

REVIEWPAPER ON NUMERICAL FLOW INVESTIGATION APPROACH ON LOITERING MUNITION.

Dr. Madhu KS
AssociateProfessor
Department of Aeronautical Engineering
SJC Institute of Technology
Chikkaballapur Karnataka India
manjunathnacharya49@gmail.com

Deekshitha KS Siri
Students
Department of Aeronautical Engineering
SJC Institute of Technology
Chikkaballapur Karnataka India
deekshusiri27@gmail.com

Kushal CR
Students
Department of Aeronautical Engineering
SJC Institute of Technology
Chikkaballapur Karnataka India

Manjunath EN
Students, Department of Aeronautical Engineering, SJC Institute of
Technology, Chikkaballapur Karnataka India
manjunathnacharya49@gmail.com

Sri Sai Harshith
Students, Department of Aeronautical Engineering
SJC Institute of Technology
Chikkaballapur Karnataka India
deekshusiri27@gmail.com

Abstract: This paper presents a computational fluid dynamics (CFD) investigation of a loitering munition configuration operating in subsonic flight conditions. The study analyzes the external flow field, pressure and velocity distribution, and aerodynamic characteristics using steady Reynolds-Averaged Navier–Stokes (RANS) simulations. Geometry modelling, computational domain setup, mesh generation, turbulence model selection, and boundary condition specification are included. Pressure and velocity contours are examined to understand flow behaviour, wake formation, and aerodynamic loads acting on the vehicle. The results provide insights into aerodynamic efficiency, stability, and control performance while suggesting improvements for future simulations and validation.

Keywords: Loitering Munition, Aerodynamic Analysis, Computational Fluid Dynamics, Unmanned Aerial Vehicles, Flow Separation, Wake Dynamics

I. INTRODUCTION

Loitering munitions are unmanned aerial systems capable of remaining airborne for extended durations while searching for targets before engagement. Aerodynamically, they resemble small unmanned aerial vehicles with slender fuselages, mid-mounted wings, and cruciform tail configurations. The aerodynamic design of such vehicles must ensure high endurance during low-speed cruise conditions, stable flight during loitering operations effective aerodynamic control and maneuverability, and minimal disturbance in airflow. Computational Fluid Dynamics (CFD) is widely used to analyze aerodynamic performance during early design stages because it allows engineers to evaluate pressure distribution, velocity fields, and aerodynamic forces without expensive experimental testing.

II. GEOMETRY AND DESIGN DESCRIPTION

The vehicle consists of a slender cylindrical fuselage with a rounded nose section to reduce aerodynamic drag. Mid-mounted wings generate sufficient lift during low-speed cruise and loitering operations. A cruciform tail configuration is used to enhance directional stability and improve control effectiveness during flight.

Design Purpose (Aerodynamic & Systems Intent)

- Provide a stable, efficient aerodynamic platform for long-endurance flight at low to moderate subsonic speeds.
- Maintain controllability and trim margin across the loiter flight envelope.
- Minimize aerodynamic disturbances in the sensor field-of-view and ensure smooth airflow over guidance sensors and control surfaces.
- Allow room for propulsion integration without producing detrimental flow separation or strong asymmetric wakes.

(Design purpose intentionally stated at a system/mission level

— no manufacturing/armament details.)

III DESIGN DETAILS (GEOMETRY OVERVIEW — CONCEPTUAL)

- Fuselage: slender cylindrical body with rounded nose and aft nozzle/attachment region. Smooth transitions at wing and tail junctions.
- Wings: low-aspect mid-span or shoulder-mounted, rectangular or tapered, sized for low-speed cruise and loiter lift.
- Tail: cruciform/elevon arrangement for pitch and yaw control.
- External stores/attachments: small strakes or structural mounts evident in the images; attention paid to junction smoothing following the equation. Use “Eq. 1” or “Equation 1”, not “(1)”, especially at the beginning of a sentence: “Equation 1 is . . .”

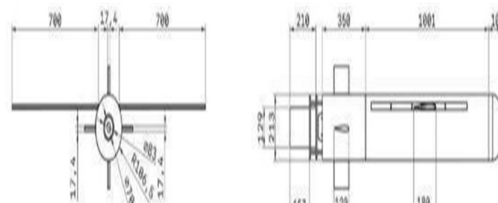


Fig1 Three dimension 3D representation of the loiter munition

IV. Computational Domain Setup

The computational domain is created around the vehicle to simulate external airflow conditions. The domain extends approximately 10–20 body lengths upstream and 20–40 body lengths downstream to minimize boundary effects.

Boundary conditions include:

- Pressure far-field boundary condition
- Gauge pressure of 101325 Pa
- Freestream Mach number between 0.2 and 0.3
- Turbulence intensity of 5%
- Turbulent viscosity ratio of 10
- No-slip wall condition on vehicle surfaces

V. Mesh Generation

A high-quality computational mesh is generated to capture flow behaviour accurately. An unstructured tetrahedral mesh is used with prism boundary-layer inflation near wall surfaces. The mesh contains approximately 176,778 nodes and 938,817 elements. Boundary layer refinement helps capture velocity gradients and surface shear stresses accurately.

VI. TURBULENCE MODELING

Turbulence Model (choice and justification)

- Selected model: Standard two-equation $k-\epsilon$ (RANS)
— a robust, economical model that performs acceptably for attached and mildly separated turbulent external flows. It is widely used for initial design iterations.
 - Alternative candidates (for later stages): $k-\omega$ SST (better near-wall/adverse-pressure-gradient prediction), Reynolds Stress Models (RSM) for strong anisotropy, or Scale-Resolving methods (DES/LES/WALES) for high-fidelity wake dynamics.
- Near-wall treatment
- Solver shows Standard Wall Functions for $k-\epsilon$. That is suitable when the mesh has y^+ in the wall function range; otherwise use low-Re corrections or switch to $k-\omega$ SST for lower y^+ sensitivity.
 - Far-field turbulence intensity: 5% (value visible in your screenshot). This is a moderate turbulence intensity suitable for many preliminary CFD cases; adjust for realistic atmospheric turbulence if required.
 - Turbulent viscosity ratio: 10 (as shown). This is a common default and controls initial eddy viscosity scaling.

VII. EXPECTED RESULTS AND DISCUSSION

Pressure contour analysis shows strong stagnation pressure at the nose region and suction pressure over the upper wing surfaces. Pressure recovery occurs along the fuselage while a low-pressure wake region forms behind the vehicle. Velocity contour analysis indicates accelerated airflow over the wing surfaces, wake formation downstream of the fuselage, and vortex structures near the wing tips. These flow characteristics influence aerodynamic stability and control performance.

Discussion — Technical Interpretation & Implications

1. Drag sources: The major contributors appear to be pressure (form) drag from the blunt nose and wake, and viscous (skin-friction) drag along the fuselage and wing surfaces. Wake size and strength correlate with aft geometry and presence/shape of tail booms. Reducing early separation on the aft fuselage or improving pressure recovery will reduce pressure drag. (Design-level, not prescriptive manufacturing guidance.)
2. Trim & control: The pressure distribution across wings and tails indicates the centre of pressure location relative to the vehicle CG — this affects trimming and required control authority. If the CP is aft of CG, increased tail authority is needed; earlier simulations varying incidence and CG would quantify required control moments.
3. Sensor/avionics placement: Areas of steady airflow with minimal recirculation are preferred for sensors. The images show recirculation in aft sections — avoid placing critical sensors there.
4. Turbulence model limits: $k-\epsilon$ with wall functions provides a cost-effective baseline but has known limitations near strong adverse pressure gradients and separated flows. For accurate prediction of separated wake and vortex dynamics (which influence stall and control), consider $k-\omega$ SST or scale-resolving approaches (DES/LES) in subsequent phases.
5. Mesh sensitivity: The mesh used ($\approx 0.94M$ elements) is moderate; refine the boundary-layer region and wake sampling for improved accuracy. Pay attention to y^+ distributions if moving to low-Re treatments.
6. Operational envelope: The screenshots indicate a solver case with Mach Number set to 1 (likely a default). For loitering mission analysis, ensure freestream Mach is set to the realistic cruise speed (e.g., Mach 0.15–0.35 for small loiterers) to avoid incorrect compressibility effects.

VIII. RESEARCH GAP

Although several studies have been conducted in the fields of Computational Fluid Dynamics, Unmanned Aerial Vehicle Aerodynamics, and Aerodynamics, there are still important limitations in existing research related to aerodynamic investigations of loitering munition configurations.

1. Limited aerodynamic analysis for loitering munition configurations

Most previous studies focus on conventional Unmanned Aerial Vehicles or fixed-wing drones. However, loitering munitions have unique geometric characteristics such as slender fuselages, compact wings, and cruciform tail arrangements. These features create complex aerodynamic flow patterns that are not well studied in current literature.

2. Insufficient CFD-based studies on wake behaviour

Many research works primarily evaluate lift and drag characteristics but do not deeply analyse wake formation and vortex structures behind the vehicle. Wake

characteristics significantly influence aerodynamic drag, stability, and control effectiveness during loitering operations.

3. Lack of investigation under realistic loitering conditions

Some simulations are performed at higher Mach numbers or simplified conditions that do not accurately represent real loitering missions. Loitering munitions typically operate at low subsonic speeds, where flow separation and turbulence effects become more dominant.

4. Limited turbulence model comparison

Previous studies often rely on a single turbulence model without comparing the accuracy of different models. Advanced models such as $k-\omega$ SST or scale-resolving methods are rarely investigated for such small aerial vehicles.

5. Inadequate understanding of flow separation zones

The interaction between the fuselage, wings, and tail surfaces may produce localized separation regions that affect aerodynamic efficiency. However, these separation regions have not been sufficiently studied in existing aerodynamic investigations.

6. Lack of design optimization studies

Most existing research focuses on basic aerodynamic analysis rather than identifying design improvements that could reduce drag or improve aerodynamic performance.

VIII. CONCLUSION

The CFD investigation successfully identified the aerodynamic behaviour of the loitering munition configuration. The analysis showed stagnation pressure at the nose, suction peaks over the lifting surfaces, and wake formation behind the fuselage. These results provide valuable insights for aerodynamic performance evaluation.

Future work should include mesh convergence studies, comparison of turbulence models, transient simulations, and experimental validation through wind tunnel testing. The steady RANS study produced meaningful pressure and velocity fields revealing expected stagnation pressure at the nose, suction peaks on lifting surfaces, and a sizeable wake behind the fuselage and tail.

Preliminary aerodynamic loads can be estimated from these simulations; however, for accurate control sizing and performance estimates, further work is required: mesh convergence, turbulence model comparison ($k-\epsilon$ vs. $k-\omega$ SST), and transient simulations to capture unsteady wake dynamics.

The main aerodynamic improvement targets are reduction of aft separation and improved pressure recovery to lower form drag, and adjustment of planform or tail sizing to ensure required trim margins.

REFERENCES

- [1]. J. D. Anderson, Fundamentals of Aerodynamics, 6th ed., McGraw-Hill, 2017.
- [2]. H. K. Versteeg and W. Malalasekera, An Introduction to Computational Fluid Dynamics: The Finite Volume Method, 2nd ed., Pearson, 2007.
- [3]. T. J. Chung, Computational Fluid Dynamics, Cambridge University Press, 2010.
- [4]. S.B. Pope, Turbulent Flows, Cambridge University Press, 2000.
- [5]. J. Tu, G. Yeoh, and C. Liu, Computational Fluid Dynamics: A Practical Approach, Elsevier, 2018.
- [6]. F. R. Menter, "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications," AIAA Journal, vol. 32, no. 8, pp.1598–1605, 1994.
- [7]. D. C. Wilcox, Turbulence Modeling for CFD, 3rd ed., DCW Industries, 2006.
- [8]. M. Young, The Technical Writer's Handbook. Mill Valley, CA: University Science, 1989.