CFD-Based Aerodynamic Modeling and Validation of Airbrake Integration for Apogee Control in a Sounding Rocket

Bhavya Chaudhary*
Undergraduate Student, College of Engineering, Athens, GA, AIAA Member ID: 1866806

Austin Applestone†
Undergraduate Student, College of Engineering, Athens, GA, AIAA Member ID: 1924172

Jason Rupram‡
Graduate Student, College of Engineering, Athens, GA, AIAA Member ID: 1785154

Brandon Hour§
Undergraduate Student, College of Engineering, Athens, GA, AIAA Member ID: 1924171

This paper presents the analysis on the University of Georgia Rocketry Team's 10,000 ft Commercial-Off-The-Shelf (COTS) entry in the International Rocket Engineering Competition (IREC). This year, the primary focus has been on the integration and deployment of air brakes, which is difficult to model in current simulation software, such as OpenRocket, due to complicated turbulent flow. To validate the precision of the fluid flow over the airbrakes, a mesh convergence study was performed. The method used was the convergence curve, where the critical result parameter of a simulation was plotted against the mesh resolution. The criteria for convergence was set at 1%, so a change in the result parameter of 1% or less for subsequent iterations meant that convergence was achieved. A CFD model, using ANSYS Fluent with the k-omega SST turbulence model, was developed to analyze the effectiveness of air brakes in controlling the rocket's velocity, trajectory, and apogee during flight by accurately capturing high-speed flow characteristics.

Nomenclature

cd	Coefficient of Drag
ρ	Density
P	Pressure
R	Characteristic Gas Constant
T	Temperature
c	chord
μ	Dynamic Viscosity
μ_t	Shear Velocity, describes shear related motion between moving fluids
τ_w	Shear wall stress
R_{ex}	Reynolds Number
C_f	Coefficient of Skin Friction
U_∞	Freestream Velocity
k	Turbulence Kinetic Energy
ϵ	Turbulence Dissipation
ω	Specific Turbulence Dissipation
$k-\omega, k-\epsilon$	Standard two equation turbulence models
u'	Root-Mean-Square of the Turbulent Velocity Fluctuation
$SST\ \ k-\omega$	Shear Stress Transport turbulence model

I. Introduction

The International Rocket Engineering Competition (IREC) is one of the world's largest student-built rocket competitions, connecting students from across the United States and around the globe. IREC is built upon motivating and encouraging students to pursue science, technology, engineering, and math (STEM) based careers, allowing students to develop real-world engineering problems through designing, building, and flying multistage rockets. The University of Georgia (UGA) has designed, built, and launched IREC-ready rockets for three years with different design focuses. This year, the primary focus has been on the integration and deployment of air brakes. IREC challenges teams to reach a target altitude of 10,000 ft, and as a result, teamwork and engineering innovation are required to achieve precise apogee control, ultimately determining the competition scoring.

A background study of the rocket's speed, altitude, and drag calculations was conducted through OpenRocket software. OpenRocket is a free software for student organizations and hobbyists for designing, simulating, and analyzing the flight path of model rockets. The ability to simulate a flight path of the model rocket, providing the weights of the internal components, motor information, and model geometry, provides data for max apogee, max velocity, center of pressure, center of gravity, as well as stability information that can be used as preliminary values for further CFD calculations. OpenRocket uses a 6 Degree of Freedom Runge-Kutta-4 method, which provides accurate correlations to real-world flight dynamics. This method is great for its high accuracy and moderate computational time, but can struggle with highly oscillatory/chaotic systems. The air brakes cause a lot of turbulence and flow separation, which creates a highly chaotic flow form and causes OpenRocket's software to be disrupted. This limitation of OpenRocket further amplifies the need to conduct higher fidelity simulations within Ansys Fluent.

II. Relevant Literature Review

The paper, *Improved Two Equation $k-\omega$ Turbulence Models For Aerodynamic Flows*[1], tested a baseline (BSL) model that combined the standard two equation models, $k-\omega$ and $k-\epsilon$, and a Shear-Stress Transport model on various flowfields, including flat plate boundary layers, free shear layers, and complex adverse pressure gradient flows. The SST model's predictions were independent of freestream values and demonstrated superior accuracy when compared to experimental data for adverse pressure gradient boundary-layer flows, particularly in predicting flow separation.

The research presented in Menter's technical memorandum is highly relevant to the computational fluid dynamics(CFD) analysis of rocket airbrakes. Deployable aerodynamic drag devices fundamentally rely on the controlled generation of strong adverse pressure gradients, massive flow separation, and complex wake structures. Menter's development of the Shear-Stress Transport (SST) model directly addresses the historical limitations of turbulence modeling in these exact aerodynamic regimes. As such, the SST $k-\omega$ model was used in ANSYS Fluent for the analysis of the viscous flow over the surface of the rocket and airbrakes.

III. CFD Setup

The preliminary design of the rocket was modeled in OpenRocket to get the maximum velocity of the rocket using the M3400 motor configuration, which was determined to be Mach 0.985. This means that the freestream velocity of the air flow around the rocket is in the transonic region. Fluid flow analysis in the transonic region is particularly difficult due to the complex, non-linear, and mixed-flow phenomena where supersonic and subsonic zones coexist.

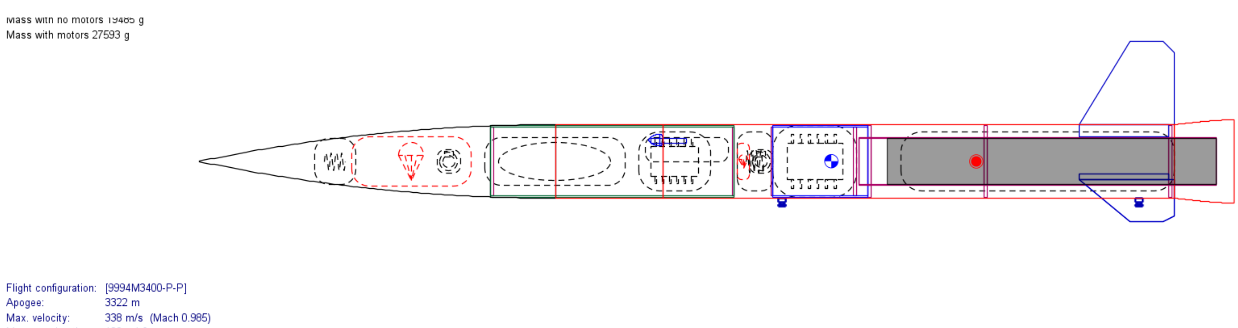


Fig. 1 OpenRocket Model

The airbrakes are designed to be deployed after motor burnout to ensure that the change in center of pressure does not destabilize the rocket in flight: motor burnout causes the center of gravity to shift towards the forward end due to the reduction in weight from the loss of solid fuel, which causes the rocket to become more stable during flight. Using this constraint, the analysis was simplified to Mach 0.7 subsonic conditions at 5000 ft altitude for the airbrake analysis, which occur sufficiently after motor burnout as seen in Figure 2.

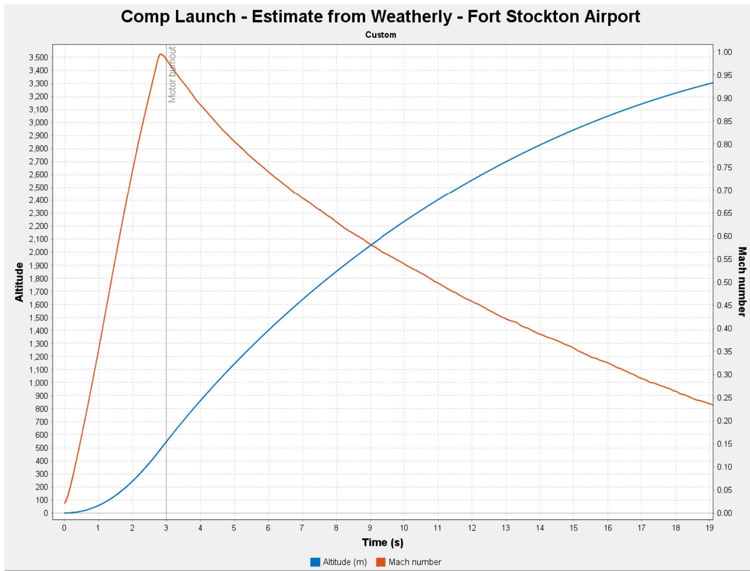


Fig. 2 Time Dependent Mach Number and Altitude During Flight

A. Geometry

To analyze the provided flow behavior, a pressure farfield boundary condition was used in ANSYS Fluent to model free-stream conditions, with free-stream Mach numbers and static conditions being specified. The farfield boundary is set up using a pill-shaped fluid domain, where the surfaces of the rocket act as walls. The pill shape was chosen to improve the cell locations to be optimized closer to the rocket while reducing the overall number of cells compared to a perfect sphere geometry to streamline calculation times. The enclosure was extended to 10 body lengths towards the front end of the rocket and 5 body lengths towards the aft end to ensure that the flow has stabilized once it has reached the rocket.

A body of influence that extends 2.5m towards the aft end of the rocket, and 0.85m towards the front end, was also added to further refine the mesh in the region of interest. The body of influence completely encapsulates the fins and airbrakes. Named selections were made for the nose cone aluminum tip, body tube, tailcone, fins, and

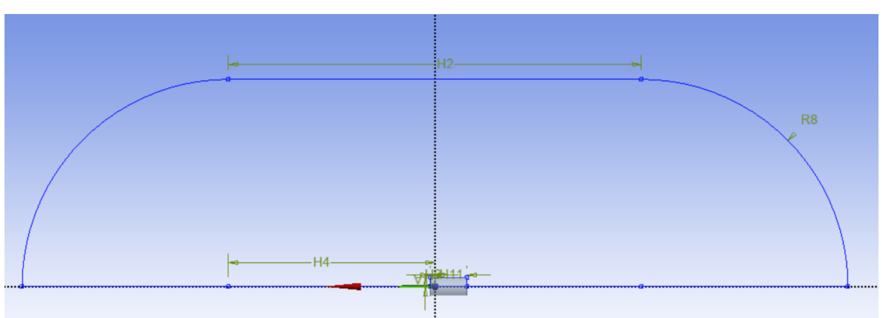


Fig. 3 Farfield Enclosure Sketch

B. Mesh Generation

1. Governing Equations

$$\Delta s = \frac{y^+ \mu}{\mu_t \rho} \tag{1}$$

$$\rho = \frac{P}{RT} \tag{2}$$

$$\mu = \mu_0 \left(\frac{T}{T_0}\right)^{\frac{3}{2}} \frac{T_0 + C}{T + C} \tag{3}$$

$$\mu_t = \sqrt{\frac{\tau_w}{\rho}} \tag{4}$$

$$\tau_w = \frac{1}{2} C_f \rho U_\infty^2 \tag{5}$$

$$R_{ex} = \frac{\rho U_\infty L}{\mu} \tag{6}$$

To capture velocity, pressure, and density of the flow that vary non-linearly near the surface of the rocket, inflation layers were added to refine the mesh near the walls. The inflation layers will allow the flow to be accurately captured near the wall of the rocket's no-slip condition, and define the turbulent flow and its behavior accurately compared to no inflation layers applied. The height and quality of these inflation layers is defined by its y+ value through a last-ratio boundary condition. The y+ is a dimensionless quantity that characterizes how well the mesh resolves the viscous sublayer near a solid wall, where a lower y+ can lead to a more structured and finer mesh near a wall. From a research study evaluating external flow over airfoil geometry, the difference in y+ detail differing behaviors in flow seperation near the wall and reattachment areas, which is critical in determining proper drag values for the rocket, with or woithout airbrakes [3].

A target y^+ of 2.5 was used for analyzing turbulent flows using the SST $k - \omega$ model, since it approximates solutions for turbulent behavior near the wall. Using equation 1 the height of the first inflation layer can be determined. Equations 2-6 are used for calculating the shear velocity of the flow using known values of pressure, temperature, freestream velocity, length of the rocket, and dynamic viscosity of air. The first layer height, $\Delta s = 2.4064 \times 10^{-5}$ was used for all meshes.

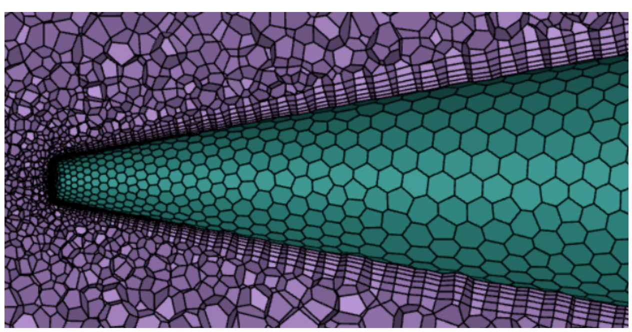


Fig. 4 Inflation Layers Near the Nose Cone

C. Physics Setup

1. Governing Equations

$$I \equiv \frac{u'}{U} \tag{7}$$

$$I = 0.16 R_{ex}^{-\frac{1}{8}} \tag{8}$$

$$k = \frac{3}{2} u'^2 \tag{9}$$

Turbulence intensity is defined as the ratio between the root mean square of the turbulent velocity fluctuations and the Reynolds averaged mean velocity, U. This can be approximated using equation 8, with the Reynolds number. A turbulent intensity and turbulent viscosity ratio boundary condition was used for the farfield in ANSYS Fluent, where the turbulent viscosity ratio can be approximated using the turbulent kinetic energy and specific dissipation ($k - \omega$).

Setting	Current Value	Default Value
Setup		
Models		
Energy	On	Off
Cell Zone Conditions		
Boundary Conditions		
Inlet		
farfield (pressure-far-field, id=33)		
Turbulent Intensity	1 [%]	5 [%]
Z-Component of Flow Direction	-1	0
X-Component of Flow Direction	0	1
Mach Number	0.7000000000000001	0.6
Gauge Pressure	84300 [Pa]	0 [Pa]
Reference values		
Viscosity	1.8461798e-05 [kg/(m s)]	1.7894e-05 [kg/(m s)]
Velocity	242.96409 [m/s]	1 [m/s]
Temperature	300 [K]	288.16 [K]
Pressure	84300 [Pa]	0 [Pa]
Length	2.2765 [m]	1 [m]
Enthalpy	31377.67 [J/kg]	0 [J/kg]
Density	0.97894918 [kg/m^3]	1.225 [kg/m^3]
Area	0.02624302 [m^2]	1 [m^2]

Fig. 5 Farfield Boundary Conditions

IV. Mesh Convergence Study

Computational Fluid Dynamics (CFD) uses the finite element method to achieve an approximate solution to complex partial differential equations that describe fluid flow. This is done using a process called discretization, which breaks down a problem with no analytical solution into smaller discrete subproblems for which an approximation can be determined. Discretization is used by specialized programs, such as ANSYS Fluent, for mesh generations that divide complex geometries into finite elements and connect them by nodes, resulting in a finite element mesh. The accuracy of the approximation is dependent on the resolution of the mesh. A mesh convergence study was conducted to ensure that the mesh resolution is refined enough to result in a solution dependent only on the physics setup.[2] To perform the study, a constant setup was used on the same geometry with increasing mesh resolution until the coefficient of drag monitor changed by less than 1% while reducing the cell counts of the mesh as much as possible.

Target y+	layers	cell count	boi length	fins sizing	tailcone sizing	nose tip sizing	body tube s	BOI sizing	cd	% Relative Error
2.5	15	5601151	1.5	0.001	0.005	0.001	0.01	0.01	0.3018	N/A
2.5	15	7891206	2	0.001	0.005	0.001	0.01	0.0075	0.2876	4.705102717%
2.5	15	8185362	2.5	0.001	0.005	0.001	0.0075	0.0075	0.2664	7.371349096%
2.5	15	9985312	2.5	0.0008	0.005	0.001	0.0075	0.0075	0.2695	1.163663664%
2.5	15	11092964	2.5	0.0008	0.005	0.0008	0.0075	0.0075	0.2683	0.4452690167%
2.5	15	12787665	2.5	0.0007	0.005	0.0008	0.0075	0.0075	0.2678	0.1863585539%

Fig. 6 Mesh Convergence Table

Cell counts for the mesh were increased using finer local sizing particularly on the fins and the nose cone aluminum tip. Subsequent CFD iterations used the same setup from **Figure 5**. The relative error percentage measured the change in the coefficient of drag from the previous iteration.

From **Figures 6 and 7** the local sizing for final two iterations changed the cd by less than 1%, which matches our criteria for an independent mesh. These mesh parameters were used for the rocket and airbrakes analysis.

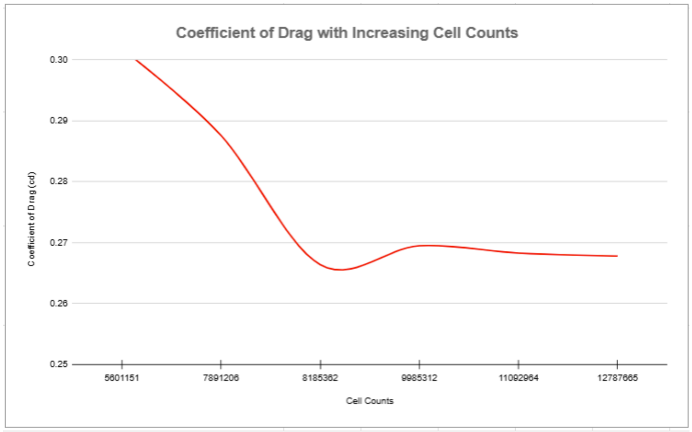


Fig. 7 Coefficient of Drag with Increasing Cell Counts

V. ANSYS Fluent CFD Analysis

A. Governing Equations

$$\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)+f_x$$

$$\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)+f_y$$

$$\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial p}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)+f_z$$

Navier Stokes Equations

Fig. 8 Navier Stokes Equations

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0$$

Fig. 9 Continuity Equation

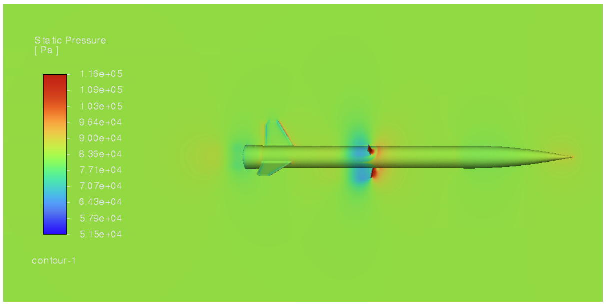
B. Results

The mesh convergence study used the baseline rocket without airbrakes, to established best practices for local face sizing, body-of-influence placement, farfield enclosure dimensions, and y^+ targeting. The resulting data informed the setup used to integrate a simplified airbrake model and capture accurate flow behavior.

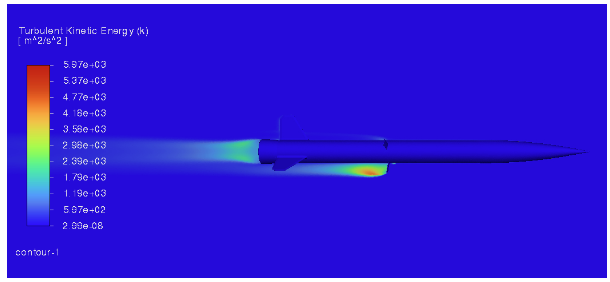
With the previous face sizings located in **Figure 6**, along with an assumed face size of 0.00075 for the airbrake flaps, a Pressure-Based simulation was run with an SST $k-\omega$ model with boundary conditions consistent with an altitude of 1,600 m, which is consistent with our OpenRocket altitude simulations. This simulation was conducted to identify flow behavior on and around the airbrakes as well as determine the force being applied on the flaps themselves to support design decisions for the airbrakes subteam.

Simplified airbrake models were simulated at deployment angles of 80° and 59.9° to compare drag force, fluid interactions, and control authority. The 80° configuration produced the greatest structural loading due to the larger projected area exposed to the flow. In contrast, the 59.9° configuration corresponds to the angle at which maximum motor current draw occurs.

The 59.9° value was determined using a low-fidelity polynomial model relating motor torque to current draw as a function of airbrake deployment angle. Because this torque–current relationship is nonlinear, the peak electrical demand does not occur at full deployment 80°, but instead at 59.9°. These force and torque calculations were required to validate the electrical power and system draw requirements in collaboration with the Embedded Systems team.

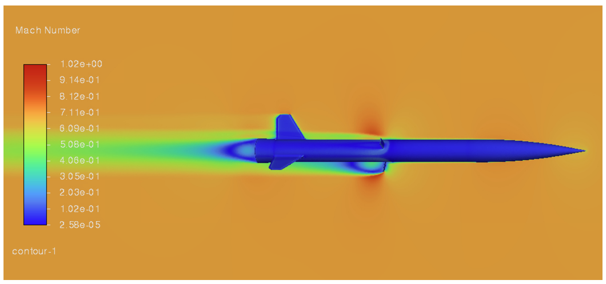


(a) Static Pressure

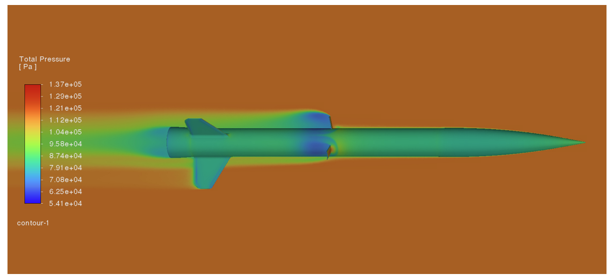


(b) Turbulence Kinetic Energy

Fig. 10 Contours of static pressure and turbulence kinetic energy for the 80° airbrake configuration.

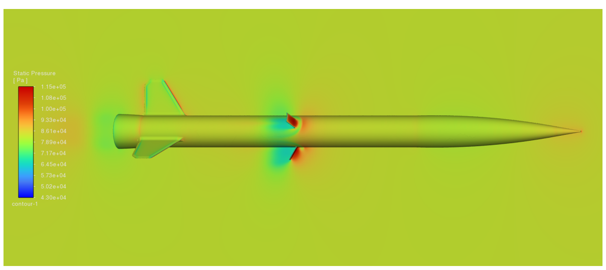


(a) Mach Number

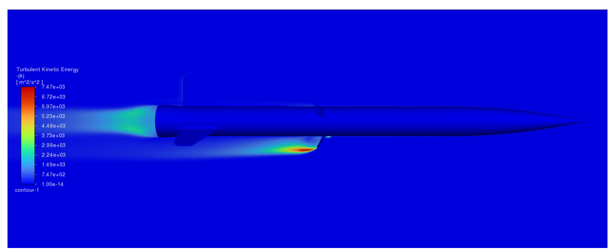


(b) Total Pressure

Fig. 11 Contours of Mach number and total pressure for the 80° airbrake configuration.

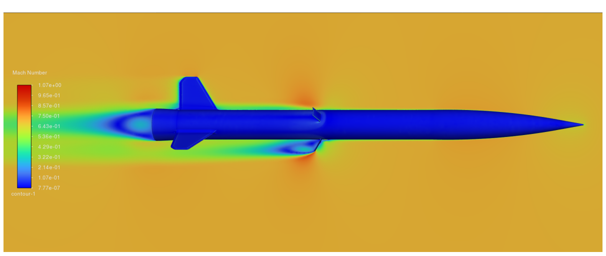


(a) Static Pressure

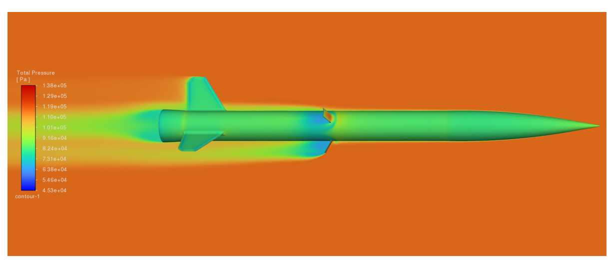


(b) Turbulence Kinetic Energy

Fig. 12 Contours of static pressure and turbulence kinetic energy for the 59.9° airbrake configuration.



(a) Mach Number



(b) Total Pressure

Fig. 13 Contours of Mach number and total pressure for the 59.9° airbrake configuration.

The contours above allowed our team to investigate the simulated flow region surrounding the airbrakes and validate what was analytically discovered through ANSYS. Regions of interest are specifically at the end of the airbrake section as well as the recirculation zone underneath the airbrakes. The prominent issue in the Mach contours of both simulations was an unnatural acceleration of the air well above freestream velocity. Identifying these regions of interest is important for future simulations, as the behavior shown could be caused by improper use of pressure vs. density solvers or by boundary conditions that need further investigation. The mesh convergence study helped identify the contour inconsistencies as a physics-driven root cause because the mesh quality and convergence metrics were satisfied in this simulation.

The force on the airbrake flaps for the 80 degree setup from ANSYS was identified to be 320.09865 N, which breaks down to 106.69955 N per flap, or 23.9870131 pounds-force. To validate the ANSYS values provided, the following hand calculations were conducted:

$$F = \frac{1}{2}\rho u^2 C_d A \tag{10}$$

$$F = \frac{1}{2}(1.04)(223.7)^2(1)(0.00387096)cos(10) \approx 106N$$

The hand calculations above verified our simulated force values on the airbrakes, as the rough values that were computed by hand contributed to a deviated final value, but with the computed values from ANSYS and hand being within 1% of each other, the values from ANSYS are an appropiate analytical solution.

VI. Conclusion

The computational analysis presented in this study, supported by the integration of Menter's Shear-Stress Transport (SST) turbulence model within ANSYS Fluent and provides a framework for validating airbrake performance on a sounding rocket. By synthesizing the SST $k - \omega$ model with a mesh convergence study and parameterized ANSYS Fluent sweeps, several key conclusions regarding airbrake design and integration can be drawn.

For high-precision missions like the International Rocket Engineering Competition (IREC), the ability to accurately model the "shadowing" effect and the interaction between airbrake wakes and aft stabilizing fins is paramount. This analysis confirms that the airbrakes can be utilized as an effective active control surface for apogee targeting. By accurately modeling the pressure and drag on the airbrakes, the University of Georgia Rocketry Team can design the airbrake assembly and feedback control system with greater confidence.

The University of Georgia Rocketry Team will conduct a test flight in the near future, during which flight telemetry will be recorded. These data will be used to compare and validate current and future high-fidelity simulations.

Ultimately, this research bridges the gap between theoretical turbulence modeling and practical aerospace engineering. The move toward an SST-centered CFD workflow allows a more aggressive airbrake design, optimizing for weight and deployment speed, while maintaining a high degree of confidence in the vehicle's structural integrity and trajectory predictability. Future work will focus on transient simulations to capture the dynamic loads during the deployment phase, further refining the safety margins for the 10,000 ft COTS entry.

References

[1] Florian R. Menter, “Improved Two Equation $k - \omega$ Turbulence Models For Aerodynamic Flows”, *NASA Technical Reports*, 1992.

[2] “The Importance of Mesh Convergence”, *NAFEMS*, 2026, https://www.nafems.org/publications/knowledge-base/the-importance-of-mesh-convergence-part-1/?srsltid=AfmBOopIfjf0kUBN6s11MElzlFITrZBpXKPEzWTryROdCVfqhkInw4-U.%20Accessed%2017%20Feb.%202026.

[3] Durmus Sinan Korpe, Ozdemir Ozturk Kanat, and Tugrul Oktay, “he Effects of Initial y plus: Numerical Analysis of 3D NACA 4412 Wing Using $\gamma R_{e\theta}$ SST Turbulence Model”, 2019