flow solver. To alleviate this problem, the interpolated value is limited between the extrema of the donor member values. The Laplacian interpolation scheme is based on the weighted averaging procedure, which estimates the value at an interpolated cell by a weighted average of the surrounding cells. The weight factors are derived such that the Laplacians are satisfied. Clipped Laplacian approach is a variation of the Laplacian scheme in which all the weights are clipped in the range (0, 2). A typical pressure distribution in front of the cylinder is shown in Figure
4.27c. It can be seen from the figure that the shockwave propagates through the overlapping region without any disturbances. The predicted pressure distribution on the cylinder surface using different interpolation schemes are compared in Figure 4.28. The predicted pressure distribution contours for all different interpolation methods behave very similar with little disturbances around the stagnation region of the cylinder.

Figure 4.27. Overset Mesh with Structured Background Mesh and Hybrid Body Fitted Mesh and Pressure Distribution from the Simulation.
4.6.2. Effect of Different Overset Mesh Topologies on Solution Accuracy

A mesh topology study is performed to observe the effect of different mesh topology to the solution accuracy. The study the effect of mesh topology used two different overset meshes has been created. Theses meshes are 1) structured background mesh and structured cylinder mesh and 2) structured background mesh and hybrid cylinder mesh. Details of these overset meshes are given in Table 4.2. The pressure distribution on the cylinder surface using the overset simulations are being compared with the result from the single mesh solution and the experimental data in Figure 4.29. The results are in good agreed with the modified Newtonian theory and single mesh result. The results show that by changing the mesh topology do not affect the prediction of pressure distribution.

Figure 4.28. Coefficient of Pressure across the Cylinder Surface for Different Interpolation Scheme.
Table 4.2. Mesh Information for Overset Meshes with Different Topology.

<table>
<thead>
<tr>
<th>Overset Grids</th>
<th>Number of Nodes</th>
<th>Number of Cells</th>
</tr>
</thead>
<tbody>
<tr>
<td>Structure – Structure</td>
<td>172,026</td>
<td>111,200</td>
</tr>
<tr>
<td>Prism – Structure</td>
<td>232,004</td>
<td>292,770</td>
</tr>
</tbody>
</table>

Figure 4.29. Pressure Distribution using Overset Meshes with Different Mesh Topologies.

4.6.3. Effect of Different Overset Non-Dimensional Wall Distance on Solution Accuracy

The effect of non-dimensional wall distance ($y+$) on the solution accuracy was studied using different normal distances of the first point off the cylinder on an overset mesh with a structured background and cylinder meshes. Four different $y+$ value were used in the simulations were 2.3, 1.8, 1.7, and 0.8. The diameter of the cylinder was taken as one unit. A geometric progression with a progression ratio of 1.1 was used for distributing points in the radial direction. The details of the meshes used for the simulations are given in Table 4.3. The computed pressure distribution on the cylinder from these simulations is compared with the data from the modified Newtonian theory in Figure 4.30. The computed pressure distribution from all the four overset meshes are in
good agreement with the modified Newtonian theory. It can be concluded that the mesh resolution has very little effect on predicting the pressure distribution.

Table 4.3. Structure-Structure Overset Mesh with Different $y^+$ to the Wall Information.

<table>
<thead>
<tr>
<th>$y^+$</th>
<th>Normal Distance to the Wall</th>
<th>Background Mesh</th>
<th>Body Fitted Mesh (Cylinder)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Topology</td>
<td>Number of Cells</td>
</tr>
<tr>
<td>2.3</td>
<td>0.002</td>
<td>Structure</td>
<td>101,442</td>
</tr>
<tr>
<td>1.7</td>
<td>0.001</td>
<td>Structure</td>
<td>101,442</td>
</tr>
<tr>
<td>1.4</td>
<td>0.0005</td>
<td>Structure</td>
<td>101,442</td>
</tr>
<tr>
<td>0.8</td>
<td>0.00025</td>
<td>Structure</td>
<td>101,442</td>
</tr>
</tbody>
</table>

Figure 4.30. Comparison of Pressure Distribution from Overset Meshes with Different Mesh.

4.7. Shock/Shock Interaction in a Laminar Hypersonic Flow

A shock/shock interaction heat transfer study is very critical in developing hypersonic vehicles because of the high temperature produced in the interaction region. According to Edney\textsuperscript{[32]} in a shock/shock interaction study, there are six different classifications of shock/shock interaction patterns which depends on the position of the
incoming shock pattern. Type III and type IV interactions are the most critical, since they are characterized by the largest values of wall pressure and heat transfer. In order to study this type of shock/shock interaction, several experiments have been conducted at ONERA in wind tunnel R5Ch\textsuperscript{[33]}. The experimental set up of the shock/shock interaction is shown in Figure 4.31, and all the dimensions are in millimeters. The shock is generated using an isosceles triangle prism with a base of 100 mm and a leading edge angle of 10°. In this experiment, the shock generator was rotated by 10° as shown in Figure 4.31. The angle of the shock created is 25°. The freestream temperature of 52.5K, pressure of 5.9 Pa, Mach number of 9.95, and Reynolds number per meter of 1.66 x 10\textsuperscript{5} is used for the experiment.

![Figure 4.31. Shock/Shock Interactions Experimental Set Up at ONERA.](Picture is reproduced from D’Ambrosio\textsuperscript{[33]})

An overset mesh approach is used for performing the shock/shock interaction study. The overset meshes used for the study are shown in Figure 4.32 and Figure 4.33. In Figure 4.32, the shock generator was rotated by an angle 10 degrees, while in Figure 4.33 the shock generator was rotated by 12 degrees. The pressure contours from the simulations using these meshes are also plotted in Figure 4.32 and Figure 4.33. It can be noted that the shock created by the shock generator passes smoothly across the
overlapping region to the cylinder. However, the orientation of the shock generator with 10 degrees produces an Edney Shock Type II shock/shock interaction pattern, while the 12 degree orientation produces Edney Shock Type IV.

Figure 4.32. Pressure Contour for Shock/Shock Interaction Study with 20° Shock Generator.
Figure 4.33. Pressure Contour for Shock/Shock Interaction Study with 22° Shock Generator.

The computed pressure contours are compared with the results from D’Ambrosio\cite{33} in Figure 4.34. The minimum, maximum, and average $y+$ values for the shock/shock interaction simulation case were 0.033, 0.7, and 0.184 respectively. The Type IV shock/shock interaction pattern compares well with D’Ambrosio’s results as shown in Figure 4.34. The pressure distribution and heat transfer results are shown in Figure 4.35 and Figure 4.36 respectively, where the highest peaks for both predictions are shifted downward as compared with the experimental result. The error could be caused by changing the angle of the shock from the shock generator or slight variation in incident shock strength.
Figure 4.34. Close View at the Shock/Shock Interaction Patterns.

Figure 4.35. Pressure Distribution on a Cylinder Surface.
Figure 4.36. Heat Transfer Distribution on a Cylinder Surface.
5. CONCLUSION AND FUTURE WORK

Most commonly used shock capturing schemes for transonic and supersonic problems suffer from spurious solution when used for hypersonic problems. The phenomenon is known as carbuncle phenomenon. In order to prevent the carbuncle phenomenon, there are many aspects that need to be carefully chosen. High dissipative Riemann solver, such as the HLLE scheme, is proven to be free from the carbuncle effect. However, the quality of the solution for such dissipative scheme is questionable because of the lack of accuracy inside the boundary layer calculation. There are two major fixes that have been proposed in the last decade: 1) the entropy fix approach and 2) adaptive scheme approach. The entropy fix approach tries to add numerical dissipation to the convective part of the governing equations. This approach can be very difficult to implement and also violates the real physical behavior of the governing equations. The second approach is simpler than the first one. The adaptive approach uses a more dissipative Riemann Solver in the strong shock region and uses a more accurate one in rest of the domain. Two adaptive schemes, Roe-HLLE and HLLC+, are implemented and evaluated for numerical accuracies.

The results from validation study show that both adaptive scheme prevent the carbuncle phenomenon and give better pressure predictions. It is also shown that adaptive Roe-HLLE and HLLC+ schemes do not require mesh alignment to produce good quality
solution. The benefit of using the combination of HLLC+ than Roe-HLLE is that both the HLLC and HLLE schemes are positively conserved which lead to a better stability. A minor drawback of using both adaptive schemes is the difficulty on deciding the switching parameter values. Currently value used for the switching parameter is chosen based on experience and it will require different value for each specific case.

The use of overset meshes could benefit the mesh generation process, especially for complex geometry and moving body problems. The most important parameter that needs to be carefully checked when using overset mesh framework is the interpolation method between the overset meshes. A primitive variable interpolation has shown better accuracy for testcases involving shock across overlapping region as compared to conserved variable interpolation. The mesh size in the overlapping region needs to be checked carefully to get the best interpolation solution. In order to get the best shock representation across the overlapping region, mesh size need to be similar for each mesh in the overset system. A study has been conducted to investigate the effect of interpolation schemes, mesh topology, and mesh resolution on solution accuracy. None of these variables has shown any significant effect on the prediction of pressure distribution for the testcases that were used in this study.

Predicting the heat transfer in a hypersonic flow is a challenging task and is tougher than other flow variables. From the heat transfer study, it was noted that different mesh topology and overset mesh interpolation method did not affect the heat transfer predictions. However, the heat transfer predictions were improved by decreasing the distance of the first point off the wall.
5.1. Future Work

Heat transfer prediction for hypersonic flow over a cylinder has been studied using single and overset meshes. The boundary condition for the cylinder surface is taken as an isothermal surface with wall temperature as the freestream temperature. However, the experimental investigation\cite{31} was carried out using a different temperature for the cylinder. Therefore, the predicted values of the heat flux were inaccurate because of the incorrect wall temperature used for the simulation as compared to the experiment. Numerical experiments has been conducted to compare the predicted heat flux distribution from a single block mesh with overset mesh simulations using different interpolation schemes, mesh topology, and mesh resolution. In all these calculations, the wall temperature is taken as the freestream temperature. In the first step of this study, results from different numerical schemes were compared using a single mesh and the comparison of the predicted heat transfer is shown in Figure 5.1. The adaptive scheme Roe-HLLE and HLLC+ give the smoothest distribution of the heat flux over the cylinder.

In the second step of the study, Roe-HLLE scheme was used to predict the heat flux to the cylinder using overset meshes. Three different parameters were used for these simulations: 1) interpolation schemes, 2) mesh topology, and 3) mesh resolution. The results from this study are summarized in Figure 5.2 - Figure 5.4. The heat transfer distributions for all different interpolation method and different mesh topology behave in a similar fashion with slight disturbances around the stagnation region of the cylinder (Figure 5.2 and Figure 5.3). The comparison of predicted heat flux using the overset meshes with different point distribution normal to the wall are shown in Figure 5.4. It can be seen from the figure that the computed heat flux are getting closer to the
experimental data as the distance of the first point off the cylinder decreases. It can be concluded from Figure 5.4 that the non-dimensional wall distance has stronger influence on solving the heat transfer than pressure distribution.

Figure 5.1. Comparison of Heat Flux Distribution on the Cylinder Surface.

Figure 5.2. Comparison of Heat Flux Predictions from Different Interpolation Techniques.
Figure 5.3. Comparison of Heat Flux Predictions using Overset Meshes with Different Mesh Topologies.

Figure 5.4. Comparison of Heat Flux Distribution from Overset Meshes with Different Mesh Resolution.
The boundary condition for isothermal wall has been fixed and numerical simulations have been carried out for the hypersonic flow past the cylinder using a single block mesh. New single block mesh with finer non-dimensional wall distance has been generated to improve the heat transfer predictions. The new mesh consists of 48,000 cells and 65,556 nodes. Figure 5.5 - Figure 5.7 show the temperature contour, pressure distribution, and heat transfer distribution on the cylinder surface for different flux evaluation schemes. Even with the prediction of the pressure distribution agrees with the modified Newtonian theory, the heat transfer rates are over predicted by all numerical schemes. The adaptive schemes, Roe-HLLE and HLLC+ schemes, predicts heat transfer rates within 10-15% accuracy. The heat flux contours for different flux schemes are shown in Figure 5.8. It can be seen from the figure that the symmetry of the computed heat flux along the spanwise direction is not preserved by different numerical convective flux evaluation schemes. In these simulations, three layers of cells have been used along the spanwise directions. From this figure, it can also be seen that HLLC+ gives a reasonable and almost symmetric heat transfer contour while others give spurious and asymmetric contour. The density and pressure contours on the cylinder surface are given in Figure 5.9 and Figure 5.10. Both results did not show as much disturbances as heat flux predictions. Therefore, further study is required to evaluate the cause for this spurious behavior for the heat flux predictions. Also, simulations need to be conducted to evaluate the effect of different overset parameters on simulation accuracy using corrected isothermal boundary conditions.
Figure 5.5. Temperature Contour for Cylinder Testcase with Single Mesh.

Figure 5.6. Cp Value at the Cylinder Surface.
Figure 5.7. Heat Transfer at the Cylinder Surface.

Figure 5.8. Heat Transfer Contour at Cylinder Surface for Different Flux Schemes.
Figure 5.9. Density Contour at Cylinder Surface for Different Flux Schemes.

Figure 5.10. Pressure Contour at Cylinder Surface for Different Flux Schemes.
LIST OF REFERENCES


