Variable-Length, Drag-Reducing Aerospike

Alder Louviere[1]
Matthew Rickman[2]
Jake Peterson[3]

Mississippi State University, Mississippi State, MS 39762

Drag reduction is an always important part of aerospace engineering to get the most efficiency out of a design, especially if that design can be optimized for a wide range of velocities. One such design that could be optimized for various speeds is a blunt body with a drag-reducing aerospike, which is based on a possible high-powered rocket design. Using CFD, this paper tries to find the optimal length-to-body diameter ratios of a blunt body aerospike for each flow state and then establishes a design for a physical model that utilizes the ratios found in CFD to vary the aerospike length with velocity, which results in more efficient flight profile. Through simulations, it is shown that a variable-length aerospike can reduce the drag coefficient compared to the reference body by up to 50%.

Nomenclature

CFD = Computational Fluid Dynamics

I. Introduction

THE drag-reducing aerospike is a device commonly used on the nose of aerodynamic vessels in supersonic conditions. The aerospike reduces drag by separating flow from the main body of the aircraft, creating a detached shock ahead of the body. This detached shock leaves room for the air to circulate between it and the forebody, creating a streamlined flow profile similar to flow on a nosecone that reduces drag. Aerospikes offer a few advantages over a blunt body, as they excel in supersonic flows (Mach 1.2+), can reduce heating due to friction from aerodynamic forces, and have the ability to create multiple small shocks instead of one large one. However, these benefits are heavily tied to the length of the spike and the speed of the vehicle, so their adaptability is limited and are best suited for highly specific use cases. This project attempts to investigate the feasibility of an adjustable drag-reducing aerospike that could vary its length and/or diameter to optimize performance for a range of free-stream velocities, specifically transonic to low supersonic (Mach 0.8 - 2.0). This aerospike length and/or diameter optimization project was designed around a high-powered rocket manufactured by The Space Cowboys at Mississippi State University.

II. Methods

The method for analyzing the aerospike's performance is by using CFD to run simulations studying the flow around an aerospike attached to a blunt body. The CFD program used in this experiment was ANSYS Fluent as it is a good platform to generate multiple spikes and test them all in multiple conditions. A baseline geometry without an aerospike was created in Fluent and analyzed under different flow conditions to serve as the control for this experiment. This created a reference value for the aerospikes to be tested and compared against, as well as ensuring that error would be minimized as all of the spikes would be added to the reference geometry so the only changing variable was the spike length.

When using ANSYS Fluent, there are 4 main steps to the process; geometry, mesh, set-up, and results. Each step builds into the next, and the steps help allow the user to set up a CFD simulation easily. The axisymmetric reference geometry consisted of a 3-inch radius cylinder with a length of 100 inches which sat in a cylindrical chamber that

[1] Senior, Department of Aerospace Engineering, Mississippi State University, Undergraduate Team, AIAA Student Member
[2] Senior, Department of Aerospace Engineering, Mississippi State University, Undergraduate Team, AIAA Student Member
[3] Senior, Department of Aerospace Engineering, Mississippi State University, Undergraduate Team, AIAA Student Member

extended past the reference body by 24 inches in each direction and by 48 inches above the axis. This geometry was created using ANSYS Design Modeler, which is one of the native ANSYS modeling tools. Next, the geometry was imported into the meshing step, and the mesh was configured based on the following parameter; a maximum mesh element size of 0.2 inches for the overall geometry. After the mesh was generated, the faces of the mesh were added to named selections, which allowed ANSYS to select which face does a certain job in the set-up phase.

Next, the mesh was finalized and then transferred into the setup step of the process, which is the CFD calculation part of the ANSYS Fluent block in ANSYS. Multiple parameters were set in the solver, particularly the turbulence model being set to K-omega (with compressibility effects turned on) and having energy effects enabled as well. The next major change was modifying the default fluid material properties for air. Density was set to “Ideal Gas” and viscosity was set to “Sutherland” as both changes allow for the flow to function more accurately at higher speeds and temperatures. The next step, which is the most critical to collecting accurate data, is to set the reference values correctly based on the input geometry and fluid flow parameters. Once that is done, the convergence criteria are set. Convergence criteria tell the solver the confidence level you want to reach based on a certain level of accuracy, with higher accuracy taking exponentially longer time or much more computing power to achieve. The final setup step is initializing the solver and then running the solver to a specified number of iterations or the convergence criteria, whichever comes first. During these iterations, the solver tries to solve the given turbulence model for each element at the same time, which for a dense mesh, can take a long time since the flow needs to be continuous between elements to be accurate. In addition to solving the fluid equations, the solver also calculates the drag force of the reference body using the reference values and the calculated flow, and this value converges to accuracy alongside the converging turbulence model.

After the solver finishes calculating, the output and calculated data can be seen in the results step. The flow can be represented by several parameters such as velocity, pressure, and density. This can then be visualized by a contour plot, streamlines, or both. An example of the resulting image can be seen below in Figure 1. The calculated data, like the drag coefficient of the body, can be exported from the results page as well and is then used to compare against a known reference or used in another way.

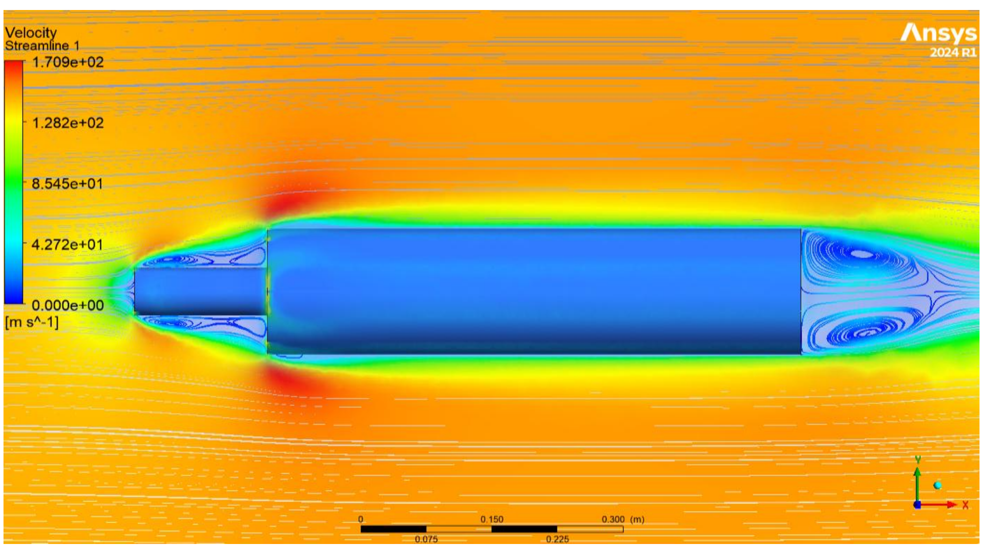

Figure 1. Velocity streamlines for a blunt body with an aerospike model at Mach 0.5.

In Figure 1 above, the streamlines on the aerospike on the leading edge, or left side, of the body show that the flow over the body circulates between the spike and leading body edge, which allows the following flow to coast over the circulation and reconnect onto the body. The spike in Figure 1 is also too long for the flow condition, which is observed by the flow separating and then rejoining onto the spike, resulting in increased drag and a less efficient flow onto the body, which could be counteracted with a shorter spike.

Using the previously described steps in ANSYS Fluent, we analyzed the base body with and without an aerospike at 5 different Mach values. These values were Mach 0.8, 1, 1.25, 1.5, and 2.0. At each Mach, the aerospike length, or how far it extrudes from the main body, was run through a sweep of values to find the optimal aerospike length to overall body drag ratio. Alongside this data, reference values were computed for a body with no leading aerospike at the same Mach numbers to quantify the drag reduction from the aerospike.

Computational data is excellent, but it often paints an incomplete picture. It is great for cheaply and efficiently testing models, but it will always be an idealized process, and some variables and details will be omitted. This is why experimental testing is important, and it is recommended to be performed if necessary and feasible. The current plan for once an optimal spike geometry has been selected is to create a system that can reliably move the spike inside and out of the body of the rocket. However, preliminary analysis found that the rocket reaches peak speed so quickly that extending the spike would have to be almost instantaneous and not wildly beneficial to reducing drag since it would only be optimizing the flight for such a short amount of time, so the spike will likely start in the extended position and be designed to be retracted into the body as it loses speed during flight. This process would likely involve a linear actuator or servo motor to provide smooth movement and be designed in a telescoping fashion to allow for easier retraction. A backup design using springs could also be considered.

The spike itself will be a unibody, linear spherical cylinder for simplicity as testing all the nuanced details of the spike like tip shape and varying diameters would take far too long to be comprehensive and would be beyond the scope of this experiment. However, spikes with different geometries such as discs and stepped aerospikes as can be seen in Figure 2 were analyzed while researching spike geometries as they do provide an interesting alternative in the goal to create a more adaptable and versatile aerospike model.

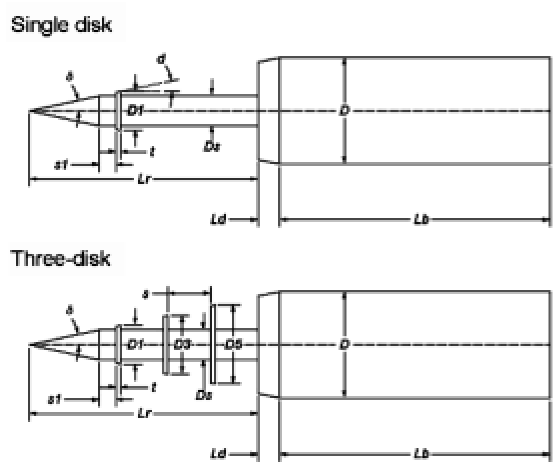

Figure 2. Schematics for a telescoping aerospike from a previous study.

III. Results

The ANSYS simulations show that attaching a 1-inch diameter aerospike to a 6-inch diameter body helps drastically reduce drag at different velocity profiles. For instance, at Mach 0.8, a 9.2-inch aerospike cuts drag by 48% as seen in Figure 3. At Mach 1.0, a 10-inch aerospike drops it by 39% as seen in Figure 4, and at Mach 1.25, a 10.4-inch aerospike gives the biggest reduction at 50% as seen in Figure 5. As speeds go up, a 9.4-inch aerospike at Mach 1.5 reduces drag by 52% as seen in Figure 6, and at Mach 2.0, a 9.6-inch aerospike drops it by 54% as seen in Figure 7.

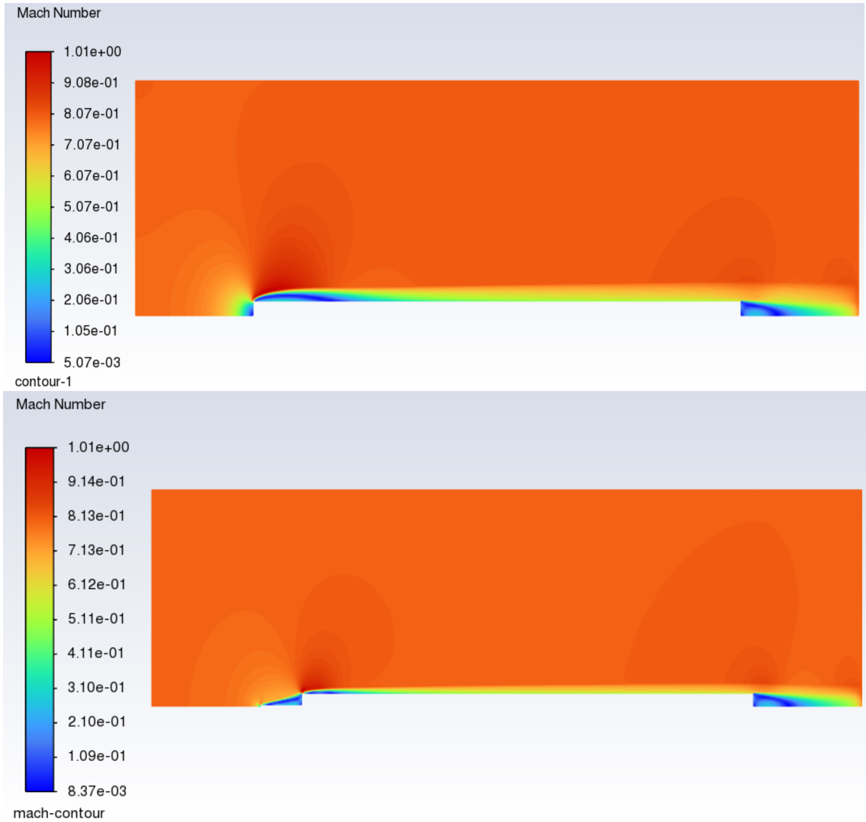

Figure 3. Mach Contour for Reference Body (top) and Optimized Spike Length (bottom) in Mach 0.8 Flow

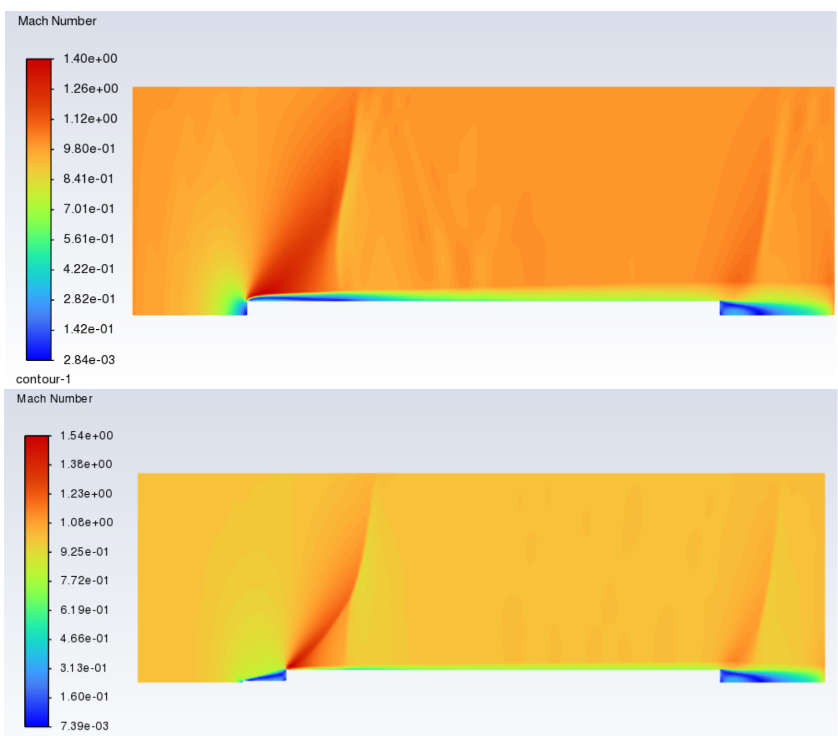

Figure 4. Mach Contour for Reference Body (top) and Optimized Spike Length (bottom) in Mach 1.0 Flow

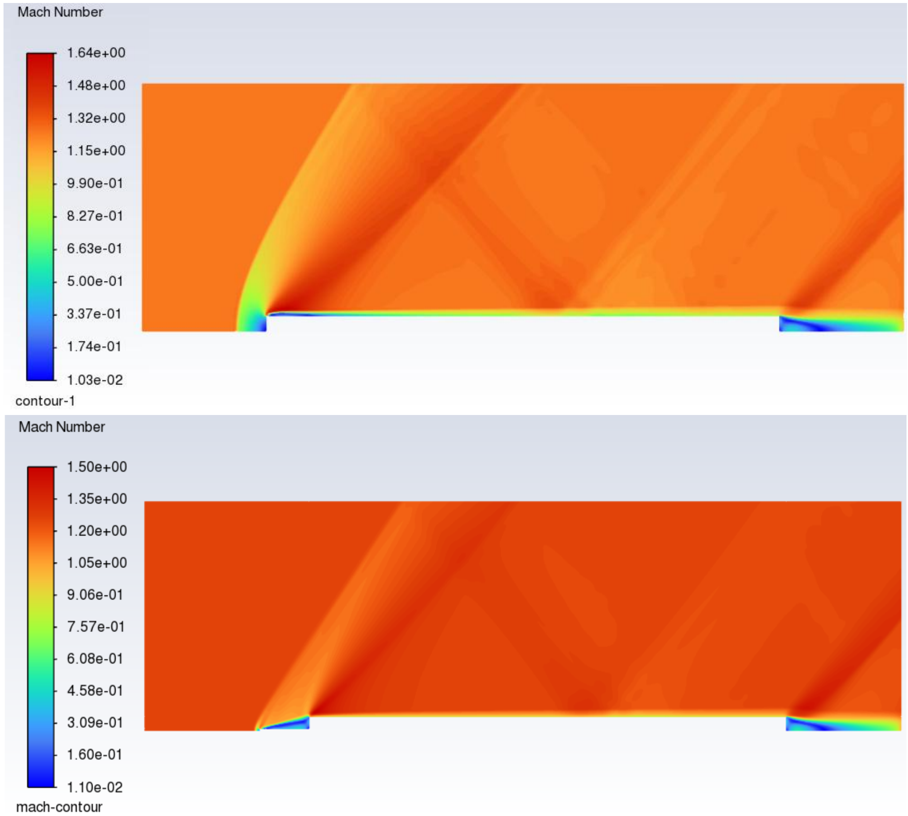

Figure 5. Mach Contour for Reference Body (top) and Optimized Spike Length (bottom) in Mach 1.25 Flow

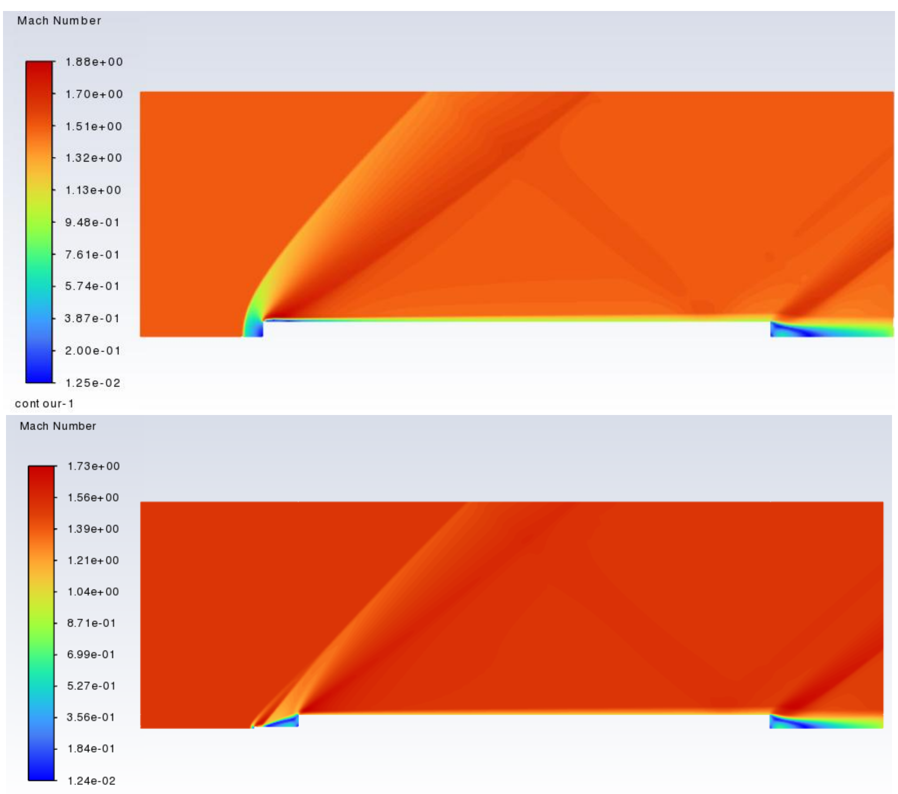

Figure 6. Mach Contour for Reference Body (top) and Optimized Spike Length (bottom) in Mach 1.5 Flow

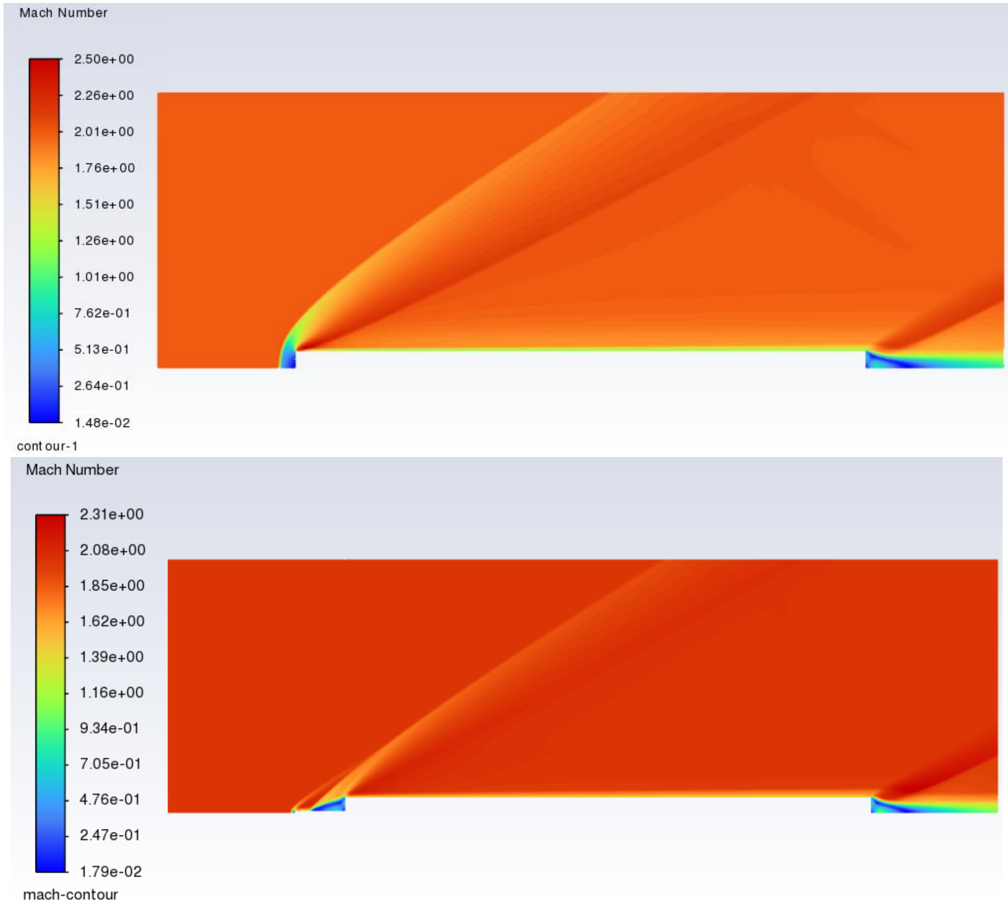

Figure 7. Mach Contour for Reference Body (top) and Optimized Spike Length (bottom) in Mach 2.0 Flow

Figure 8 shows a side-by-side comparison of the drag coefficients of the reference body and of the geometry with an optimized spike length. As stated previously, the drag coefficient of the body with the optimized spike is cut approximately in half at each Mach number compared to the reference. From the drag coefficient vs Mach plot, a spike length vs Mach number plot can also be derived as shown in Figure 9. Depending on the flight envelope of the rocket that the aerospike would be attached to, a length vs time plot would show which length the aerospike needs to be at during each section of the flight.

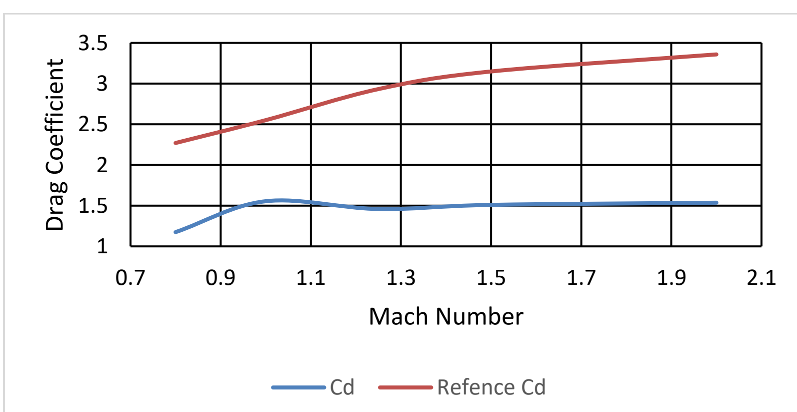

Figure 8. Drag Coefficient vs Mach Number

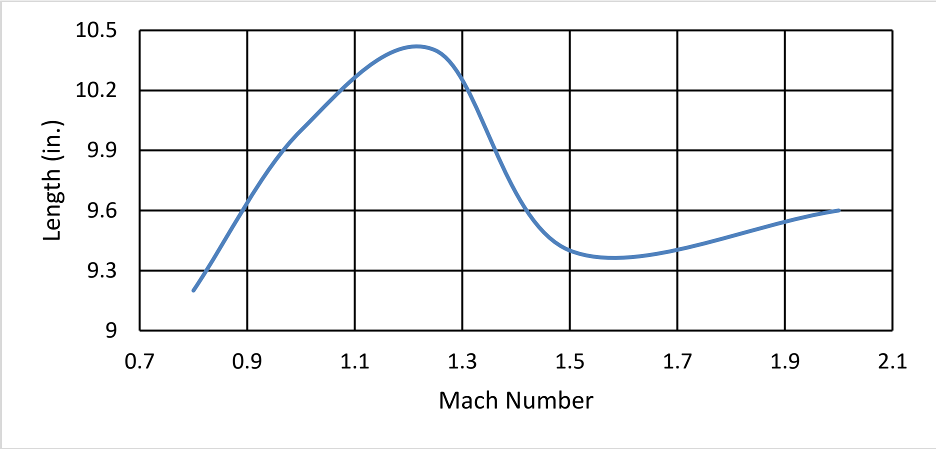

Figure 9. Optimum Length vs Mach Number

A telescoping system could change the length of the aerospike depending on the Mach number at the time. The system could track the speed and automatically adjust the aerospike to the optimal length for each situation. For example, it could extend to 9.2 inches at Mach 0.8, 10 inches at Mach 1.0, and so on. This would make the vehicle more aerodynamic and reduce drag over a range of speeds, improving performance and fuel efficiency. Further drag reduction could be expanded upon when viewing the effects of a changing diameter along with a changing aerospike length but due to computing power and time constraints, we were unable to accomplish this task.

IV. Conclusion

This study has demonstrated the drag-reducing qualities of aerospikes using CFD simulations. Further research and experimentation include examining other geometries for the aerospike, constructing physical models to be launched, and widening the covered velocity profile to create a more complete data set. Applications of this research include high-powered rockets, missiles, and potentially hypersonic aircraft. However, practical considerations such as the mechanical complexity, weight of the adjustment mechanism, and structural integrity under high-speed conditions must be carefully addressed. The resulting design would balance aerodynamic advantages with the challenges of incorporating a movable system, offering significant improvements in fuel efficiency and overall vehicle performance.

Acknowledgments

We thank Dr. Shreyas Narsipur and Dr. Keith Koenig for their continued support of our research as our faculty advisors. Their dedication to the Aerospace Engineering Department at Mississippi State University goes beyond words and many people would not be where they are today without them. We also thank the National Center for Physical Acoustics and Mr. Stephen Perry for their help with the initial stages of the project and their hospitality on the tour they graciously gave us.

References

ANSYS, Inc., *ANSYS Fluent User's Guide*, version 2024, ANSYS, Inc., 2024.

Kobayashi et al. (2007). Experimental Study on Aerodynamic Characteristics of Telescoping Aerospikes with Multiple Disks. Journal of Spacecraft and Rockets.