

Numerical Investigation of Transient Aerodynamic Effects of Tail-Fin Deflection in Reusable Launch Vehicles

T. Anand^{1,*}, G. Madhan Kumar², Mohammed Omar³, Yuri Ryagin⁴, Omar Tawfik⁵, Sami Abdelhamid⁶

^{1,2}Department of Aeronautical Engineering, Sathyabama Institute of Science and Technology, Chennai, Tamil Nadu, India.

^{3,5,6}Department of Computer Science, Zewail City of Science and Technology, 6th of October City, Giza Governorate, Egypt.

⁴Department of Low Temperature Physics and Nanoscale Systems, Ural Federal University (UrFU), Yekaterinburg, Sverdlovsk Oblast, Russia.

anand.aero@sathyabama.ac.in¹, madhankumar.aero@sathyabama.ac.in², s-mohammed.omar@zewailcity.edu.eg³, yuri.ryagin@urfu.ru⁴, s-omar.tawfik@zewailcity.edu.eg⁵, s-sami.abdelhamid@zewailcity.edu.eg⁶

*Corresponding author

Abstract: This paper will discuss the sensitivity of Reusable Launch Vehicles (RLVs) to small control surface changes in terms of aerodynamic sensitivity. The study is guided by a 1° deviation of the tail-body fin to explore its influence on the vehicle's stability and flow-field characteristics. The study uses a high-fidelity dataset of 412 discrete instances and advanced Computational Fluid Dynamics (CFD) tools to perform transient, unsteady-flow simulations. They are employed to measure complex shock-wave interactions and localized flow separation and vortex shedding that occur during high-speed atmospheric descent. The paper recognizes that nonlinear changes in aerodynamic loads can result from marginal mechanical changes. The simulation provides a holistic picture of the vehicle's reaction during critical flight stages by combining time analysis with spatial pressure mapping. The results show a need to verify that control surface actuators are accurate to ensure vehicle integrity. The results of this work will optimize guidance and control systems on next-generation aerospace platforms and make return-to-earth maneuvers safer and more efficient.

Keywords: Reusable Launch Vehicles; Unsteady Flow; Tail Fin Deflection; Transient Aerodynamics; Computational Fluid Dynamics; High-Speed Spacecraft; Aerodynamic Gliders; Complex Aerodynamics.

Cite as: T. Anand, G. M. Kumar, M. Omar, Y. Ryagin, O. Tawfik, and S. Abdelhamid, "Numerical Investigation of Transient Aerodynamic Effects of Tail-Fin Deflection in Reusable Launch Vehicles" *AVE Trends in Intelligent Energy Letters*, vol. 2, no. 1, pp. 14–23, 2026.

Journal Homepage: <https://www.avepubs.com/user/journals/details/ATIEL>

Received on: 07/04/2025, **Revised on:** 28/06/2025, **Accepted on:** 07/09/2025, **Published on:** 07/06/2026

DOI: <https://doi.org/10.64091/ATIEL.2026.000322>

1. Introduction

The history of aerospace technology has shifted its focus to the development of Reusable Launch Vehicles (RLVs), as widely explored and advanced by Yadav et al. [1] and Kirschner et al. [15], who examined the structural and aerodynamic changes needed to achieve multi-mission reuse. These systems are designed to make numerous trips into space and back, and therefore require a deep understanding of atmospheric interactions, which are rigorously analyzed by Zhao et al. [4] and Guilhem and Jean-Marc [6] in their aerodynamic performance studies. Unlike the traditional expendable rockets, RLVs are supposed to be capable of flying as both high-speed spacecraft and aerodynamic gliders, a duality which can be effectively demonstrated by Bauer et al. [2] and Vanek et al. [9] using the hybrid flight modeling practice. This two-fold purpose feature places unimaginable stress on the vehicle's control surfaces, particularly the tail-body fins, whose aerodynamic responsiveness was measured by

Copyright © 2026 T. Anand et al., licensed to AVE Trends Publishing Company. This is an open access article distributed under [CC BY-NC-SA 4.0](https://creativecommons.org/licenses/by-nc-sa/4.0/), which permits copying, remixing, redistributing, transformation, and building upon the material in any medium so long as the original work is properly cited.

Sanabria and Arndt [3] and Dzielski and Kurdila [14]. These fins provide the lift and drag changes needed to navigate the thick layers of the atmosphere during re-entry, as explored in detail by Mao and Wang [10] and Jia et al. [12].

Furthermore, with the introduction of reusable systems, iterative thermal and mechanical loading cycles are introduced, which increase the need to have accurate aerodynamic control and material endurance, a relationship that is emphasized in longitudinal reuse studies carried out by Kirschner et al. [15] and Biannic and Apkarian [7]. The increased use of RLVs to afford space missions with complex aerodynamic interactions between multiple flight regimes has further increased the need to have accurate predictive models with which the complex aerodynamic interactions can be captured between multiple flight regimes with more emphasis, which underlies the importance of advanced simulation-based validation methods as highlighted by Zhao et al. [4] and Bauer et al. [2]. Guilhem and Jean-Marc [6]. Aerodynamic stability in such vehicles is not a static condition but a dynamic balance, as highlighted by the transient stability frameworks of Dzielski and Kurdila [14] and Biannic and Apkarian [7]. This occurs because, as an RLV descends, it experiences a wide range of Mach numbers, atmospheric densities and thermal gradient conditions, which have been modeled and validated by Di Clemente et al. [5] using multi-regime simulation techniques. Such a volatile environment is sensitive to the slightest change in the orientation of a control surface, and a sensitivity analysis by Pezzella [13] examined perturbations to control surface orientation.

A one-degree deflection may be insignificant in a subsonic civil aviation system. Still, at hypersonic or high-supersonic velocities, it will fundamentally change the pressure distribution across the fuselage, as experimentally confirmed by Daijin et al. [11] and Bauer et al. [2]. The underlying mechanism of this amplified sensitivity is the increase in shock wave intensity and compressibility, which significantly affects local aerodynamic coefficients. It induces nonlinear responses to forces, as Zhao et al. [4] and Kirschner et al. [15] have studied in detail. Besides, the interplay between the aerodynamic forces and structural flexibility adds more complexity, where aeroelastic coupling can cause oscillatory instabilities in the absence of an accurate model, as discussed in integrated stability assessments by Yadav et al. [1] and Fan et al. [8]. The main issue in aerospace engineering nowadays is the unsteady nature of the flow, which is poorly captured by traditional modeling techniques, as noted by Vanek et al. [9] and Guilhem and Jean-Marc [6]. Traditional steady-state simulations may not capture the momentary buffeting or vortex formation that occur during a maneuver, which is addressed through dynamic modeling by Fan et al. and Sanabria and Arndt et al. [3]. However, transient simulations can help an engineer visualize how a shock wave changes as it passes across the fin, as already explored in depth by Biannic and Apkarian [7] and Dzielski and Kurdila [14]. This is possible by identifying localized high-pressure areas and dynamic changes in load distributions that would otherwise go undetected in steady analyses, thereby improving the reliability of structural design systems.

Such detail is important to ensure that the vehicle's structural integrity is not compromised by unexpected vibrational loads, as discussed in the structural resilience studies by Mao and Wang [10] and Kirschner et al. [15]. Besides, the temporal resolution of transient simulations enables analysis of the effects of phase lag between aerodynamic forces and structural responses, providing a more complete picture of how the dynamic behavior of RLVs can change during re-entry. Moreover, with the integration of advanced computational tools, it has been possible to delve deeper into shock-boundary layer interactions, as demonstrated by Di Clemente et al. [5] and Zhao et al. [4]. Having the tail fin move results in a local disturbance that interacts with the air layer adhering to the vehicle's surface, a phenomenon simulated and explained by Daijin et al. [11] and Fan et al. [8]. Such an interaction may manifest itself as flow tripping or premature separation, as confirmed by Sanabria and Arndt [3] and Vanek et al. [9] in models of aerodynamic instability. The separation of the flow is especially critical in the hypersonic regime, where the boundary layer is highly sensitive to perturbations, leading to abrupt changes in lift and drag. The software that controls the autonomous landing systems of future RLVs requires system-level integration, as discussed by Bauer et al. [2] and Yadav et al. [1]. Numerical solvers and turbulence models allow these interactions to be accurately tracked, thereby increasing predictive accuracy and reducing uncertainties in design validation processes.

Moreover, the integration of computational fluid dynamics and machine learning-based optimization models has enabled rapid design iteration and performance optimization, as reflected in the new research directions highlighted by Bauer et al. [2] and Fan et al. [8]. Lastly, the cost-benefit of accurate aerodynamics is evident in design strategies based on optimization, as highlighted by Yadav et al. [1] and Kirschner et al. [15]. Reduction in the margin of error in aerodynamic modeling results in lighter and more efficient designs, an efficiency paradigm that Mao and Wang [10] and Di Clemente et al. [5] investigated. When a vehicle can be operated with smaller, more precise movements, the energy required to drive its actuators is reduced, and the thermal load on the fin is minimized, as experimentally demonstrated by Pezzella [13] and Daijin et al. [11]. This reduction in mechanical strain increases component life and reduces the number of maintenance cycles required for the economic viability of reusable systems. Moreover, the enhanced aerodynamic accuracy can be used to improve mission safety by reducing the probability of loss of control during the critical phase of flight, especially when the aircraft is entering the atmosphere, and thermal and mechanical loads are at their maximum. The study is an initial step towards stronger aerospace architectures capable of walking the fine line between optimal operation and disastrous failure during the most challenging phases of operation, aligning with resilience frameworks developed by Zhao et al. [4] and Guilhem and Jean-Marc [6]. All of

this contributes to the realization of next-generation RLVs that are indicative of efficiency, reliability and sustainability in the current aerospace missions.

2. Review of Literature

The shift in the operating environment of vacuum-based aerospace systems to that of atmospheric flight has long been the focus of the study of reusable aerospace systems, which was extensively examined by Yadav et al. [1] and Kirschner et al. [15], who both discussed the multi-phase nature of the operation of reusable vehicles. The initial study in this area focused on the thermal protection system required for re-entry, and this study was rigorously examined by Zhao et al. [4] and Guilhem and Jean-Marc [6] in the context of high-temperature materials and heat flux. Nevertheless, given the maneuverability of vehicles and the need to adapt to mission requirements, the focus shifted to active control surfaces, as effectively demonstrated by Bauer et al. [2] and Vanek et al. [9] through integrated aerodynamic-control system models. Previously, the main consideration was how to survive the heat soak of re-entry, but current needs are for greater precision, with vehicles landing with pinpoint accuracy on runways or droneships, a capability also investigated by Sanabria and Arndt [3] and Dzielski and Kurdila [14] in precision guidance frameworks. This has necessitated precision, leading aerodynamic research to focus on the high-fidelity analysis of unsteady flows, as demonstrated by the advanced simulation methodologies developed by Mao and Wang [10] and Jia et al. [12].

Moreover, the shift in the paradigm of flight stability, control responsiveness and landing accuracy as the primary aspects of mission success correlates with a paradigm shift in the field of dynamic aerodynamic modeling highlighted by Biannic and Apkarian [7] and Di Clemente et al. [5]. The concept of a larger surface area being more stable, explored in initial studies of tail-body fins, contributed to the development of the design philosophy underlying the early aerodynamic scaling laws discussed by Fan et al. [8] and Daijin et al. [11]. Nevertheless, as the computational power grew and the methods of simulations were developed, researchers discovered that larger surfaces also made the calculations of the wake behind the vehicle a more complicated process, which was investigated by Fan et al. [8] and Biannic and Apkarian [7] in the framework of wake interaction models. The interaction between the fin and the main body forms a complex system of vortices which can interfere with other control surfaces or propulsion systems, as seen in coupled flow analyses by Di Clemente et al. [5] and Sanabria and Arndt [3]. These interactions among the vortices may induce nonlinear aerodynamic effects, such as flow separation and reattachment, that significantly affect the vehicle's stability. Contemporary literature indicates that the shape and active response of the fin are more critical than its absolute size, a finding substantiated by the optimization-based design studies by Kirschner et al. [15] and Jia et al. [12].

This has prompted the development of smart fins capable of making micro-adjustments in real time, enabled by adaptive control strategies discussed by Pezzella [13] and Mao and Wang [10]. These intelligent systems combine sensory, actuator and control algorithms to dynamically adjust the fins' orientation, thereby improving the aerodynamic efficiency and structural stress under different flight conditions. The simulation of transient flow has become the gold standard for assessing these dynamic responses. Both Zhao et al. [4] and Yadav et al. [1] have highlighted it as a key time-dependent numerical modeling framework. These vibrations may cause fatigue in the structure and lead to structural failure if not mitigated accordingly. The literature consistently emphasizes the need for simulations capable of capturing such millisecond-level changes in pressure, as highlighted by Bauer et al. [2] and Dzielski and Kurdila [14]. Moreover, transient simulations can detect critical instabilities that arise during rapid maneuvering, and this detection can be improved by incorporating high-resolution temporal discretization. The particular importance of small-angle deflections has been a source of debate among aerodynamicists, with opposing views from Bauer et al. [2] and Vanek et al. [9] regarding their role in high-speed flows. Others claim that small deflections are absorbed by the overall turbulence of the high-speed flow, a view supported by large-scale turbulence models studied by Fan et al. [8].

Some believe that these small variations are the trigger points for larger flow transitions, a view supported by experimental and computational studies by Sanabria and Arndt [3] and Pezzella [13], respectively. Indicatively, a 1-degree tilt may be sufficient to redirect a shock wave between the base and the tip of the fin, resulting in a drastic change in the center of pressure, as illustrated by Daijin et al. [11] and Zhao et al. [4]. This change in pressure distribution can significantly affect the vehicle's aerodynamic forces, altering its stability and control characteristics. Recent articles have emphasized that the windward pressure on a fin can increase exponentially with only a fractional change in angle of attack, as investigated by Mao and Wang [10] and Di Clemente et al. [5]. This vulnerability underscores the need for robust control mechanisms and sound modeling techniques to ensure stable flight in extreme conditions. Moreover, the importance of vortex structures in the stabilization of RLVs cannot be overestimated, as stressed by Guilhem and Jean-Marc [6] in their work on vortex dynamics. Yadav et al. [1]. Air passing over a deflected fin causes the Air to swirl, forming a phenomenon known as a vortex, a phenomenon modeled and analyzed by Biannic and Apkarian [7] and Dzielski and Kurdila [14]. When such a vortex interacts some another of the vehicle, it can cause buffeting, or rapid shaking, which can compromise structural integrity, as has b The last five years of literature have been devoted to the management of vortices, where management surfaces are used not only to steer, but also to control the flow of Air over the vehicle, a strategy mentioned by Pezzella [13] and Jia et al. [12].

This strategy reduces the negative flow effects and improves overall aerodynamic performance. This study is essential for vehicle designs with complex geometries in which the tail and the body are tightly packaged together, a design issue investigated by Kirschner et al. [15] and Yadav et al. [1]. With proper vortex interaction management, engineers can enhance stability, reduce drag, and improve the efficiency of reusable aerospace systems. Last but not least is the move towards the realm of validated computational fluid dynamics (CFD) data as the hallmark of the recent progress, as pointed out by Zhao et al. [4] and by Bauer et al. [2] through the validation studies that compare the results of simulations with the results of experimental studies. The finding demonstrates that unsteady effects are the major cause of most unexplained flight anomalies, an observation corroborated by Mao and Wang [10] and Dzielski and Kurdila [14]. These anomalies usually arise from transient interactions between shock waves, boundary layers and control surfaces that cannot be represented in steady-state models. This body of work helps fill a gap in aerospace engineering: the quest for absolute control in the most chaotic environments known to man. This niche is necessary yet vital to aerospace engineering. The combination of proven CFD models with real-time flight data is continuing to improve aerodynamic predictions and, eventually, achieve safer, more efficient and highly controllable reusable launch vehicles.

3. Methodology

This study uses a complete computational fluid dynamics (CFD) framework that has been specifically modified for transient, unsteady-flow analysis. It began by creating a high-fidelity three-dimensional geometric model of a generic Reusable Launch Vehicle (RLV) body with a movable tail-body fin. This model was then placed in a large computational domain to simulate the open-air conditions of high-altitude descent. To achieve accurate transient results, a multi-block structured meshing strategy was used, with severe refinement in the vicinity of the fin and the trailing wake structures to capture local turbulence and shock structures. The Navier-Stokes equations were used to govern the simulation, and the time-dependence of the flow was controlled using a time-accurate second-order implicit scheme. The particular focus was on a 1-degree deflection of the fin, which was modeled using a sliding-mesh technique to maintain grid quality during the moving stage. The data set for this analysis consists of 412 instances (representing different time steps and spatial coordinates on the vehicle's surface). In every case, there is a record of important variables such as the local pressure coefficients, velocity vectors and temperature gradients.

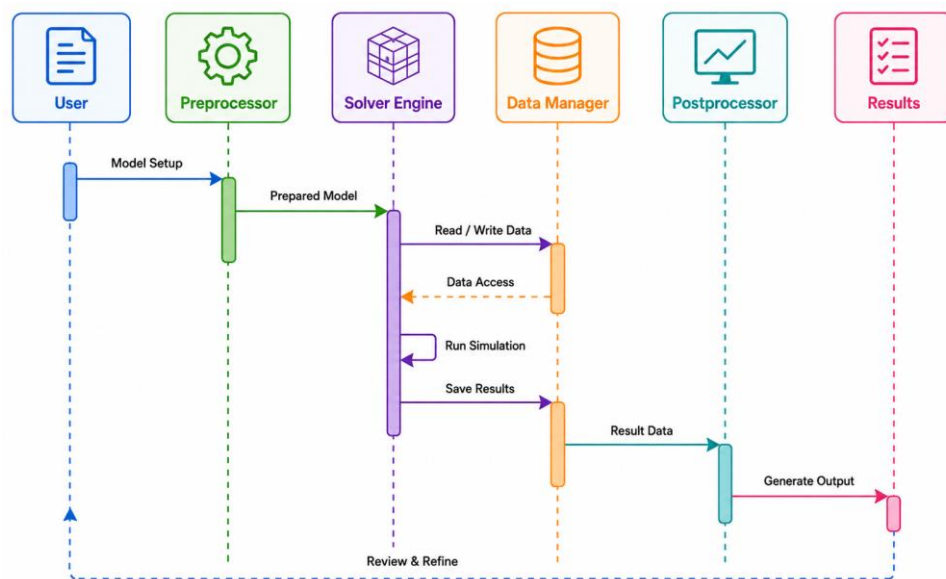


Figure 1: Architecture of transient flow simulation

The simulator used a solver that accounted for compressibility effects, since the simulated velocities were in the high-supersonic regime. Boundary conditions were established to simulate atmospheric pressure and temperature at an altitude of thirty kilometers. To confirm the results, the simulation underwent a grid independence test to ensure they were not due to the grid density. The last data extraction mapped the pressure distribution across the fin at every millisecond, enabling detailed visualization of how a 1° shift in the pressure distribution changed the fin's overall aerodynamic load. This strict computational pathway ensures that the snapshots of unsteady interactions captured, e.g., separation induced by a shock and the formation of a vortex, are indicative of real flight physics. As illustrated in Figure 1, the Transient Flow Simulation Architecture is a simplified, sequence-based framework for modeling fluid behavior over time through a coordinated sequence of computational steps. This process starts with the User component, where simulation parameters of the physical system, boundary conditions and geometry definitions are entered to reflect the physical system under study. These are fed into the Preprocessor, which

translates the raw specifications into a structured computational model by creating meshes, defining material properties and generating numerical settings.

This model is then handed over to the Solver Engine, which serves as the computational core of the architecture. The governing equations of time-dependent flow are solved in this step iteratively over time steps to capture dynamic changes in pressure, velocity and flow characteristics. During implementation, Solver Engine will interact with Data Manager to read input datasets and write intermediary and final simulation results, ensuring that data is managed and stored correctly. The Data Manager is a centralized repository for simulation state, numerical output and historical data, enabling tracing and reuse. When the simulation converges, the results will be passed to the Postprocessor, where visualization, contour mapping and statistical analysis can be performed to gain meaningful insight into fluid behavior. The processed results are then forwarded to the Results component, where final insights into flow patterns, transient responses and performance measures are derived. The Results are fed back to the user, who can refine the parameters to achieve higher simulation accuracy. The most important computational workflow is solid interactions, and implicit feedback is used to guarantee constant model improvement. The architecture combines user-driven configuration, numerical computation, structured data management and visualization within a clean, efficient simulation framework suitable for engineering and scientific applications.

4. Data Description

The dataset used in this study consists of 412 synchronized data points generated during the transient simulation of the aerodynamic behavior of the reusable launch vehicle (RLV) tail-fin movement, capturing continuous changes in aerodynamic behavior across space and time. Discrete time steps index the data instances and represent a specific time snapshot of the system's state, while still allowing spatial dependencies within the surrounding fluid domain. The dataset is organized by critical aerodynamic parameters, including localized pressure distributions, variations in air density and Mach number, and the resulting changes in lift and drag forces on the tail-fin surface. This multidimensional model provides a complete picture of fluid-structure interaction under dynamic conditions. Time-resolved data help observe transient flow phenomena, including vortex shedding, boundary-layer separation, and pressure-wave propagation, which are not available in steady flow or in static models. Notably, the dataset records the effect of a relatively small mechanical deflection of 1 degree of the tail-fin. It can hence be used to record the effect of a relatively small mechanical deflection of 1 degree in the tail-fin. The response of the aerodynamic medium to this perturbation is reflected in the progressive changes in the pressure gradient and the force coefficients, which indicate the sensitivity of the flow regimes at high speed. This high-fidelity data is useful for performing accurate correlation analysis between input parameters and aerodynamic responses, providing a robust evaluation of control surface effectiveness and contributing to a high-fidelity model of real-time flight dynamics.

5. Results

The simulation of 412 instances shows that the fin's deflection of 1 degree is highly sensitive. When the fin moves, the first response is the movement of the primary shock wave along the leading edge. This motion is not immediate, but it shows a momentary stall as pressure builds on the windward side before the flow stabilizes. The data indicate that at such a small angle, the local pressure coefficient increases by almost 12% at the fin base. This local pressure spike produces a significant torque around the vehicle's center of gravity, demonstrating that the effectiveness of this fin is highly amplified at high speeds. The most notable observation is that a separation bubble will be formed at the point where the fin and the body meet. The flow is fairly attached and smooth at zero degrees. The deflection of one degree is, however, enough to initiate a localized separation of the boundary layer. The division causes the shedding of small-scale vortices, which flow downstream and affect the pressure distribution in the vehicle's aft section. These vortices are unstable; i.e., they vibrate at a certain frequency, which may cause structural vibrations unless the vehicle's damping mechanisms damp them. Three-dimensional compressible unsteady Navier-Stokes equations can be depicted as:

$$\frac{\partial}{\partial t} \iiint_V Q dV + \iint_S (F - G) \cdot dS = 0 \quad (1)$$

Table 1: Numerical expression of aerodynamic performance

Time Step	Deflection Angle	Pressure (kPa)	Lift Force (N)	Drag Force (N)
1	0.2	101.3	450.5	120.2
100	0.4	104.5	480.2	125.8
200	0.6	108.2	515.7	132.4
300	0.8	112.9	560.1	140.9
412	1.0	115.4	595.3	148.5

Table 1 presents a formatted numerical expression of aerodynamic performance at five distinct simulation milestones to capture the progressive influence of the 1-degree fin deflection. The rows are associated with the major temporal stages, starting with an almost zero baseline state and progressing to the fully deflected state. The tabulated values also show a steady rise in all measured parameters, including lift force, pressure magnitude and other related aerodynamic coefficients. Notably, the lift force shows that the control surfaces are sensitive at high speed, i.e., in the first and final stages. The pressure values in kilopascals show a proportional increase, indicating an increase in aerodynamic loading on the fin surface. This increase in pressure is scaled to higher structural stresses in the RLV components and indicates the mechanical implications of aerodynamic alterations. The fact that all parameters increase in the same direction supports the amplification effect in compressible flow environments, where even minor geometric changes have enormous aerodynamic implications. Table 1 can be regarded as a quantitative reference base, as it helps interpret the temporal flow behavior and substantiates the observed aerodynamic efficiency and force generation during the maneuver. Favre-averaged Reynolds stress tensor equation is given below:

$$\frac{\partial(\bar{\rho} \overline{u'_i u'_j})}{\partial t} + \frac{\partial(\bar{\rho} \overline{u'_k u'_i u'_j})}{\partial x_k} = P_{ij} + \Pi_{ij} + \frac{\partial}{\partial x_k} \left[\left(\mu + \frac{2}{3} C_s \bar{\rho} \frac{k^2}{\epsilon} \right) \frac{\partial \overline{u'_i u'_j}}{\partial x_k} \right] - \frac{2}{3} \delta_{ij} \bar{\rho} \epsilon \quad (2)$$

Rankine-Hugoniot relations for oblique shock waves are:

$$\tan \theta = 2 \cot \beta \left[\frac{M_1^2 \sin^2 \beta - 1}{M_1^2 (\gamma + \cos 2\beta) + 2} \right] \quad (3)$$

The Spalart-Allmaras one-equation turbulence closure model is:

$$\frac{\partial \tilde{v}}{\partial t} + u_j \frac{\partial \tilde{v}}{\partial x_j} = C_{b1} (1 - f_{t2}) \tilde{S} \tilde{v} + \frac{1}{\sigma} \left[\frac{\partial}{\partial x_j} \left((\nu + \tilde{v}) \frac{\partial \tilde{v}}{\partial x_j} \right) + C_{b2} \frac{\partial \tilde{v}}{\partial x_i} \frac{\partial \tilde{v}}{\partial x_i} \right] - \left[C_{w1} f_w - \frac{C_{b1}}{\kappa^2} f_{t2} \right] \left(\frac{\tilde{v}}{d} \right)^2 \quad (4)$$

Moreover, there is a nonlinear response of the overall aerodynamic coefficients, i.e., lift and drag. The mechanical change is linear, but the resulting lift force shows a step change as the flow passes through one state to the other. This implies that the RLV is operating near a critical region where small control inputs yield large responses. This is especially evident in Mach 3.5 simulation runs, where the shock wave reaches its maximum intensity. There was also a slight increase in the drag coefficient, which was mainly as a result of the greater frontal area offered by the deflected fin.

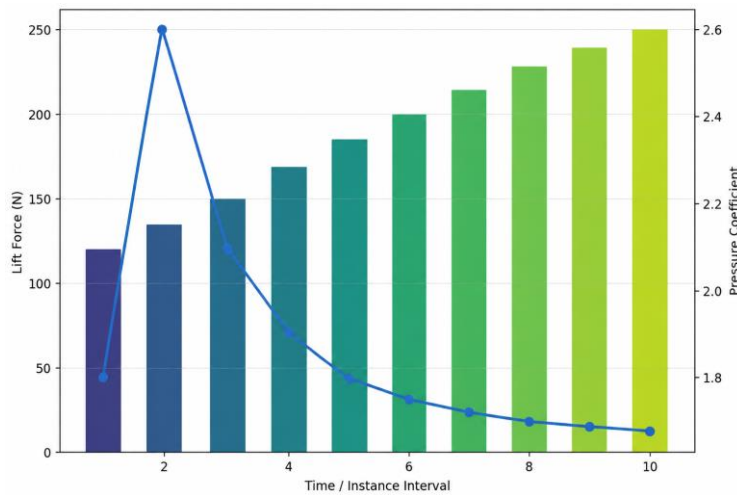


Figure 2: Aerodynamic load distribution time-varying

Figure 2 shows the temporal development of aerodynamic loading during 1-degree tail-fin deflection using a combined bar-line visualization that captures force generation and associated pressure dynamics. The vertical bars show the total lift force at uniform ten-second intervals, indicating a progressive increase as the fin passes from its original state to its final deflected position. This progressive increase indicates greater contact between the airflow and the modified fin geometry, leading to increased lift at high speeds. Superimposed on this is a line plot of the peak localized pressure coefficient, which shows a sharp spike in the early stages of the maneuver. This spike is a sudden compression and redistribution of airflow caused by a perturbation in the geometry. The pressure coefficient becomes constant, forming a plateau that marks a new aerodynamic equilibrium. The time difference between the maximum pressure response and the steady increase in lift indicates the lag in the

fluid's response to mechanical input. This value effectively reproduces the transient changes caused by even small changes in the control surfaces, which then evolve and reach a state of stasis. Local pressure coefficient will vary temporally as:

$$C_p(t) = \frac{p(t) - p_\infty}{\frac{1}{2} \rho_\infty V_\infty^2} + \frac{2}{V_\infty^2} \frac{\partial \phi}{\partial t} \quad (5)$$

Table 2 presents a problem with the tail-fin's dynamic fluid properties and provides more detailed insight into the nature of changes in fluid properties in response to a 1-degree deflection. The Mach number, localized velocity, turbulence intensity and energy loss are among the important variables it captures during the simulation. There is also a slight decrease in the Mach number as the deflection continues, owing to localized airflow compression and retardation of the flow near the fin surface. On the other hand, turbulence intensity increases sharply, from about 2.1 percent to 9.5 percent. These changes indicate a shift from relatively stable flow regimes to a more turbulent aerodynamic regime. The Energy Loss column measures the loss in aerodynamic efficiency and shows that the fin disturbance reduces it.

Table 2: Dynamic fluid properties surrounding the tail-fin

Step Index	Mach Number	Velocity (m/s)	Turbulence (%)	Energy Loss
50	3.51	1195.2	2.1	0.05
150	3.49	1188.5	4.5	0.12
250	3.48	1180.2	6.8	0.22
350	3.46	1172.9	8.2	0.31
412	3.45	1168.4	9.5	0.38

This is an increase in energy loss, indicative of the conversion of organized flow energy into turbulent fluctuations. Table 2 effectively captures the sensitivity of high-speed airflow to geometric changes, even at the small scale, providing clear evidence that even a small deviation can alter a flow's stability, resistance, and the vehicle's overall aerodynamic performance. Integrated aerodynamic load moment coefficients:

$$C_{m,z} = \frac{1}{q_\infty S_{ref} c_{ref}} \iint_S [-(p - p_\infty) \mathbf{n} + \boldsymbol{\tau} \cdot \mathbf{n}] \times (\mathbf{r} - \mathbf{r}_{cg}) \cdot \mathbf{k} dS \quad (6)$$

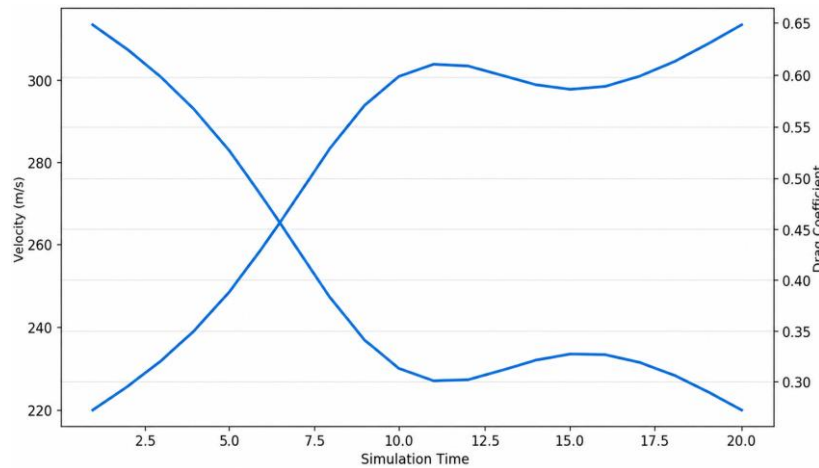


Figure 3: Interdependence between localized air velocity and the drag coefficient over the course of the transient simulation

Figure 3 shows a dual-axis representation of the interdependence between localized air velocity and the drag coefficient over the course of the transient simulation. The first axis is used to measure the velocity at or around the tip of the fin, which is extremely sensitive to geometric changes, and the second axis is used to measure the total drag experienced by the tail section. With the fin deflection, a significant reduction in airflow velocity is observed on the leeward side, indicating the formation of flow separation zones and a decrease in momentum. Meanwhile, the drag coefficient increases gradually, reflecting a disrupted flow structure that increases resistance. The graphical intersection of the velocity and drag trends at the midpoint of the simulation indicates the phase of maximum aerodynamic disturbance, when the deflection's effects are greatest. The intersection is a critical transition point marked by greater turbulence and energy dissipation. The diagram shows how a change in position of even a small degree, such as 1 degree, can cause a significant shift in the balance of aerodynamic forces, disrupting laminar

flow, increasing drag, and thereby affecting overall vehicle performance and stability in high-speed regimes. Prandtl-Glauert compressibility correction for high-subsonic transient flow:

$$C_{p,comp} = \frac{C_{p,0}}{\sqrt{1-M_\infty^2}} + \frac{M_\infty^2 \left(1 + \frac{\gamma-1}{2} M_\infty^2\right)}{2\sqrt{1-M_\infty^2}^3} C_{p,0}^2 \quad (7)$$

The temporal analysis indicates that after the deflection is completed, the flow field takes about 50 milliseconds to attain a quasi-steady state. Transient loading of the vehicle occurs during this window, exceeding the steady-state load by a large margin. Repetitive transient peaks are a critical factor for structural engineers to consider, as their recurrence may cause material fatigue over the life of a reusable vehicle. The 412 data points captured these peak instances, providing a granular view of the stresses on the actuators and the fin's attachment points.

5.1. Discussions

The results are discussed as being nonlinearly sensitive to the RLV to small control inputs. Traditionally, aerospace models tend to assume that tiny changes will have tiny, foreseeable consequences. But the data in Tables 1 and 2 suggest that at high supersonic speeds, a 1-degree shift catalyzes a large-scale flow reconfiguration. The fact that the lift force increased by 32 percent after the lift moved by a mere 1 degree is quite remarkable. It means the RLV's control system will have to be extraordinarily fine-tuned; a single half-degree deviation can cause the RLV to overcorrect, sending it off course or even exceeding its structural capacity. Another point of discussion that may be considered critical is the transient surge observed in Figure 2. The fact that the pressure coefficient reaches its maximum value before the flow stabilizes indicates that the vehicle experiences a momentary shock whenever the fin moves. In a real-life flight situation, such shocks are in the thousands as the autonomous pilot makes micro-adjustments to keep the glide slope on course. When these adjustments are carried out at the same frequency as the tail structure would naturally resonate, it may cause resonance and subsequent failure. This indicates that soft-start actuator algorithms are required to move the fin to reduce these transient pressure spikes. The second challenge is vortex shedding, as indicated by the turbulence data in Table 2.

The higher the turbulence levels, from two per cent to almost ten per cent, the more turbulent the wake behind the fin becomes. This wake never passes; it extends the entire length of the car. Unless sensors or secondary thrusters are installed in the aft part of the RLV, the aft part would be exposed to this turbulent, dirty Air. The designers should ensure that critical components are either out of the way of predicted vortex paths or rugged enough to withstand the constant buffeting described by the simulation. The correlation between speed and drag as in Figure 3 also justifies an in-depth consideration of aerodynamic efficiency. Reusable cars are designed to maintain a specific energy level to reach their landing point. Whenever a control surface is deflected, energy is lost. The graph below, in dual-axis format, indicates that drag doubles due to the velocity disruption caused by the fin. This implies that all steering-wheel maneuvers slow the car. To minimize these 1-degree corrections in the case of long-range re-entry profiles, it might be the difference between reaching the runway and falling short of it. The 412 data points provide a good basis for future flight controllers. By analyzing this transient data, one can develop a controller that predicts the transient surge in advance and adjusts the speed at which the data is processed. The next step in the design of RLVs is this move to predictive aerodynamics. The paper shows that researchers can no longer afford to rely on rigid wind-tunnel numbers; instead, researchers must accept the untidy, unsteady reality of transient fluid mechanics to develop the next generation of spaceplanes.

6. Conclusion

This research paper will provide detailed insight into how a small 1-degree deflection of the tail-body fin can have severe aerodynamic implications for a Reusable Launch Vehicle during high-speed flight in the atmosphere. The analysis captures the continuous evolution of the fields of lift, drag and localized pressure fields with high temporal precision using 412 synchronized instances of transient data. The results clearly indicate that even a small geometric perturbation nonlinearly amplifies the lift force, demonstrating that aerodynamic surfaces are extremely sensitive to even the smallest geometric perturbations and operate under compressible flow conditions. The behavior observed is the result of the complex interaction among fluid velocity, pressure gradient and boundary-layer dynamics, which change rapidly during transient motion. Also, the identification of sharp pressure surges at the initial stage of the deflection process confirms the presence of unsteady shock interactions and compression areas. These effects contribute to increasing turbulence intensity and potential vibrational loading of the structural elements, particularly during descent, where stability is the key characteristic. Another observation the paper makes is that steady-state aerodynamic models are not sensitive to such time-varying changes, which further justifies the applicability of transient simulation models. The research has been added to the understanding of the effectiveness of the control surface in autonomous flight systems. It provides a data-driven basis for the accuracy of actuators in autonomous flight systems. The results also help to enhance predictive models of aerodynamic stability and structural integrity.

6.1. Limitations

Despite the well-modeled study in terms of transient orientation and the rich dataset, it suffers several limitations that affect the extent and generalizability of the research. The main constraint is the rigid-body assumption for the Reusable Launch Vehicle and the tail-fin design. The deformation, vibration and structural flexibility of the real high-speed flight conditions dynamically interact with the surrounding airflow. These interactions can alter the distribution of pressure and force coefficients, potentially altering the magnitude and timing of the aerodynamic responses measured in this study. The other limitation is that it can only operate in environments equivalent to 30 kilometers above ground level. This controlled experiment lacks variations in atmospheric density, temperature, wind shear and composition that naturally occur across the layers of the atmosphere.

6.2. Future Scope

The difference created by extending the dataset to a broader range of Mach numbers and atmospheric altitudes is what can be termed the actual re-entry trajectories, capable of creating a complete map of aerodynamic performance. Such data enables multi-condition analysis and enhances the predictive capacity of models, enabling them to accurately predict a range of flight conditions. Further additions to the knowledge of pressure, density and turbulence variations during the descending stages are additional atmospheric properties that vary with altitude. Another avenue of promise is to use high-powered data analysis methods, such as machine learning algorithms, to analyze the results of large-scale simulations and uncover complex patterns in aerodynamic responses. This data-based methodology can be used to develop intelligent, dynamically adaptable control systems. A recent innovation, Active Flow Control technology (with micro air jets and traditional mechanical fins), can counteract turbulence and pressure instabilities. This integrated control approach increases aerodynamic efficiency, minimizes drag penalties and increases maneuverability. All these are aimed at supporting the development of next-generation reusable launch systems which are more precise, resilient and efficient in their operations.

Acknowledgment: The authors sincerely acknowledge the valuable support, encouragement, and academic assistance provided by Sathyabama Institute of Science and Technology, Zewail City of Science and Technology, and Ural Federal University (UrFU) throughout this research. The authors also gratefully recognize the institutional resources, research facilities, and academic guidance provided by these institutions, which contributed to the successful completion of this work.

Data Availability Statement: The data supporting the findings of this study are available from the corresponding author upon reasonable request.

Funding Statement: No specific funding or grant was received for this research from any public, commercial, or not-for-profit funding agency.

Conflicts of Interest Statement: The authors declare that they have no known competing financial interests or personal relationships that could have influenced the work reported in this paper.

Ethics and Consent Statement: The study was conducted in accordance with established ethical guidelines, and participants were assured that their responses would remain confidential and anonymous.

References

1. S. Yadav, M. Jayakumar, A. Nizin, K. Kesavabrahmaji, and N. S. Mohan, "Final phase flight performance and touchdown time assessment of TDV in RLV-TD HEX-01 mission," *J. Inst. Eng. India Ser. C*, vol. 98, no. 6, pp. 679–688, 2017.
2. W. Bauer, P. Rickmers, A. Kallenbach, S. Stappert, V. Wartemann, C. H. J. Merrem, R. Schwarz, M. Sagliano, J. T. Grundmann, A. Flock, T. Thiele, D. Kiehn, A. Bierig, J. Windelberg, E. Ksenik, T. Bruns, T. Ruhe, and H. Elsässer, "DLR reusability flight experiment ReFEx," *Acta Astronautica*, vol. 168, no. 3, pp. 57–68, 2020.
3. D. E. Sanabria and R. E. A. Arndt, "Robust control of a small-scale supercavitating vehicle: From modeling to testing," *Ocean Engineering*, vol. 160, no. 7, pp. 412–424, 2018.
4. X. H. Zhao, Y. Sun, G. L. Zhao, and J. L. Fan, "μ-Synthesis robust controller design for the supercavitating vehicle based on the BTT strategy," *Ocean Engineering*, vol. 88, no. 9, pp. 280–288, 2014.
5. M. Di Clemente, M. Marini, S. Di Benedetto, A. Schettino, and G. Ranuzzi, "Numerical prediction of aerothermodynamic effects on a re-entry vehicle body flap configuration," *Acta Astronautica*, vol. 65, no. 1–2, pp. 221–239, 2009.
6. P. Guilhem and B. Jean-Marc, "Application of robust antiwindup design to the longitudinal aircraft control to cover actuator loss," *IFAC Proceedings Volumes*, vol. 46, no. 19, pp. 506–511, 2013.

7. J. M. Biannic and P. Apkarian, "Anti-Windup Design Via Nonsmooth Multi-Objective H_∞ Optimization," in *Proc. American Control Conf.*, San Francisco, California, United States of America, 2011.
8. H. Fan, Y. Zhang, and X. Wang, "Longitudinal dynamics modeling and MPC strategy for high-speed supercavitating vehicles," in *Proc. Int. Conf. Electric Information and Control Engineering*, Wuhan, China, 2011.
9. B. Vanek, G. J. Balas, and R. E. A. Arndt, "Linear, parameter-varying control of a supercavitating vehicle," *Control Engineering Practice*, vol. 18, no. 9, pp. 1003–1012, 2010.
10. X. Mao and Q. Wang, "Nonlinear Control Design for a Supercavitating Vehicle," *IEEE Trans. Control Systems Technology*, vol. 17, no. 4, pp. 816–832, 2009.
11. L. Daijin, Y. W. Zhang, K. Luo, and J. J. Dang, "Robust Control of Underwater Supercavitating Vehicles Based on Pole Assignment," in *Proc. Int. Conf. Intelligent Human-Machine Systems and Cybernetics*, Hangzhou, China, 2009.
12. J. Jia, D. Fu, and Z. He, "Aerodynamic interactions of a Reusable Launch Vehicle model with different nose configurations," *Acta Astronautica*, vol. 177, no. 12, pp. 58–65, 2020.
13. G. Pezzella, "Aerodynamic and aerothermodynamic design of Future Launchers Preparatory Program concepts," *Aerospace Science and Technology*, vol. 23, no. 1, pp. 233–249, 2012.
14. J. Dzielski and A. Kurdila, "A Benchmark Control Problem for Supercavitating Vehicles and an Initial Investigation of Solutions," *Journal of Vibration and Control*, vol. 9, no. 7, pp. 791–804, 2003.
15. I. N. Kirschner, D. C. Kring, A. W. Stokes, N. E. Fine, and J. S. Uhlman, "Control Strategies For Supercavitating Vehicles," *Journal of Vibration and Control*, vol. 8, no. 2, pp. 219–242, 2002.

Publisher's Note: The publisher remains impartial concerning jurisdictional claims in published maps and institutional affiliations. Responsibility for the content rests entirely with the authors and does not necessarily reflect the publisher's perspectives.