A Detailed Look into the Aerodynamic Forces Due to the Drag-Reducing Aerospike

Philp Douglas

Follow this and additional works at: https://scholarsjunction.msstate.edu/td

Recommended Citation
https://scholarsjunction.msstate.edu/td/113

This Graduate Thesis is brought to you for free and open access by the Theses and Dissertations at Scholars Junction. It has been accepted for inclusion in Theses and Dissertations by an authorized administrator of Scholars Junction. For more information, please contact scholcomm@msstate.libanswers.com.
A detailed look into the aerodynamic forces due to the drag-reducing aerospike

By

Joseph Philips Douglas

A Thesis
Submitted to the Faculty of
Mississippi State University
in Partial Fulfillment of the Requirements
for the Degree of Master of Science
in Aerospace Engineering
in the Department of Aerospace Engineering

Mississippi State, Mississippi

December 2016
Copyright by

Joseph Philips Douglas

2016
A detailed look into the aerodynamic forces due to the drag-reducing aerospike

By

Joseph Philips Douglas

Approved:

____________________________________
Keith Koenig
(Major Professor)

____________________________________
Gregory Olsen
(Committee Member)

____________________________________
D. Keith Walters
(Committee Member)

____________________________________
J. Mark Janus
(Graduate Coordinator)

____________________________________
Jason M. Keith
Dean
Bagley College of Engineering
Name: Joseph Philips Douglas

Date of Degree: December 9, 2016

Institution: Mississippi State University

Major Field: Aerospace Engineering

Major Professor: Dr. Keith Koenig

Title of Study: A detailed look into the aerodynamic forces due to the drag-reducing aerospike

Pages in Study: 69

Candidate for Degree of Master of Science

This thesis aims to simulate previous wind tunnel experiments on the drag-reducing aerospike in order to help validate the accuracy of CFD analysis. Multiple grids were created with the Pointwise grid generation software. The CFD analysis software used was Ansys Fluent, with both planar and axisymmetric cases being tested for the primary rocket in order to compare the differences. The tests with the primary rocket followed how a spike of set length reacted at various speeds.

Two additional experiments were duplicated. These helped confirm that the results obtained via Fluent were accurate. One case was a simple transonic spike model, and the other was a more complex hypersonic model. The results from both cases matched well with wind tunnel tests, validating results for the primary rocket.

This thesis paves the way for anyone wanting to continue a more in depth study into the flow properties of any type of projectile.
DEDICATION

This paper is dedicated to my wonderful fiancée Marcie Walker, for always being a strong support, as well as my parents Rusty and Betty Douglas, who are probably more excited than I am about finishing school.
ACKNOWLEDGEMENTS

I would like to express my sincerest gratitude to several people who made it possible for me to get this far in my work. First, a big thanks to Dr. Keith Koenig for his help throughout these years. He has been an incredible mentor over the years, even before I came in as a freshman at MSU. I could not have made it this far without his guidance, even as the odd aero student among all the astros. It has been a truly educational journey.

Next, I would like to thank Dr. William E Milhollen of NASA Langley. I was fairly confident I knew exactly what I wanted to do once I finished undergrad, only to find out during my time at NASA that this was not the case at all. He helped me realize that there is never an end to one’s education, and to open yourself to any opportunity that presents itself and to expose yourself to new ideas. I have tried to keep an open mind in all aspects of life due to my time working with him, and fully expect to find new interests as I learn more about what it means to be an engineer.

I would also like to thank Dr. Greg Olsen for being a strong guide and very personal professor these past few years. He has an amazing talent in connecting with students, and it has helped many of my graduating and graduated aerobuddies face our concerns with post-school life with a smile. He has helped us obtain confidence in ourselves and our abilities, when we may not have had much ourselves at the time.

I would not have been able to get this far in my research without help from Dr. Keith Walters. He gave me the opportunity to work with all of this software, and his
knowledge in CFD was a tremendous help in trying to figure out what to do when my research stalled. I was able to see some really cool work due to being able to assist with some of his research, and it has helped me realize what I would like to do as an engineer.

Finally, I would like to shout out to my aerobuddies: Clayton Mord, Mary Kate Smith, Joseph St. Columbia, Paromita Mitra, Kenny Herbert, Ian Detweiller, Torrey Harriel, Rachel McFalls, Lewis Howard, and many more. It has been a wild ride! We have been through quite a bit over the years, and I hope that our adventures continue as we enter the work force as actual engineers and adults, as terrifying as that is to think about. Start together, finish together!
# TABLE OF CONTENTS

DEDICATION ........................................................................................................................................... ii

ACKNOWLEDGEMENTS .......................................................................................................................... iii

LIST OF TABLES ...................................................................................................................................... vii

LIST OF FIGURES ................................................................................................................................. viii

NOMENCLATURE ................................................................................................................................. xi

CHAPTER

I. INTRODUCTION ................................................................................................................................... 1

II. AEROSPIKE VARIATIONS ................................................................................................................. 4

III. GRID GENERATION .......................................................................................................................... 8

   3.1 3D Software .................................................................................................................................. 8
       3.1.1 SolidMesh ............................................................................................................................... 8
       3.1.2 Pointwise .............................................................................................................................. 12

   3.2 2D Software ................................................................................................................................ 14
       3.2.1 GridPro ............................................................................................................................... 15
       3.2.2 Pointwise ............................................................................................................................ 18

IV. TEST SETUP ...................................................................................................................................... 20

   4.1 Incompressible vs Compressible ................................................................................................. 20
   4.2 Steady State vs Transient ............................................................................................................. 21
   4.3 Planar vs Axisymmetric ............................................................................................................... 22
   4.4 Solution setup .............................................................................................................................. 22
       4.4.1 Defining the Pressure ............................................................................................................ 22
       4.4.2 Fluent Equations .................................................................................................................... 23
       4.4.3 Materials ............................................................................................................................. 24
       4.4.4 Boundary Conditions ............................................................................................................. 24
   4.5 Solution ......................................................................................................................................... 25
       4.5.1 Solution Methods .................................................................................................................. 26
V. SIMULATIONS AND RESULTS ................................................................. 29
  5.1 Refocusing the Project ................................................................. 29
  5.2 Comparison to previous models ............................................. 30
    5.2.1 Transonic Blunt Body Comparison ................................... 31
    5.2.2 Hypersonic Blunt Body Comparison .............................. 33
  5.3 Primary Tests ............................................................................ 36
    5.3.1 Planar vs Axisymmetric .................................................. 36
      5.3.1.1 Planar Results ....................................................... 37
      5.3.1.2 Axisymmetric Results ........................................... 38
    5.3.2 Results At High Speeds .................................................... 40
      5.3.2.1 Mach 1.5 ................................................................ 40
      5.3.2.2 Mach 3.0 ................................................................. 42
      5.3.2.3 Mach 4 and Up ....................................................... 43

VI. CONCLUSIONS ............................................................................... 45

APPENDIX

A. GRID GENERATION ........................................................................ 50
B. HYPERSOONIC COMPARISONS .............................................. 53
C. MACH 1.5 COMPARISONS ......................................................... 57
D. MACH 3.0 COMPARISONS .......................................................... 61
E. MACH 4 VS MACH 5 ................................................................. 64
F. FLUENT CASES ............................................................................ 67
LIST OF TABLES

4.1 Standard atmospheric conditions for testing ..................................................22
5.1 Dimensions for Primary and Transonic Rocket .............................................30
F.1 Transonic Cases .............................................................................................68
F.2 Supersonic Cases ..........................................................................................69
LIST OF FIGURES

2.1 Various shocks for a blunt body with and without a spike [7] .........................5
2.2 Long spike below and above Mach 1 [8] .........................................................6
2.3 Trident I missile with aerospike [1] ..................................................................7
3.1 Major Tom, second stage .................................................................................8
3.2 Major Tom in .igs format .................................................................................9
3.3 Modified 4-Fin version of Major Tom ...........................................................10
3.4 SolidMesh grid version of 4-Fin Major Tom ....................................................11
3.5 Full SolidMesh grid of 4-Fin Major Tom ........................................................11
3.6 Pointwise grid version of 4-Fin Major Tom .....................................................13
3.7 3D Pointwise grid ............................................................................................14
3.8 GridPro 2D grid ................................................................................................16
3.9 Internal surfaces via GridPro ............................................................................17
3.10 Internal surfaces – expanded view ....................................................................17
5.1 Diagram of the rockets ....................................................................................30
5.2 Hypersonic aerospike surface dimensions (in inches) ....................................31
5.3 Transonic comparison case at Mach 0.91 ......................................................32
5.4 Direct comparison to original experiment .....................................................32
5.5 Dynamic pressure comparison of small vs large test sections .........................34
5.6 Side by side comparison of the hypersonic case .............................................35
5.7 Direct comparison to original experiment .....................................................35
5.8 Contours of dynamic pressure (planar) at Mach 1.5 ........................................38
5.9 Mach 1.5 - velocity contours for planar vs axisymmetric ......................41
5.10 Mach 1.5 - static temperature contours for planar vs axisymmetric .......41
5.11 Mach 3 - total pressure contours of planar vs axisymmetric ..................42
5.12 Mach 3 – absolute(static) pressure contours of planar vs axisymmetric ....43
5.13 Mach 4 vs Mach 5 – axisymmetric total pressure .................................44
5.14 Mach 4 – velocity vectors vs dynamic pressure .....................................44
A.1 4-fin Major Tom mesh via Pointwise .........................................................51
A.2 2D basic rocket grid with blocks via GridPro ............................................51
A.3 2D basic rocket grid without blocks via GridPro ......................................52
B.1 Hypersonic velocity comparisons – small vs large test section ...............54
B.2 Hypersonic static temperature comparisons – small vs large test section ........................................................................................................54
B.3 Hypersonic total temperature comparisons – small vs large test section ........................................................................................................54
B.4 Hypersonic static pressure comparisons – small vs large test section ......55
B.5 Hypersonic static pressure – large test section with a low range ..........55
B.6 Hypersonic static pressure – large test section with higher ranges .........56
C.1 Mach 1.5 dynamic pressure comparisons – planar vs axisymmetric ........58
C.2 Mach 1.5 static pressure comparisons – planar vs axisymmetric ............58
C.3 Mach 1.5 total pressure comparisons – planar vs axisymmetric ............59
C.4 Mach 1.5 total temperature comparisons – planar vs axisymmetric ........59
C.5 Mach 1.5 – Axisymmetric dynamic vs static pressure ............................60
C.6 Mach 1.5 – Axisymmetric direct comparison of dynamic and static pressure ........................................................................................................60
D.1 Mach 3.0 dynamic pressure comparisons – planar vs axisymmetric ..........62
D.2 Mach 3.0 static temperature comparisons – planar vs axisymmetric ..........62
D.3 Mach 3.0 total temperature comparisons – planar vs axisymmetric ..........62
D.4 Mach 3.0 velocity comparisons – planar vs axisymmetric .......................63
E.1 Mach 4 vs Mach 5 – Axisymmetric velocity ........................................65
E.2 Mach 4 vs Mach 5 – Axisymmetric absolute (static) pressure ...............65
E.3 Mach 4 vs Mach 5 – Axisymmetric dynamic pressure .............................65
E.4 Mach 4 vs Mach 5 – Axisymmetric static temperature ...........................66
E.5 Mach 4 vs Mach 5 – Axisymmetric total temperature .............................66
NOMENCLATURE

L     Spike Length
L_b   Length of Rocket Body
D_2   Cylinder Diameter
D_1   Spike Diameter
R     Dome Radius
CFD   Computational Fluid Dynamics
HPC^2 High Performance Computing Collaborator
CHAPTER I
INTRODUCTION

It has long been known that having some sort of nose on a projectile will help the projectile go further and be more accurate. The typical conical nosecone has been a staple of rockets and projectiles for years. Science has changed the exact shape of the conical nosecones over the years, but for the most part it is still used as a good way to help control the rocket and reduce the drag.

As more missiles were being developed through WWII, researchers started to study various aspects of missiles and rockets to try and figure out how to make them more efficient. One way to do this would be to reduce the drag even more than the current conical shape would allow. This led to the realization that one could keep the front of the rocket as a blunt body, and attach a “spike” at the top that would essentially replace the conical nosecone that was being used before. This would allow for a lighter load, and in certain cases more room to store missiles that may have been too tall otherwise.

A clear example of this being used to great advantage was the C4 Trident I missile, developed by Lockheed Missiles and Space Corporation [1]. Previous to the Trident I, submarines were able to easily fit the missiles into the missile bays. As missiles became larger due to increasing payloads and needing to fly further distances, a way had to be found to save space in order for the missiles to fit.
As Lockheed was trying to find various methods to save space, they found that the telescoping aerospike was perfect for what they needed. One beneficial result they found was that the aerospike was able to reduce drag by approximately 50% at supersonic speeds. This was a huge reduction, and the Trident I was a very successful early case of how useful the aerospike could be.

Despite this advantage in drag-reduction, the aerospike has not been a very common shape to use on rockets. There are several problems one could run into with developing the spike, chief among these being structural stability. As a rocket reaches higher speeds, the spike must become longer in order to keep a steady flow around the nose of the rocket. However, if the spike is too long, stability becomes an issue. The longer spike interferes with the air flow and flow separation occurs downstream at some point on the shaft of the spike instead of at the tip. Being able to reliably create a telescoping aerospike, as used on the Trident I, could be a difficult task to implement. The simplest solution would be to have a spike of set length based on the maximum speed of the rocket or missile, but if there is some variance with this speed then the spike might do more harm than good. There are possible fixes for this that have been implemented in previous cases [1].

One thing that may be an issue is testing how a spike will perform. The primary method of testing has been to use wind tunnels with a spike and blunt body model. One area of interest with the spike is how it will perform at supersonic speeds, so supersonic wind tunnels are often used. A possible problem with a lot of these high speed tunnels is the fact that the test sections are very small in order to reach said speeds. The test then could possibly be compromised due to the presence of the walls being so close to the
model. CFD simulations are an easy way to bypass this possible restraint, as it is generally a simple matter to create a model in an empty space with no constraining walls. It should be relatively painless to simulate some previous physical tests and see how they compare to a case that is not constrained by a small test section.
CHAPTER II
AEROSPIKE VARIATIONS

While not very practical, testing has been done over the years to see how a blunt body responds when introduced to a flow field. It has been shown that for high speed flow, there is a bow shock wave formed in front of the blunt body [2]. This leads to high surface pressure, which results in high drag. In order to reduce the surface pressure, and thus the drag, a spike was placed on the front of the blunt body in order to create a conical shock wave. As has been noted in many experiments, the conical shock is much better at reducing the surface pressure and aerodynamic drag, and increasing the lift coefficient. One major reason this spike is so effective is that a region of recirculating separated flow is produced that protects the blunt surface from the flow. [2].

One of the largest benefits to the spike is that it reduces drag, hence the name drag-reducing aerospike. The documentation of and mechanisms for the drag reduction are described in detail in the various papers referenced. The focus of the present study is the details of the spike-induced flow field including temperature and pressure distributions, and flow separation caused by the aerospike.

The aerospike has had many possible variations that have been studied. Some of the cases, particularly in the transonic region, deal with a simple blunt spike [3]. Other cases, such as with hypersonic flows, have variations like a pointed spike [4] or flat tip on the front of a narrow spike [5]. There are some cases that have a variety of spike tip
configurations, like a sphere or a cone that are much more pronounced than the diameter of the spike [6]. Each different type of spike tip produces a slightly different shock wave, so the tip chosen will depend on what exactly is needed from the object.

For the blunt body with no spike, a strong detached bow shock forms that is extremely detrimental to the flow. In general, having a spike that doesn’t exceed a certain length will improve flight by producing a conical shock wave. The spike can have a variety of shapes that impact the conical shock in different ways. A pointed spike produces a sharp shock wave with a thin shear layer. The reattachment shock almost hugs the body along the front domed section.

Other variations for the tips of the spike include a rounded tip, which produces a shock around the spike. The separation shock for this occurs along the length of the spike. Examples of these cases can be seen in Figure 2.1.

![Figure 2.1](image)

The detached bow shock can be seen for the blunt body (left), with the more conical shocks appearing for the case that has a spike (center and right). The tip of the spike determines what general shape the shock wave will have.

For the aerospikes to be effective, they must not exceed a certain length. One major problem with this is that as the rocket moves faster, the optimal length of the spike
increases. Being slightly too short is not too terrible of a problem, as the beneficial effects of the spike will still apply to the rocket. However, the real problem occurs when the spike is longer than the maximum optimal length. At this point, issues with the flow appear and there is generally a very large and detrimental increase in the drag on the rocket. The area of flow separation decreases at faster speeds, so at high speeds a long spike will protrude in front of the point of flow recirculation instead of being the point at which this occurs. An example of this can be seen in Figure 2.2.

![Figure 2.2](imageurl)

**Figure 2.2**  Long spike below and above Mach 1 [8]

These pictures show the detrimental effects of the spike extending out past its optimal length for a set speed. The picture on the left is travelling at Mach 0.91, and the picture on the right is travelling at Mach 1.11. The spike is too long in both cases, with more noticeable problems showing in the supersonic case.

This is where it gets difficult to implement the spike, unless a telescoping aerospike is implemented, as seen on the Lockheed Martin Trident I missile. This was the first missile to actively use the aerospike, and it did so with great success. One of the key
points of this missile was that Lockheed actually made the spike slightly shorter than what the optimal length at max speed was, just to account for the fact that the missile might, under the right circumstances, go slightly faster than expected [1]. This safety measure ensured that even if it did go faster than it should, the Trident I missile would not experience negative effects due to the spike being too long.

![Trident I missile with aerospike][1]

These pictures of the Lockheed Trident I missile show the flow around the missile before and after the spike is deployed. The blunt body by itself causes a large increase in pressure that increases the drag, while the aerospike creates an area of lower dynamic pressure that in turn decreases the static pressure and lowers the total drag [14].
By far the largest obstacle for this research was initially learning how to create the grids that were needed to run simulations of the rockets under various flight conditions. Several types of grid generation software were used for various reasons, which are outlined below. The rocket that was originally being created was a replica of Major Tom, the rocket used by the MSU Space Cowboys in 2013-2014, due to the availability of flight data from Major Tom’s launch that would be compared to the data obtained from simulations. Due to complications mentioned below, this idea was abandoned and a simpler idea was used based on other research [3].

Figure 3.1   Major Tom, second stage

The second stage of Major Tom had a total length of 98 inches.

3.1.1   SolidMesh

SolidMesh is an older grid creation software [9] that is infrequently used today, but at the time of this project it was still a viable option at HPC². Initially, the default 3-
fin version of Major Tom’s second stage was used, as shown below in Figure 3.2, but due to wanting to improve symmetry, an attempt was made to create a 4-fin version of the rocket. This ended up not working, due to the fins apparently not being duplicated and rotating at 90 degree angles around the body of the rocket. A different rocket was created later that was symmetric, and this allowed for accurate testing.

Figure 3.2   Major Tom in .igs format

This was the original file for Major Tom, after exporting and editing it in SolidMesh.

Due to how the .igs file was made, this proved to be more difficult than originally planned as none of the lines from the file were lined up in an easy to format method. A little bit of guessing went into figuring out how far to rotate certain aspects of the rocket to line up with each other, and eventually a seemingly accurate 4-fin .igs file of Major
Tom was created, shown below in Figure 3.3. As can be seen from the picture, additional fins were created and there were multiple additions to the tail section in order to line up all of the necessary additions together. The 3-fin case was much simpler, and required very little maintenance to get a working model.

![Figure 3.3 Modified 4-Fin version of Major Tom](image)

Due to the possibility of needing a two dimensional model, the original 3-fin model of Major Tom was converted into a version with four fins.

At this point, it should have been a simple step to turn each section into a surface, create a cylindrical outer boundary around the rocket, and export the case into the CFD analysis software, Ansys Fluent [10]. For the most part, this worked out just like just like it should have. A full grid was created, and a very good looking model of Major Tom was all but ready to be exported, as can see below in Figures 3.4 and 3.5.
Each surface of Major Tom was connected and a grid was created of the rocket.

This is the full test grid created in SolidMesh for the 4-fin version of Major Tom.
Unfortunately, it was about this time that the operating systems at HPC\(^2\) were upgraded, and the new operating systems turned out to be incompatible with SolidMesh due to the software being outdated. A couple of options presented themselves for what other software could be used and learned quickly. One of these was Pointwise, and the other was GridPro. The difficulties of each of these are detailed below.

3.1.2 Pointwise

After SolidMesh became unavailable, it was necessary to find another grid generation software that would be fairly simple to learn in a short period of time. Pointwise is one of the newer grid generation software available for use at HPC\(^2\) [11], so this was the next software that was used. It is much more user friendly than SolidMesh, and is not incredibly difficult to learn. Figure 3.6 shows the Pointwise version of Major Tom.
A hybrid grid was determined to be the best grid to work with due to the time constraints. In order to create a hybrid grid, a boundary layer must be composed of structured cells, and then the rest of the grid is filled with unstructured cells. The benefit of a hybrid grid is that the unstructured portion makes tests run quicker, however the structured cells near the body still give very accurate results. The grid generally looked very nice, as seen in Figure 3.7, but Fluent was quick to point out errors in the grid such as an absurdly high aspect ratio (13 million) in certain unstructured cells near the edge of the grid. Multiple attempts were made to decrease the aspect ratio in these bad cells, but nothing brought it down to useable levels.
The three dimensional hybrid grid of Major Tom created in Pointwise. While much finer than what was created in SolidMesh, the unstructured cells close to the inlet and exit were bad enough to cause the grid to be unusable. A structured grid could have theoretically been used, but would have made for extremely resource intensive tests.

3.2 2D Software

This was the point that the switch was made from 3D to 2D, and the Major Tom rocket was completely scrapped. Overall this was a good idea, as it was discovered that due to the editing of the .igs file to try and line everything up for a 2D model, the file was actually unable to be completely created as a 2D model. The fins were slightly off, from both the body and each other, and trying to recreate them in two dimensions would have been a nightmare. Pointwise has also mostly been used for 3D models by the researchers.
at HPC², with very little creating anything in 2D. The methods for creating a 3D and 2D grid are not quite compatible, so the people that had helped with earlier 3D models were unable to do much in regards to switching to 2D models. Eventually, a switch was made back to Pointwise to work on 2D models, after running into some problems with GridPro.

### 3.2.1 GridPro

The next software used was GridPro, a free to use grid creation software [12]. This software is interesting in that internal surfaces need to be created throughout the mesh in order to create a precise and accurate grid. As opposed to regular surfaces, internal surfaces allow the mesh to be created on both sides of the surface, and this allows the user to be able to have a very refined mesh around any possible shape or at any angle they need. An example of the grid being refined around the spike due to internal surfaces can be seen below in Fig 3.8. Figures 3.9 and 3.10 also show the internal surfaces along the nose and expanding out from the body of the rocket at angles in order for the mesh to spread out somewhat evenly.

A fairly decent grid was created, with some possible issues showing up near the tip of the spike. This area of extremely clustered cells can be seen below in Fig. 3.8. For the most part this method seemed to work fine. However, the method to set a boundary layer and create user input grid spacing near the surface of the rocket was unknown, so overall the mesh was impracticable for the tests that it would be used in. A practical grid would require an initial cell height with a growth rate of around 1.2-1.3 to accurately simulate the area within the boundary layer, and a way to do this in GridPro was not found. After consulting with Dr. Walters, another attempt was made with Pointwise, with
the new rocket data that had been created instead of the rather unique shape of the rocket with the dome that was present on Major Tom.

Figure 3.8   GridPro 2D grid

The grids created with the 2D rocket in GridPro looked very nice, however the clustered cells above and below the spike tip could have possibly caused problems for tests. The mesh in this area is determined by internal surfaces, and creating an internal surface and smoothing out the mesh around these surfaces and the actual services proved to be problematic.
The internal surfaces used to create the finer mesh around the nose of the rocket can be seen. They make a rectangular box around the nose. This idea was later implemented to some extent into the 2D Pointwise grids.

Internal surfaces were created to expand outwards from the rocket in order to make the grid lines follow a certain pattern. The origin of these lines can be seen in Fig 3.8
3.2.2 Pointwise

One major reason that the grids were not able to be imported from Pointwise and were not being read into Fluent was due to the fact that the domains, or grid blocks, that represent each section were apparently oriented incorrectly. It is a simple matter to correct this with unstructured grids, as the orientation menu shows which way the unstructured blocks are pointing normal to the surface. However, structured grids are a little trickier as they do not have an easy way to see which way they are “pointing”. The easiest method is to use the right hand rule, and orient the directions of the outer boundaries of each domain in a way that would follow this rule. The issue with this is that there did not seem to be a way to determine which point was the initial point, and thus one vector may be oriented correctly but another might not be. It took a little playing around with before the direction to make each domain positive was found according to Fluent.

There were various video tutorials available for Pointwise online, although most of these tutorials were for 3D models. After a lot of searching, a very useful tutorial was discovered that essentially covered everything needed for a 2D mesh for the current rocket data. A multigrid C-grid was used (shown below theoretically), and refined until a good mesh was available to use. This was the first grid used, and some fairly decent results were obtained, but most cases were having problems. The reason for this was the entire “C” portion of the grid was the inlet, and the velocity components were causing issues. Part of the tests incorrectly had the flow normal to the boundary, which meant that the top and bottom corners of the “C” had 100% Y-direction velocity, and no X-direction velocity. The Y-direction velocity needed to be 0 at all points, but because of how the
inlet was shaped, there needed to be at least some velocity in the Y-direction at the corners where the edge was completely horizontal. Because of this, a simple case with a flat inlet was used.

The grid that used a flat vertical line for the inlet and exit was the final design for all grids used in these tests. The entire grid is essentially a giant rectangle, this allows for the air to flow in the X-direction without any interference in the Y-direction. The front (left side) of each grid was set as a velocity-inlet, and the back (right side) of the grid was set as a pressure-far-field type of surface in Fluent.
CHAPTER IV
TEST SETUP

While the final simulations generally had one similar setup, there were multiple boundary conditions and model types that went through various stages while trying to find an ideal set of conditions. Different models were used, depending on if the current simulation was compressible or incompressible, and if the geometry was planar or axisymmetric.

4.1 Incompressible vs Compressible

The initial tests were made as incompressible tests, because the grids that were not yet refined were easier to set up and run as incompressible tests, and because using the same grid with a density based solver resulted in non-convergent solutions. Until a stable grid was created, incompressible cases were used, even though it was known beforehand that the solutions obtained from these pressure based cases would not be as valid as the cases obtained from the density based cases. While this was somewhat time consuming, it allowed some interesting results to be obtained once a working grid was created. Fluent is mostly a compressible flow solver, so it was interesting to see how the flow was simulated when run as an incompressible case.

One major obstacle from the pressure based solver is that the flow is calculated outward from the rocket, so the flow at the velocity inlet is calculated and greatly impacted by what is happening at the rocket. This is clearly not what should happen, and
the density based solver does not have this problem. As a pressure based solver, however, this caused interesting points to form at the inlet that were greatly impacting the overall flow. The density solver calculates the flow at the inlet and exit, and works inward from there. This essentially means the flow at the inlet is largely unaffected by what happens at the rocket, and will stay constant regardless of what happens downstream. Even the preliminary calculations using a density based solver showed much more stable flow upstream of the rocket, as opposed to the pressure based cases where there was almost an immediate jump in pressure at the compromised points along the inlet.

4.2 Steady State vs Transient

It should be noted that the majority of these cases were run as steady state cases, and not as transient cases. As such, these results are slightly different from what the transient cases would have given. The steady state cases still seem to be fairly accurate at high speeds (M > 1) when compared to the experiments done previously, however the transonic cases have some uneven pulsing at the rear of the rocket that would most likely not have been present if they were not steady state. While this will probably affect the results in some way, it seems to mostly impact the flows trailing off the end of the rocket and not the flows around the nose.

While running transient cases would have been helpful from a comparison sense, these transient cases took far more resources than the steady state cases. An attempt was made to simulate one of the transonic cases as a transient case, but the time required to get anything meaningful was enormous, and it would have been incredibly tough to have gotten more data through the transient model within the short period of time that was available for testing.
4.3 Planar vs Axisymmetric

Initial simulations used the planar models for each test. This provided vastly different results, as the planar case assumes that the 2D model is infinitely wide, instead of considering that the model might have a separate shape, like the round shape of the rockets. Planar cases were generally easier to set up and run, so most of the initial cases were planar in order to get familiar with the CFD software. After similar results were obtained using the planar method, the cases were switched to axisymmetric, which allowed Fluent to correctly set up the physics of the model and simulate the flow around it much more accurately. A summary of the differences in the planar and axisymmetric cases are covered in more detail in section 5.3, with comparison pictures of the two cases at the same Mach numbers included.

4.4 Solution setup

The final type of model tested was an axisymmetric, compressible (density based), steady state model. However, there are many options other than these that are required for Fluent to run the case properly. The conditions used for testing are included in Table 4.1 below.

Table 4.1 Standard atmospheric conditions for testing

<table>
<thead>
<tr>
<th>Temperature (K)</th>
<th>Pressure (Pa)</th>
<th>Density (kg/m$^3$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>288.15</td>
<td>101325</td>
<td>2.451404</td>
</tr>
</tbody>
</table>

4.4.1 Defining the Pressure

One area that was not very clear was how Fluent defined the pressure. The inputs for pressure in Fluent are listed as gauge total pressure and gauge or supersonicinitial
gauge pressure, which are not particularly well defined in the Fluent User’s Guide. However, two different articles were eventually found that did a slightly better job of explaining what Fluent meant by gauge and gauge total pressure. The gauge pressure, and the supersonic/initial gauge pressure, should refer to the static pressure. The gauge total pressure refers to the total pressure, and the recommendation for certain cases was to simply leave this blank. For the gauge and supersonic/initial gauge pressure, the standard atmospheric value of 101325 Pa was used for most cases, with a few early cases using 0 Pa just for the sake of comparing how that changed things. From the results of the two, no serious differences were detected, but the pressure contours were just different enough that the standard atmospheric value was used throughout due to the confusion with the User’s guide implying that the gauge pressure is not necessarily the actual gauge pressure. This is currently the point of most concern, since Fluent was not very clear with what each pressure was supposed to represent and there seemed to be some pressures that should be the same, but had different names.

4.4.2 Fluent Equations

The Fluent equations that were used for each case were the energy equation and the viscous equation. For the viscous equation, the Spalart-Allmaras (SA) 1-equation model was used because it is a common model used for aerospace related tests. More details about this model can be found in Section 4.2 of the Fluent Theory Guide [10], and many CFD websites and texts. The strain/vorticity option was picked for the SA viscous model, but other than that the default options were kept, as per the recommendation from the User’s Guide. The SA performed very well when compared to previous wind tunnel experiments.
4.4.3 Materials

Due to setting the right side of the grid as a pressure far field in order to specify the Mach number at the exit, the material properties of air needed to be edited slightly in the Fluent options dialog box. In order to use the pressure far field boundary condition, the air is required to be set as an ideal gas. Other than that, the materials page was left alone.

4.4.4 Boundary Conditions

For the boundary conditions, each edge of the grid needed to be set. The left edge of the grid was set as the velocity inlet, and the right edge was set as the pressure far field. The velocity and Mach number for these two conditions were set in the dialog box for each case, as were the static pressure and temperature. A check after this was to go to the reference values and choose the exit and inlet as a reference in order to see if the values mostly matched each other. Once the pressure and temperature were set, the velocities for the velocity inlet and the pressure far field matched each other almost perfectly in every case. It was very helpful to see that setting the Mach number instead of an exit velocity still gave the same result as when the velocity by itself was set.

One important section that was unfortunately ignored in early tests was the velocity specification method in the velocity inlet and pressure far field dialog boxes. The first grids created were C-grids. The velocity inlet was initially rounded instead of being a flat rectangle as required by this format. Since this test just needed the flow to be travelling in the X-direction, this caused some problems due to the curved shape of the inlet. For a flat, vertical velocity inlet, a flow that is normal to this inlet would only travel in the X-direction. However, due to the curved inlet, the flow being normal to the inlet
meant that components of air flowing in the Y-direction would be included, which caused a lot of problems near the edges of the inlet and the outer symmetry planes. This is due to Fluent trying to add an almost completely vertical flow at a point where there should be none.

Eventually this error was caught and an attempt was made to only have the velocity flowing in the X-direction, however again there were problems with this due to the near horizontal sections of the inlet where the inlet was connected to the rest of the grid. This was the point where the C-grid was abandoned and the rectangular grid came into existence.

The other edges were much simpler to set up than the velocity inlet and pressure far field. For the planar models, the top and bottom edges were set as symmetry boundaries, which essentially set these as open areas and not any kind of solid wall. The axisymmetric case was similar in that the top edge was set to be a symmetry boundary, which means that the air is not confined within a surface like it would if the top edge was a solid wall. Due to this grid being cut in half, the bottom edge was set as an axis boundary so Fluent would know to use this edge as the axis of rotation. The two cases that were used to compare to actual experiments had the top edge set as a wall in order to simulate the wall of a wind tunnel test chamber, and the rocket surface was defined as a wall.

4.5 Solution

Underneath the solution setup in Fluent are the solution methods and controls. While first attempting to switch to density based from pressure based, this was the area
that kept causing problems for each case. The pressure based options are very simple, and as such it is easier to test a particular grid using pressure based to make sure it works.

The density based model has a few options that are a little less intuitive, for those that might be unfamiliar with CFD. Like the pressure based model, there is an option for implicit vs explicit. The explicit solution relies on less information, and can handle some messy cases that implicit might not be able to power through. One difference in the explicit method for density based models is that fact that an option for multi-grid exists. This is not an option available in the more solution based implicit method.

4.5.1 Solution Methods

The first option under the Solution Methods tab is whether this case will run as an implicit or explicit case. Running it as an explicit case gave less accurate results due to not requiring solving multiple equations for each value, but allowed for greater flexibility in being able to handle grids and meshes that might be somewhat more problematic.

Almost every case that ran as a density based model started as an explicit case, and was switched to implicit after it started to smooth out and stabilize. With explicit, one can set up a multi-grid, which can make running simulations much less resource intensive. However, for this case the multi-grid approach was only used until each model was past the point of blowing up if run as an implicit case.

For early cases, the various equations and spatial discretization methods were initially set up as second order problems instead of the more first order problems. Due to being somewhat unfamiliar with creating grids early on, a lot of the models created were made somewhat poorly. Running the case as the more accurate second order problem allowed the software to obtain better results from the model where it might otherwise
cause issues. As each case ran, it would be changed to first order, explicit, and then after a few hundred more iterations the case would be changed to first order, implicit. As the grid creation software became easier to use, and better quality grids were created, it became unnecessary to start with as a second order model, and the stable first order method was used. Finer grids that had better cells throughout each model made the calculations easier to run, especially for the initial iterations, and these improvements allowed for more accurate results as time went on.

Each case was initialized with hybrid initialization, as this was one of the recommendations of the Fluent User’s Guide, and set up to autosave every 100-200 iterations. This allowed an easier inspection of points that would blow up, with some modifications easy to implement for a short period if needed. This became less frequent as the better quality grids were created.

Most of the cases would run for a minimum of 30,000 iterations, with some cases being upwards of 150,000 iterations and the transient case being several times that. The large variance in the number of iterations was due to how closely each case had to be monitored, with the initial lower quality grids requiring much smaller values for the under-relaxation factor of each equation. Essentially, this meant that the cases with smaller under-relaxation factors were taking smaller steps, which helped smooth out some of the errors that the better quality grids might not have. Most of the problems would arise in the first 1000 or so iterations, so for the most part it was smooth sailing after that number was hit.

There were about 24 different two dimensional grids created in Pointwise, with various stages of partial success being achieved for the first 16 of those grids. Most of
them each had about 500-1000 iterations, if they were able to run at all. The 17th grid created in Pointwise was the first completely successful one, and this was the first case that had a case for each Mach number ranging from 0.8-5.0, with a 0.5 Mach step from Mach 1.5 – Mach 5.0. There were about five tests for each of these cases, and they ranged from 30,000 to 150,000 iterations. This was the only time a transient case was attempted, and there were about 400,000 iterations for that model. Even with that, it did not cover enough time to truly be useful, as it covered less than half a second when about five seconds were needed overall.

Axisymmetric models were created starting with the 18th mesh. These only ranged from Mach 1.5 to Mach 5.0, at 0.5 Mach intervals, and these each had about 35,000-50,000 iterations.

The 19th and 20th meshes were the variable spike length models, but unfortunately time ran out before these could be checked out. Only a few hundred iterations were completed with these before physical access to the computer was lost.

There were several extra grids that were created afterwards for completely new rockets, and a familiarity with the Pointwise software allowed grids with domes to be created. These were the verification runs, and were run remotely through the cluster at HPC². Given enough time, it would have been helpful to run the domed, transonic-only rocket as an axisymmetric model. This would have allowed for greater comparison between the axisymmetric and planar models at more speeds, and would have been a good additional way to verify the results that were obtained. Tables of all the usable grids and tests, starting with the 17th mesh, are included in Appendix F. This is to help show how many models were created and successfully used.
5.1 Refocusing the Project

Once the best conditions were found for testing the grids, a plan was created for what exactly would be tested, and how these tests could be proven as being accurate. The initial plan was to create a grid for an aerospike of a set length, and compare it at various speeds. A spike of different length could be then be tested at one of these previous velocities, in order to show how spike length could change the flow. Another part of the testing schedule was to have a conical nosecone of similar length to the original spike, and run this nosecone at the same speeds as the tests for the first spike. This would allow the flow properties of an aerospike to be compared to those of a conical nosecone.

Unfortunately, not all of this was feasible within the timeframe. Since the original focus of the paper was on comparing the flow properties of the spike to that of the conical nosecone, a change in the tests that excluded the nosecone would require a different focus and a new way to determine whether or not the results obtained could be seen as being fairly accurate.

The new plan focused more on comparing simulations to wind tunnel data. The methods of validation are detailed below.
5.2 Comparison to previous models

In order to validate the accuracy of the CFD analysis with the conditions that were used, certain previous blunt body experiments were duplicated with Fluent, using grids created from Pointwise. There were two cases in particular that were duplicated before the ability to continue testing was removed. The primary case dealt with a blunt body, with a simple spike protruding from the end. This primary rocket was tested at speeds ranging from Mach 0.8 to Mach 1.5. The other was a transonic-only model [8], which had similar dimensions to the primary rocket but included a dome on the front of the rocket. The dimensions for both the primary blunt-faced rocket and the transonic-only domed rocket are shown below in Table 5.1 and Figure 5.1.

<table>
<thead>
<tr>
<th></th>
<th>(L_b) (mm)</th>
<th>(D_1/D_2)</th>
<th>(L/D_2)</th>
<th>(D_2) (mm)</th>
<th>(D_1) (mm)</th>
<th>(L) (mm)</th>
<th>(R)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Primary</td>
<td>889</td>
<td>0.368</td>
<td>2</td>
<td>127</td>
<td>46.7</td>
<td>254</td>
<td>N/A</td>
</tr>
<tr>
<td>Transonic</td>
<td>889</td>
<td>0.248</td>
<td>0.96</td>
<td>372</td>
<td>92.3</td>
<td>357</td>
<td>0.32 (D_2)</td>
</tr>
</tbody>
</table>

Figure 5.1 Diagram of the rockets

The primary blunt-body rocket (left) and transonic-only domed rocket (right). The primary rocket was tested more due to the simpler shape.

The other case that was duplicated was a hypersonic case with a tipped spike [13]. This rocket was more complex, due to the tipped spike as well as the fact that the dome
on the front of the blunt body did not exactly match that of the rest of the body. The dimensions for the hypersonic rocket are shown in Fig 5.2.

![Hypersonic aerospike surface dimensions (in inches).](image)

Figure 5.2 Hypersonic aerospike surface dimensions (in inches).

In order to be able to use these bodies as axisymmetric models, they were all cut in half. This allowed for Fluent to rotate about the center of each body as the axis of rotation. The dimensions listed are for the full bodies, not the axisymmetric versions. The angle of the slanted portions of the tipped spike was not known, so an estimate was made when creating this rocket. Slight variations can be seen in the dimensions of the tips in the comparisons of the CFD simulation and the physical test in Figure 5.7.

### 5.2.1 Transonic Blunt Body Comparison

A previous experiment [8] for subsonic cases was used as the initial testing for the accuracy of the CFD models.

A model was made to duplicate one of the Mach 0.91 cases in order to see if the flow field from the CFD simulation would match this case. As Figures 5.3 and 5.4 show, the CFD simulation almost perfectly duplicates the flow field around the rocket.
A previous experiment [8] was duplicated in CFD. This is a side by side comparison of the experiments.

For the transonic experiment, the original photo was made transparent and placed over the CFD results. The results are essentially matching exactly with the original experiment, which was very helpful in verifying the accuracy of these tests. The picture on the left is less transparent, in order to show the matching flow fields around the spike. The picture on the right is more transparent in order to show the expansion wave similarity.
A model was made that would be able to duplicate a similar test that used a longer spike, however the case was unable to be run due to the inability to connect to the license servers via HPC².

5.2.2 Hypersonic Blunt Body Comparison

This particular hypersonic model [13] was tested in the NASA Langley Research Center 20-Inch Mach 6 Tunnel. The test section for this tunnel, like many other high speed wind tunnels, is fairly small at 20.5 x 20 inches. There was some concern that the results might be influenced by the relatively small boundaries of the test chamber. As such, two cases were run for this particular test. The first case that was run had a similar sized test section in order to accurately model the previous experiment. The second case had a much larger grid area in order to try to accurately simulate how this model would fare in the atmosphere instead of in a test section.

The dynamic pressure contours for both sizes are shown below in Fig 5.5, and there seems to be a small disturbance in the small test chamber case that would point to the walls having at least some minor impact on the overall results. This is still a relatively minor difference overall, and none of the other results show anything quite as clearly as this particular case. Regardless, it was interesting to see that there was some variation between the two, and helps point to CFD being a very useful aid when used in conjunction with wind tunnel tests.
Figure 5.5  Dynamic pressure comparison of small vs large test sections

This is a direct comparison of the same case, with the picture on the left simulating the small test section and the picture on the right simulating open atmosphere. The small test section shows a faint disturbance away from the body, which would seem to imply that the walls of the test section do have cause some small interference with the test.

While some differing results were expected, in reality the contours for both cases were nearly identical. The assumption for this is that due to the test being run at such a high Mach number, around Mach 6, the shock waves are at a shallow enough angle that they do not spread out far enough for the walls to have any serious impact on the test. Both CFD simulations seemed to match closely with the original test experiments, which, along with the transonic comparison, helps validate the accuracy of the test conditions that have been set up for other cases.

A comparison of the original experiment with the CFD simulation can be seen below in Figs. 5.6 and 5.7. Further comparisons for the hypersonic cases can be found in Appendix B.
A CFD simulation was created to mimic this hypersonic case. The spike tip had a slight difference in shape.

Similar to the transonic case, the photo for the original experiment was made transparent and placed on top of the CFD results. While there is a tiny amount of variance in the flow, the results are very similar to those of the original experiment. Some possible reasons for the small differences could be the slightly different shape of the spike tip, as well as the small area of the test chamber having an effect on the original experiment.
5.3 Primary Tests

At the time these grids were created, there were some problems in trying to get a dome to work on the front of a rocket using any of the grid generation software. This was one of the reasons that a replica of Major Tom was abandoned. This problem with creating a dome was eventually solved later on, which allowed the tests to compare the accuracy of the models to be conducted as mentioned in section 5.2.

Before this issue was solved, however, a simpler model was needed that would implement a simple flat faced blunt body with a plain spike that had no extra features on the tip. Such a model was found in a previous experiment [4]. The majority of the tests used this particular rocket. Due to the straightforwardness of the rocket, it was fairly simple to create a .dat file that had the basic coordinates of the rocket. The major coordinates needed were at the corners of the rocket, and other than that there were no special points that needed to be added. However, the initial rocket included many coordinate points because it was not clear how Pointwise would react to having only a few points to import. Later grids had far fewer points in order to speed up the process of creating and importing the rockets.

The original experiment ran the various rockets from Mach 0.8 to Mach 1.5, and those were the initial cases that were run with the CFD model. Eventually, cases were added that went into higher supersonic speeds. These ranged from Mach 2 to Mach 5, in 0.5 Mach intervals.

5.3.1 Planar vs Axisymmetric

Section 4.3 covered the reason why both planar and axisymmetric cases were used. Seeing the two cases side by side is an excellent way to visualize and understand
“three dimensional relief”. This shows the smaller shock angles and smaller shock standoff distance that would not be possible with the two dimensional planar model. The axisymmetric model seems to do an excellent job of simulating a 3D body, which can be extremely useful if a quick comparison or model is needed and the resources are not available for a full three dimensional model and subsequent test.

5.3.1.1 Planar Results

Initial tests were run using a planar model, due to the fact it was much simpler to set up the simulation using the planar model. It was incorrectly assumed that due to how the grid was made, Fluent would treat these planar cases similar to how it would treat the axisymmetric cases. The initial contours of the planar cases seemed to back this up incorrect assumption, as the shock waves and flow separation were fairly similar to what related experiments had shown. It was not realized until later, with help from the Fluent User’s Guide, that the grid needed to be halved in order to be correctly used as an axisymmetric model. It was only when compared to the separate axisymmetric cases that this assumption was shown to be incorrect.

The flows from the planar cases give very pronounced reactions, especially compared to those of the axisymmetric cases later on, as can be seen below in Figure 5.8. The planar cases have more separation and a stronger bow shock than what would actually occur for this cylindrical body.
Figure 5.8  Contours of dynamic pressure (planar) at Mach 1.5

The dynamic pressure contours shown above for the planar case helps illustrate some of the versatility of Fluent. Due to the case being run as planar, the flow reacts as it would to an infinitely flat plate instead of a cylinder. The flow is much more pronounced in this way than it is for axisymmetric cases.

5.3.1.2 Axisymmetric Results

Initial attempts to make an axisymmetric grid were a complete failure. Regardless of the different options, the axisymmetric cases would always fail. Eventually, using Section 6.3.17 of the Fluent User’s Guide [10], it was realized that the centerline must be set as ‘axis’ type of boundary, and this axis line is what Fluent uses as the line of rotation. Fluent cannot rotate halfway around the center of the grid, which was initially how it was assumed the axisymmetric model worked. Once this point was driven home, it was a fairly simple matter to halve the input coordinates so only the top half was available to import into Pointwise. Since the grid was already created for the full rocket, the only
necessary step was to halve each grid domain that was along the axis, which ran through the center of the rocket. This meant that the majority of domains were left untouched, which made recreating the grid and exporting it to Fluent a painless procedure.

Once the actual axisymmetric grid was imported into Fluent, each case was set up in the exact same way as the previous methods. The only change was setting the bottom of the grid to be an axis type of boundary, and selecting ‘axisymmetric’ instead of ‘planar’ for the type of model. Running the simulations this way gave no problems, and for the most part no more Fluent errors were encountered for the rest of the tests.

The axisymmetric cases experienced a “three dimensional relief” effect that the planar cases did not. This meant that there were smaller shock angles and a smaller shock standoff distance, which for the axisymmetric case was expected. The shock waves and other flow patterns looked much less extreme than the planar cases, since Fluent was actually treating this as the round object it was instead of a flat, infinitely wide object.

These cases were run only from Mach 1.5 to Mach 3, again at 0.5 Mach intervals. Speeds lower than Mach 1.5 were not chosen, as the primary focus was on how the aerodynamic properties varied at supersonic speeds. Results from speeds higher than Mach 3.5 were in another category. Starting with the Mach 4 case, the spike proved to be too long for these speeds. Instead of the flow separating at the tip of the spike, the separation was instead occurring further down along the shaft. There have been multiple experiments that show how detrimental this is to an object’s flight, and these best show at what Mach number this particular rocket configuration would become a.

Contours for the speeds between Mach 1.5 and Mach 3 are the ones that are most helpful for this study. The spike is still short enough at these speeds that it can be
effective for the flight of the rocket. As is expected, the initial shock wave becomes more angled as the speed is increased from Mach 1.5 onwards. At the higher speeds, the initial zone of dead air becomes less pronounced near the tip of spike, possibly foreshadowing the jump backwards away from the tip at an even higher speed, such as with the Mach 4 case.

5.3.2 Results At High Speeds

Below are the various contours for the basic rocket at various speeds. These will mostly be the supersonic cases (> Mach 1.5). It should be noted that these will mostly be the axisymmetric cases contrasted against the planar cases at the same speeds, in order to show just how different the types of models are.

5.3.2.1 Mach 1.5

The Mach 1.5 case is the slowest velocity case for both the planar and axisymmetric models. This more so than any of the higher speed cases seemed to highlight the differences in how Fluent treated the planar and axisymmetric cases. The exaggerated level of the shock waves and other flow properties of the planar model compared to what happens with a cylindrical, three dimensional object are clearly visible. The velocity and static temperature contours for both the planar and axisymmetric case are shown in Figures 5.9 and 5.10 below.
Figure 5.9  Mach 1.5 - velocity contours for planar vs axisymmetric

The planar case shown on the right has a much more pronounced reaction to the flow, compared to the more mild results of the axisymmetric case on the right. This is due to a “three dimensional relief” effect that helps weaken the shock waves.

Figure 5.10  Mach 1.5 - static temperature contours for planar vs axisymmetric

The temperature contours were scaled to the same range for the planar and axisymmetric cases, and the planar case shows a much larger jump. There is also a higher static temperature in the area around the spike in the planar case when compared to that of the axisymmetric case.
5.3.2.2 Mach 3.0

While the Mach 1.5 case is very useful for showing just how different the models are, it is on the lower side of supersonic speeds and the highest level of what could possibly be considered transonic. The Mach 3 cases do not have this possible issue. The speeds are adequate enough for everything one would expect in a supersonic case to appear. The shocks are much more angled, and the rocket has not exceeded the speed that would cause the aerospike (for this particular configuration) to produce unstable flow.

At this point, the planar cases differ greatly from the axisymmetric cases. At Mach 1.5, the planar model shock waves were at a similar angle the axisymmetric models. For higher supersonic and hypersonic cases, the axisymmetric shock waves are much more shallow and the radial gradients of pressure and temperature of the axisymmetric flow are large. This can be seen fairly clearly below, with contours of both total pressure and absolute pressure shown in Figures 5.11 and 5.12 below.

![Figure 5.11 Mach 3 - total pressure contours of planar vs axisymmetric](image)

At high supersonic speeds, the rocket should be passing through the atmosphere with very little impact away from the main body. This is clearly the case in the axisymmetric case shown on the right.
Figure 5.12  Mach 3 – absolute(static) pressure contours of planar vs axisymmetric

These two cases were scaled together to give a better comparison. The planar case on the left is showing extremely large areas of pressure differences, while the axisymmetric case on the right is showing some very small jumps, as would be expected for a rocket flying at three times the speed of sound. Of particular note is the small spot where the pressure fluctuates slightly near the tip of the spike.

5.3.2.3  Mach 4 and Up

The Mach 4 case was the first case to show a clear negative effect caused by the rocket reaching speeds too fast for this particular aerospike length. Figure 5.13 shows a greater disturbance in the total pressure around the rocket at the higher speed. In general, the total pressure made less of an impact as the rocket sped up, but once past the optimal speed it actually makes things worse at increasing speeds. A close picture of the nose for the Mach 4 case is shown in Fig. 5.14, and further comparisons for the Mach 4 vs Mach 5 cases will be included in Appendix E.
Figure 5.13  Mach 4 vs Mach 5 – axisymmetric total pressure

The left figure shows the total pressure contours at Mach 4, and the right picture shows the same for Mach 5. The Mach 4 case is just past the optimal point for this spike length, so the distribution is not as terrible as it could be. However, at the higher Mach number the total pressure distribution only got worse. Shortening a spike would allow this particular configuration to fly smoothly at these speeds.

Figure 5.14  Mach 4 – velocity vectors vs dynamic pressure

The velocity vectors were plotted for the Mach 4 case, and were colored by the dynamic pressure contours in order to show how the velocity and dynamic pressure interact with each other. For this particular case, it can be seen that due to the suboptimal spike length, the flow separation occurs further back than it normally would. The area of flow separation and recirculation is much further back here than it is for slower cases.
CHAPTER VI
CONCLUSIONS

While some of the initial goals of this project were not met due mostly to time constraints, a lot of useful information was still obtained from the various tests and previous research. Some of the difficulties about implementing a spike onto more missiles were made clear, although more research would have helped.

Some of the large issues of the spike seem to be based on length, whether it is too short or too long for the speeds the missile is tested for. While it is easy enough to make sure a spike is long enough, the problem comes in making sure the spike doesn’t extend too far out at high speeds. There are several ways to work on this. One is to do what the Trident I and successor missile did, by implementing a fixed length spike that is slightly shorter than optimal. This allows the spike to have some wriggle room, on the off chance something crazy happens and the rocket exceeds its maximum speed. The spike will not be so long that it disrupts the flow.

Another option is to have an onboard system that will automatically adjust the length of the aerospike based on the speed of the rocket. This would ensure maximum drag reduction throughout the flight, but would potentially be costly in the fact that more electronics are needed that will take up more room. Depending on the purpose and the payload, the extra room for the electronics might not be feasible.
This project began with a somewhat shaky understanding of a lot of the basic concepts of CFD, but even with that foundation a fairly reliable method was found that could compare accurately to previous wind tunnel experiments. There was most likely some human error in setting up the code used for the axisymmetric cases, however these cases still held up against the physical tests conducted in previous years.

While it is not perfect, current grid generation and CFD software and codes are becoming more reliable for being able to accurately simulate complex tests that previously could only be held under certain, less than ideal conditions. The results obtained in this paper only covered certain specialized cases, such as running at zero angle of attack, but these cases were varied enough from each other that the assumption of CFD being fairly accurate seems to be valid.

In particular, Ansys Fluent has shown that it can be both very harsh and quite forgiving to new users. As long as the user is able to obtain a grid of decent quality, they can set up a case in a variety of ways and get similar results. If the user is able to get the initial conditions down correctly, Fluent will be able to accurately predict how a fully three dimensional object will react under certain conditions, even if there is only a two dimensional model available. This would be extremely useful for any quick tests or analysis that might need to be performed where a full three dimensional test may not be plausible.

Seeing the clear contrast between how the planar model and axisymmetric models differ with the same flow conditions really drives home how small changes can make a world of difference. The initial assumption that the planar case would be able to accurately model the flow for an axisymmetric object was clearly incorrect, but until the
actual axisymmetric cases were run there was no way to know this if one did not know what to look for. There were clear shock waves and separations of flow approximately where they needed to be, and the fact that Fluent can differentiate and change the entire flow field for the same grid speaks volumes for the versatility in the software. As mentioned previously, these tests used a fairly limited portion of Fluent’s full scope of features; there are far more functions and complicated models that theoretically could be utilized for a similar level of accuracy.

In conclusion, there have been many experiments over the years that have helped verify how accurate and useful CFD could be compared to possibly more expensive or less ideal physical tests. It may never reach the point of completely replacing physical tests, but it can certainly be used in conjunction with physical experiments to help verify and speed up certain processes. This is just one more example of such an experiment, and leaves the door open for those who may wish to pick up where this left off and reach the initial final goal for the project; namely, in showing why the drag reducing aerospike sees such little use in today’s world.
REFERENCES


APPENDIX A

GRID GENERATION
Figure A.1  4-fin Major Tom mesh via Pointwise

Figure A.2  2D basic rocket grid with blocks via GridPro
Figure A.3   2D basic rocket grid without blocks via GridPro
APPENDIX B

HYPersonic comparisons
Figure B.1  Hypersonic velocity comparisons – small vs large test section

Figure B.2  Hypersonic static temperature comparisons – small vs large test section

Figure B.3  Hypersonic total temperature comparisons – small vs large test section
The static pressure is obviously highest at the front of the rocket. However, due to the extreme speeds of this hypersonic case, the static pressure is so great that any other values cannot be seen with such a scale in place.

Due to the extreme pressures at hypersonic speeds, the actual variations are unable to be seen when compared to the highest pressure values at the tip of the rocket. As such, the left picture has been scaled down to a max range of 7.5e05 Pa, and the right picture has been scaled to 1.00e06 Pa in order to better show the distribution of the static pressure.
As mentioned in Figure A.5, the difference in static pressure is so high at these speeds that only the highest levels of the static pressure can be viewed, unless one scales down so the high pressure is no longer in the picture. These pictures have slightly higher ranges than those of A.5, with the left picture having a max value of 1.50e06 Pa and the right picture having a max value of 5e06 Pa. This is simply to give a better look of how the pressure is reacting away from the tip of the rocket.
APPENDIX C

MACH 1.5 COMPARISONS
Figure C.1  Mach 1.5 dynamic pressure comparisons – planar vs axisymmetric

These figures have the same range for the dynamic pressure, however the planar case shows the pressure level to be much lower in general than the axisymmetric case.

Figure C.2  Mach 1.5 static pressure comparisons – planar vs axisymmetric
The only results that give me pause. The “fingers” in the front of the planar case are very interesting, as is the bleeding effect in the axisymmetric case. This may be one of the cases that is incorrect due to human error.

This is the only case that is extremely similar for both the planar and the axisymmetric cases. One interesting aspect of the planar case is that the “swirl” or fluctuations show up along the rear of the rocket. This was present in all transonic planar cases, and was partially due to the use of the steady state model as well as the planar. The transient case theoretically does a better job of smoothing that out, however, only part of a single transient case was run. Thankfully this did not seem to have an impact on the flow around the spike, and was not an issue at most supersonic speeds or any axisymmetric cases.
Figure C.5  Mach 1.5 – Axisymmetric dynamic vs static pressure

A partially transparent picture of the dynamic pressure contour was placed on top of the static pressure contours, in order to show how the dynamic and static pressure interact with each other when a spike is introduced.
APPENDIX D

MACH 3.0 COMPARISONS
Figure D.1  Mach 3.0 dynamic pressure comparisons – planar vs axisymmetric

Figure D.2  Mach 3.0 static temperature comparisons – planar vs axisymmetric

Figure D.3  Mach 3.0 total temperature comparisons – planar vs axisymmetric
Figure D.4  Mach 3.0 velocity comparisons – planar vs axisymmetric
APPENDIX E

MACH 4 VS MACH 5
Figure E.1  Mach 4 vs Mach 5 – Axisymmetric velocity

Figure E.2  Mach 4 vs Mach 5 – Axisymmetric absolute (static) pressure

Figure E.3  Mach 4 vs Mach 5 – Axisymmetric dynamic pressure
Figure E.4  Mach 4 vs Mach 5 – Axisymmetric static temperature

Figure E.5  Mach 4 vs Mach 5 – Axisymmetric total temperature
APPENDIX F

FLUENT CASES
<table>
<thead>
<tr>
<th>Case No.</th>
<th>Geometry</th>
<th>Planar/Axis</th>
<th>Compressible/Incompressible</th>
<th>Mach</th>
<th>Steady/Transient</th>
</tr>
</thead>
<tbody>
<tr>
<td>1703_1</td>
<td>Primary</td>
<td>Planar</td>
<td>Incompressible</td>
<td>0.8</td>
<td>Transient</td>
</tr>
<tr>
<td>1701_2</td>
<td>Primary</td>
<td>Planar</td>
<td>Incompressible</td>
<td>0.9</td>
<td>Steady</td>
</tr>
<tr>
<td>1701_3</td>
<td>Primary</td>
<td>Planar</td>
<td>Compressible</td>
<td>0.9</td>
<td>Steady</td>
</tr>
<tr>
<td>1701_4</td>
<td>Primary</td>
<td>Planar</td>
<td>Compressible</td>
<td>0.9</td>
<td>Steady</td>
</tr>
<tr>
<td>1701_5</td>
<td>Primary</td>
<td>Planar</td>
<td>Compressible</td>
<td>0.9</td>
<td>Steady</td>
</tr>
<tr>
<td>1701_2</td>
<td>Primary</td>
<td>Planar</td>
<td>Incompressible</td>
<td>1.0</td>
<td>Steady</td>
</tr>
<tr>
<td>1701_3</td>
<td>Primary</td>
<td>Planar</td>
<td>Compressible</td>
<td>1.0</td>
<td>Steady</td>
</tr>
<tr>
<td>1701_4</td>
<td>Primary</td>
<td>Planar</td>
<td>Compressible</td>
<td>1.0</td>
<td>Steady</td>
</tr>
<tr>
<td>1701_5</td>
<td>Primary</td>
<td>Planar</td>
<td>Compressible</td>
<td>1.0</td>
<td>Steady</td>
</tr>
<tr>
<td>1703_3</td>
<td>Primary</td>
<td>Planar</td>
<td>Compressible</td>
<td>1.1</td>
<td>Steady</td>
</tr>
<tr>
<td>1701_3</td>
<td>Primary</td>
<td>Planar</td>
<td>Compressible</td>
<td>1.1</td>
<td>Steady</td>
</tr>
<tr>
<td>1701_4</td>
<td>Primary</td>
<td>Planar</td>
<td>Compressible</td>
<td>1.1</td>
<td>Steady</td>
</tr>
<tr>
<td>1701_5</td>
<td>Primary</td>
<td>Planar</td>
<td>Compressible</td>
<td>1.1</td>
<td>Steady</td>
</tr>
<tr>
<td>1703_3</td>
<td>Primary</td>
<td>Planar</td>
<td>Compressible</td>
<td>1.2</td>
<td>Steady</td>
</tr>
<tr>
<td>1701_1</td>
<td>Primary</td>
<td>Planar</td>
<td>Incompressible</td>
<td>1.2</td>
<td>Steady</td>
</tr>
<tr>
<td>1701_3</td>
<td>Primary</td>
<td>Planar</td>
<td>Compressible</td>
<td>1.2</td>
<td>Steady</td>
</tr>
<tr>
<td>1701_4</td>
<td>Primary</td>
<td>Planar</td>
<td>Compressible</td>
<td>1.2</td>
<td>Steady</td>
</tr>
<tr>
<td>1701_5</td>
<td>Primary</td>
<td>Planar</td>
<td>Compressible</td>
<td>1.2</td>
<td>Steady</td>
</tr>
<tr>
<td>1701_2</td>
<td>Primary</td>
<td>Planar</td>
<td>Incompressible</td>
<td>1.3</td>
<td>Steady</td>
</tr>
<tr>
<td>1701_3</td>
<td>Primary</td>
<td>Planar</td>
<td>Compressible</td>
<td>1.3</td>
<td>Steady</td>
</tr>
<tr>
<td>1701_4</td>
<td>Primary</td>
<td>Planar</td>
<td>Compressible</td>
<td>1.3</td>
<td>Steady</td>
</tr>
<tr>
<td>1701_5</td>
<td>Primary</td>
<td>Planar</td>
<td>Compressible</td>
<td>1.3</td>
<td>Steady</td>
</tr>
<tr>
<td>1701_3</td>
<td>Primary</td>
<td>Planar</td>
<td>Compressible</td>
<td>1.5</td>
<td>Steady</td>
</tr>
<tr>
<td>1701_4</td>
<td>Primary</td>
<td>Planar</td>
<td>Compressible</td>
<td>1.5</td>
<td>Steady</td>
</tr>
<tr>
<td>1701_5</td>
<td>Primary</td>
<td>Planar</td>
<td>Compressible</td>
<td>1.5</td>
<td>Steady</td>
</tr>
<tr>
<td>R05M09T01</td>
<td>Transonic</td>
<td>Axisymmetric</td>
<td>Compressible</td>
<td>0.91</td>
<td>Steady</td>
</tr>
<tr>
<td>Test</td>
<td>Supersonic Cases</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>-------</td>
<td>------------------</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1701_3</td>
<td>Primary Planar Compressible 1.5 Steady</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1701_4</td>
<td>Primary Planar Compressible 1.5 Steady</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1701_5</td>
<td>Primary Planar Compressible 1.5 Steady</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1701_1</td>
<td>Primary Planar Unknown (Comp) 2.0 Steady</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1701_5</td>
<td>Primary Planar Compressible 2.0 Steady</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1701_5</td>
<td>Primary Planar Compressible 2.5 Steady</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1701_1</td>
<td>Primary Planar Compressible 3.0 Steady</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1701_5</td>
<td>Primary Planar Compressible 3.0 Steady</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1701_5</td>
<td>Primary Planar Compressible 3.5 Steady</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1701_1</td>
<td>Primary Planar Compressible 4.0 Steady</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1701_5</td>
<td>Primary Planar Compressible 4.0 Steady</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1701_5</td>
<td>Primary Planar Compressible 4.5 Steady</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1701_5</td>
<td>Primary Planar Compressible 5.0 Steady</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1701_5</td>
<td>Primary Planar Compressible 5.0 Steady</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1801_1</td>
<td>Primary Axisymmetric Compressible 1.5 Steady</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1801_1</td>
<td>Primary Axisymmetric Compressible 2.0 Steady</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1801_1</td>
<td>Primary Axisymmetric Compressible 2.5 Steady</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1801_1</td>
<td>Primary Axisymmetric Compressible 3.0 Steady</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1801_1</td>
<td>Primary Axisymmetric Compressible 3.5 Steady</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1801_1</td>
<td>Primary Axisymmetric Compressible 4.0 Steady</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1801_1</td>
<td>Primary Axisymmetric Compressible 4.5 Steady</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1801_1</td>
<td>Primary Axisymmetric Compressible 5.0 Steady</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Test_04</td>
<td>Hypersonic Large Axisymmetric Compressible 6.06 Steady</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Test_05</td>
<td>Hypersonic Small Axisymmetric Compressible 6.06 Steady</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>