Computational Fluid Dynamics Analysis of Aerodynamic Drag on a 3-Inch High-Power Rocket

Chyler Bitsoi
2 February 2026
Active Rocket Control
ME 486C - Section 001

Fall 2025-Spring 2026

Introduction:

High-power rockets experience significant aerodynamic loads in-flight that influence vehicle stability, loadings, and performance. The ME 486C capstone project is in the process of developing an active rocket control (ARC) system for a 3-inch diameter rocket, so appropriately determining the external aerodynamic performance of the finalized rocket geometry will be important for this project. Aerodynamic drag loads in particular influence the altitude the rocket can achieve, the control authority of the fins and systems that are integrated into the airframe, and the way in which those loads will be distributed into the fin structures, their actuators, and the airframe.

Aerodynamic loadings were estimated early in the design process using simple analytical models and drag coefficients. Such methods may be useful to initially size the vehicle, since they have relatively simple requirements for input data, but they also assume ideal flow characteristics and fail to accurately model the complex flow around an actual rocket with fins, nose geometry, and fin aspect ratios. With a final geometry now chosen, however, a more rigorous approach should be taken to ensure that loadings are reasonable and that the chosen vehicle design will not encounter difficulties under the influence of those loads.

This report presents a CFD analysis of the external aerodynamics of the finalized 3-inch rocket airframe, as performed using ANSYS Fluent software. The analysis intends to provide detailed velocity, pressure, and drag information, as well as information on the wakes that the rocket and its fins will generate. The outcome of this CFD analysis will determine if the current design can maintain reasonable aerodynamic profiles and drag effects, and it will help to guide its structural design and active control system decisions.

Assumptions:

The following assumptions were used during this CFD analysis to maintain physical relevance to the intended flight conditions of the rocket.

a. The flow is assumed to be in a steady state, so the transient effects during flight are neglected.
b. Air is modeled as an incompressible fluid, as our expected max velocity using rocksim data is expected to be below mach one.
c. The flow is assumed to be fully turbulent, and its effects are modeled using Reynolds-averaged navier strokes approach.
d. Properties like air, viscosity, density were assumed to be constant throughout the simulated flight.
e. The rocket walls are modeled as a no slip wall condition, with zero velocity at the solid fluid interface.
f. Thermal effects, like heat transfer and temperature variation are assumed to be negligible since our simulated flight is not within supersonic speeds.
g. Gravitational effects are negligible, since they do not significantly affect external aerodynamics.
h. The rocket is assumed to be rigid, and deformation under load is negligible.

Such flow assumptions and turbulence modeling approaches are consistent with the standard CFD analysis methods used for analyzing the aerodynamic performance of this type of vehicle. References: [1]–[3]

Governing Equation:
The external aerodynamic flow field around the rocket is described by the conservation of mass and momentum. For incompressible flow, conservation of mass is described by the continuity equation given in Eq. (1), which describes the requirement that the divergence of the velocity field is zero.

Conservation of momentum is given by the RANS equations, shown in Eq. (2), which represent the inertial, pressure, and viscous forces acting on the fluid. The extra term for Reynolds stress represents the influence of turbulence due to velocity fluctuations, and is modeled using a closure model for turbulence in ANSYS Fluent. The RANS approach enables turbulent flow to be represented using time-averaged quantities rather than resolving instantaneous fluctuations.

$$\nabla \cdot V = 0 \tag{1}$$

$$\rho\,(V \cdot \nabla)V = -\nabla p + \mu \nabla^2 V - \rho \nabla \cdot (u'u') \tag{2}$$

Computational Method and model:
The analysis of external aerodynamics was carried out in ANSYS Fluent to model the external flow around the 3-inch rocket's final geometry. The computational domain for the model consists of the complete rocket body, nose cone, and fins to ensure that the separation of flow, wake region, and pressure effects are accurately modeled. The 3-inch rocket geometry modeled in ANSYS Fluent is depicted in Figure 1.
An external flow computational domain was created around the rocket geometry to simulate the free stream. A velocity inlet boundary condition was applied to the inflow boundary of the domain, while a pressure outlet condition was used at the outflow boundary. The surface of the rocket was modeled as a no-slip wall to impose a zero velocity condition at the boundary between solids and fluids. The computational domain surrounding the rocket was sized adequately to prevent any interference by the boundaries with the physical conditions (e.g. pressures) within the domain.
The ANSYS Fluent solver used to solve the governing equations was a pressure-based, steady-state solver. The turbulence effects were modeled using a Reynolds-Averaged Navier–Stokes (RANS) approach that utilizes a two-equation turbulence model to account for turbulence mixing and wake flows.
A hybrid mesh containing areas of differing cell sizes (i.e., with localized mesh refinement) was used in regions surrounding the rocket body. This was done so that sufficient accuracy was provided in areas around the rocket, and specifically in the wake region downstream of the rocket.
The convergence of the ANSYS Fluent solution was monitored using residuals from each governing equation. These included continuity, momentum, and turbulence equations. The residuals were required to stabilize before being extracted from the simulation once it had completed running. Convergence was also monitored through histories of forces acting on the rocket geometry.

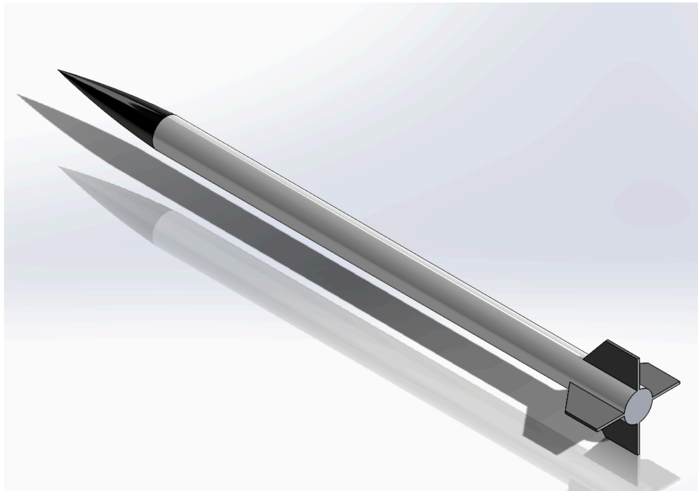

Figure 1:

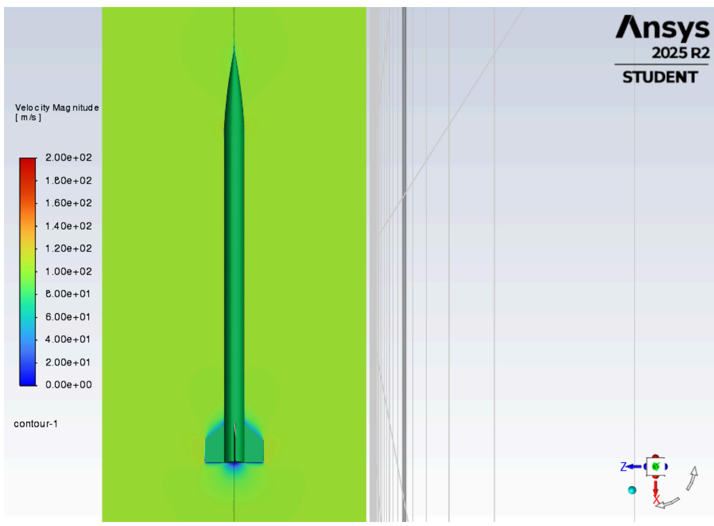

Figure 2:

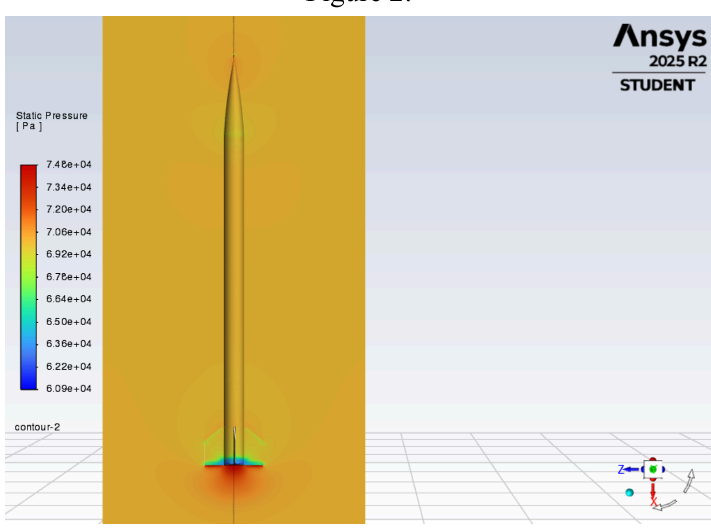

Figure 3:

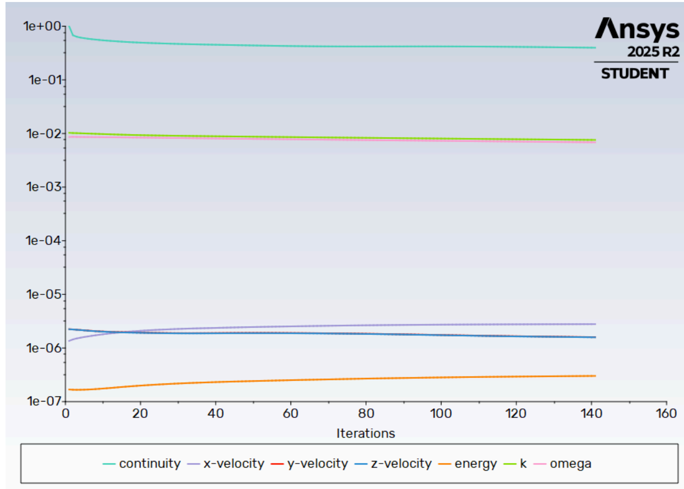

Figure 4:

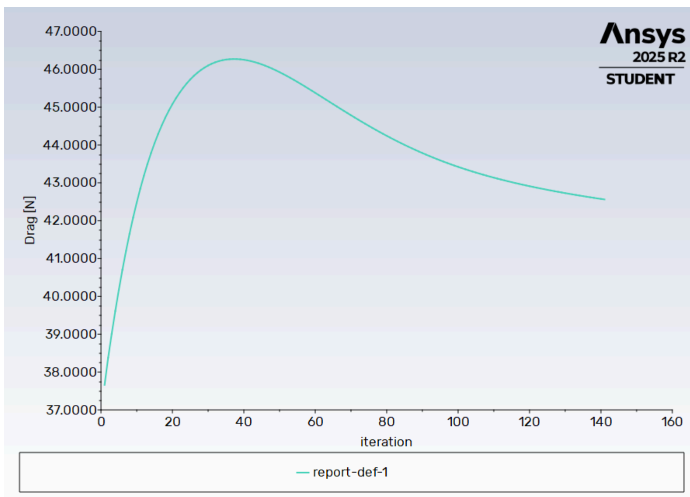

Figure 5:

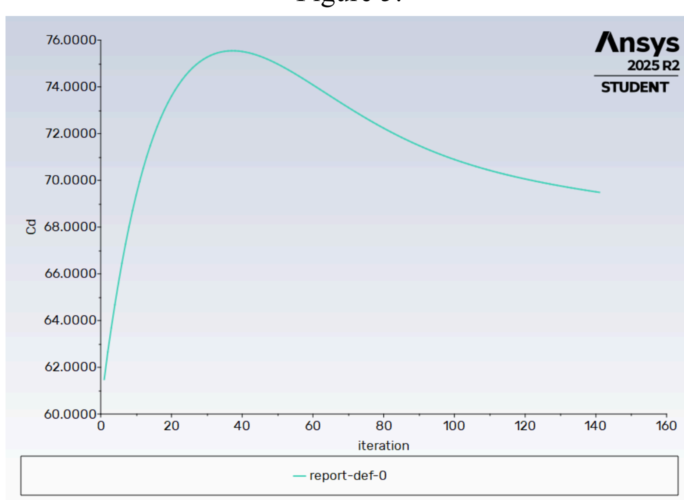

Figure 6:

Results/Discussion/Impact:

The velocity magnitude contours, shown in Figure 2, show that the velocity value is higher around the nose and fins of the rocket. The velocity value is much lower behind the rocket, creating a wake for some lengths behind the rocket. The static pressure contours are shown in Figure 3. The stagnation pressure value is very high at the nose of the rocket but lower along the body and the fins of the rocket. At the base of the rocket is a low-pressure region that indicates the base drag acting on the rocket. These drag regions indicate that there is a pressure difference between the nose of the rocket and its base, leading to a significant amount of drag acting on the rocket. The contour values of the pressure are expected values for subsonic flow.

The solution has fully converged to steady-state after 100 to 150 iterations, as seen in the residual value graphs from Figure 4. After reaching these residual values and ensuring that the solution has fully converged, the drag and lift force values were used to determine the stability of the solution after many iterations were performed. Figure 5 shows the drag force values as the solution converges. The drag force increases rapidly before it reaches its peak value of approximately 45 to 47 N; however, the drag force eventually converges to a constant value between 42 and 43 N. As seen in Figure 6, the lift force values as the solution converges begin oscillating before it reaches its peak lift force of approximately 75 N before finally stabilizing at a constant value between 69 and 70 N once the solution fully converges. The value of the drag and lift forces have both stabilized, which ensures that the solution has reached steady-state convergence. The magnitude of drag force makes sure that the aerodynamic loads acting on the rocket are well within acceptable limits to complete the rocket's flight trajectory. This validates this current rocket geometry, further allowing the understanding of how to optimize this current design in any future iterations of this rocket model, especially by knowing the pressure and velocity distribution values.

Conclusion:

Through the use of CFD analysis within ANSYS Fluent, an analysis of the aerodynamics of the rocket was able to be performed, providing both qualitative and quantitative insight into the aerodynamics that are exhibited by the rocket. While the results of these simulations indicated that the magnitudes of the aerodynamics that were exhibited by the rocket is not consistent with those that are expected for a rocket of the size and flight speed of the modeled rocket, such limitations indicate that further iteration of the solution is required in order to achieve full convergence of the CFD solution. Thus, while the results indicate that further iteration is required in order to increase the accuracy of those results, this type of analysis helps to provide insight into the correct CFD setup for those analyses. The techniques and processes developed through this effort can be indicative of a correct CFD setup, and further provide valuable experience with the use of these types of CFD analyses. Overall, then, these types of analyses provide a baseline through which more accurate results may be obtained, as well as future studies that are to be performed on this project.

Reference:

[1] ANSYS, Inc., *ANSYS Fluent Theory Guide*, Release 2023 R1, Canonsburg, PA, USA, 2023.

[2] J. D. Anderson, *Fundamentals of Aerodynamics*, 6th ed., New York, NY, USA: McGraw-Hill Education, 2017.

[3] F. M. White, *Fluid Mechanics*, 8th ed., New York, NY, USA: McGraw-Hill Education, 2016.

[4] P. A. Durbin and B. A. Pettersson Reif, *Statistical Theory and Modeling for Turbulent Flows*, 2nd ed., Hoboken, NJ, USA: Wiley, 2011.