CFD-Based Optimization of Winglet Shape for Induced Drag Reduction

Adwaith R[1], Adithyan M[2], Jissmon K Raji[3], Kiran K Murali[4], Jobin John[5]

[123]B.Tech S6, Department of Aeronautical Engineering, Toms College of Engineering, Kerala, India
[45]Assistant Professor, Department of Aeronautical Engineering, Toms College of Engineering, Kerala, India

---***---

Abstract - *The aviation industry continuously strives to improve fuel efficiency and reduce operating costs, and induced drag generated by wingtip vortices remains one of the major contributors to aerodynamic losses in fixed-wing aircraft. This project presents a Computational Fluid Dynamics (CFD) based study aimed at optimizing winglet shape to minimize induced drag and enhance overall aerodynamic efficiency. Three winglet configurations blended, canted, and a proposed curved-tip winglet featuring a high-velocity trailing-edge jet were modelled in CATIA and analysed under subsonic cruise conditions (Mach < 0.3) using ANSYS Fluent with the k-ε turbulence model. The lift, drag, and lift-to-drag (L/D) ratio of each configuration were evaluated and compared. Results show that the proposed curved-tip winglet achieves the highest L/D ratio of 16.942, generating a lift of 26.498 N at a drag of 1.564 N, outperforming the blended winglet (L/D = 16.17) and the canted winglet (L/D = 15.566). The curved geometry weakens the wingtip vortex more effectively, indicating improved aerodynamic performance, reduced fuel consumption and lower emissions. The study establishes a repeatable CATIA–ANSYS Fluent workflow for evaluating future winglet designs and recommends experimental validation and higher-fidelity CFD as future work.*

Key Words: Winglet, Induced Drag, Computational Fluid Dynamics (CFD), ANSYS Fluent, CATIA, Curved-Tip Winglet, Lift-to-Drag Ratio, Aerodynamic Optimization

1. INTRODUCTION

The aviation industry continuously seeks improved fuel efficiency and lower operational costs, and a significant share of the drag experienced by a fixed-wing aircraft in cruise is induced drag, which arises from the trailing vortices generated at the wingtips due to the span wise pressure difference between the upper and lower wing surfaces. This induced drag is a major factor reducing aerodynamic efficiency and increasing fuel burn, and consequently has a direct bearing on the operating cost and environmental footprint of an aircraft.

Winglets small, often near-vertical aerodynamic surfaces fitted at the wingtip are widely used to weaken these wingtip vortices and reduce induced drag. However, the precise shape, curvature, cant angle and other geometric parameters of a winglet strongly influence its effectiveness, and conventional winglet shapes are not always optimized for maximum aerodynamic benefit. This project focuses on optimising winglet design using Computational Fluid Dynamics (CFD) to minimize induced drag. An improved winglet geometry is expected to deliver better aerodynamic performance, reduced fuel consumption and lower emissions. CATIA is used for the geometric modelling of the wing and winglet configurations, while ANSYS Fluent is used to perform CFD analysis of the different configurations under subsonic flow conditions.

2. PROBLEM STATEMENT

Aircraft wings inherently generate induced drag due to the formation of wingtip vortices, which reduces the overall lift-to-drag ratio and aerodynamic efficiency of the wing. Conventional wing designs are not fully optimised to minimize these losses, and although winglets are commonly fitted to modern aircraft, their shape and configuration are not always optimised for maximum aerodynamic performance.

The increased drag resulting from sub-optimal wingtip and winglet geometry directly leads to higher fuel consumption, increased operating costs for airlines, and a greater environmental impact in terms of carbon emissions. There is, therefore, a need to systematically evaluate and compare different winglet configurations using CFD in order to identify a shape that more effectively reduces induced drag while improving overall aerodynamic efficiency.

3. OBJECTIVES AND SCOPE

Main Objective: To optimize winglet shape using CFD in order to reduce induced drag and improve the aerodynamic efficiency of a wing.

- To design different winglet geometries (blended, canted/simple, and curved-tip configurations).
- To perform CFD analysis of each configuration and evaluate the resulting reduction in induced drag.
- To compare the aerodynamic performance (lift, drag and lift-to-drag ratio) of the wing for the different winglet configurations.

Scope: The scope of the present work is limited to a CFD-based analysis of winglet configurations under subsonic flow conditions (Mach < 0.3) to evaluate induced drag reduction and the resulting improvement in aerodynamic performance.

Limitations: The study is limited to numerical (CFD) analysis without experimental validation, and considers a simplified wing geometry and steady flow conditions.

4. LITERATURE REVIEW

A summary of the key literature reviewed in support of this project is presented in Table 1.

Author / Year	Title	Methodology
S. A. Ning, I. Kroo (2010)	Tip Extensions, Winglets, and C-wings: Conceptual Design and Optimization	Numerical optimization using Trefftz-plane analysis and vortex lattice modelling with structural weight estimation; compared winglets, tip extensions and C-wings under lift, stall-speed and bending-moment constraints.
S. Reddy et al. (2016)	Multi-Element Winglets: Multi-Objective Optimization of Aerodynamic Shapes	CFD using RANS in Open FOAM for turbulent flow, combined with a multi-objective genetic algorithm and response-surface modelling to optimise lift, drag and pitching moment of a three-element winglet.
K. Takenaka et al. (2008)	Multi-Disciplinary Design Exploration for Winglet	CFD (Euler solver) coupled with FEM-based structural analysis in a multidisciplinary optimization framework; drag components decomposed via mid-field drag decomposition, with Kriging models and genetic algorithms used for optimization.
R. Whitcomb (1976)	A Design Approach and Selected Wind Tunnel Results at High Subsonic Speeds for Winglets	Experimental wind-tunnel testing evaluating winglet performance, demonstrating reduced induced drag and improved lift-to-drag ratio through modified wingtip geometry.

From the above review, it is observed that most studies focus only on induced drag, with limited use of high-fidelity (RANS/LES) CFD or realistic flight conditions such as varying angle of attack, Mach number and flight phase. Advanced winglet configurations including multi-element, blended and bio-inspired designs as well as experimental validation of CFD predictions, remain comparatively under-explored.

5. RESEARCH GAP ANALYSIS

Based on the literature reviewed, the following research gaps were identified and form the basis for the present work:

- **Traditional Wing Design-**Limited focus on wingtip modifications; conventional designs result in high induced drag due to strong wingtip vortices.
- **Limited Advanced Winglets-**Most studies use relatively simple or conventional winglet geometries; multi-element, blended and bio-inspired winglets remain comparatively under-explored.
- **Incomplete CFD Analysis-**Many studies consider induced drag alone, often omitting profile drag, viscous effects, flow separation and high-fidelity RANS/LES turbulence models.
- **Fixed Flight Conditions-**Research is largely restricted to cruise conditions; variations in angle of attack, Mach number and flight phase (take-off, climb, etc.) are rarely considered.
- **Lack of Experimental Validation-**Heavy reliance on numerical simulation, with limited wind-tunnel testing or real flight data available for validation.

6. PROPOSED SYSTEM AND METHODOLOGY

6.1 Proposed System

The present work proposes and evaluates a Curved-Tip Winglet configuration. The proposed configuration features a wingtip geometry that curves smoothly from the wing into the winglet, intended to weaken the wingtip vortex and thereby reduce induced drag. In addition to the curved-tip geometry, the design incorporates a high-velocity jet issuing through a trailing-edge slot near the tip, intended to further influence and disrupt the formation of the tip vortex. The configuration was modelled in CATIA (Figures 1 and 2) and evaluated against blended and canted winglet configurations under subsonic cruise conditions (Mach < 0.3).

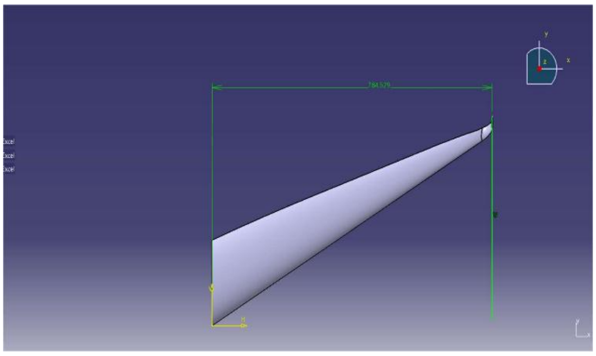

Fig -1: Planform view of the wing with curved-tip winglet, showing overall dimensions (CATIA)

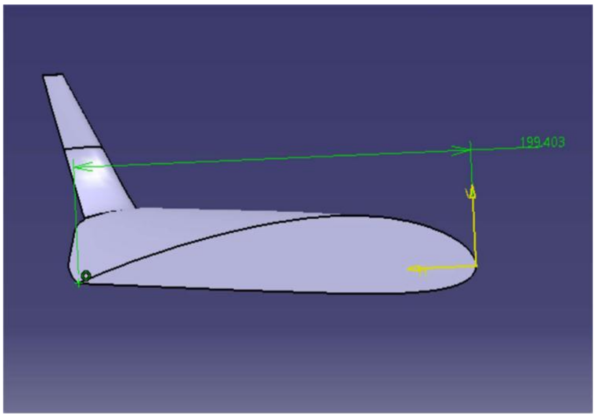

Fig -2: Isometric view of the wing-winglet assembly (CATIA)

6.2 Methodology

The methodology adopted for the present study follows the sequential workflow outlined below:

- **Literature Review**-Review of existing research on winglet aerodynamics and induced-drag reduction.
- **Design of Wing and Winglet**-Selection of blended, canted and curved-tip winglet geometries for comparison.
- **CAD Modelling**-Geometric modelling of the wing-winglet configurations in CATIA.
- **Meshing**-Generation of the computational mesh for each configuration in ANSYS Meshing.
- **CFD Simulation**-Steady-state CFD analysis of each configuration in ANSYS Fluent.
- **Result Analysis**-Post-processing of lift, drag and pressure-contour results for each configuration.
- **Optimization**-Comparison of the configurations to identify the geometry offering the best lift-to-drag ratio.
- **Conclusion**-Summary of findings and recommendations for future work.

7. TOOLS AND SOFTWARE

The CAD and simulation environment used in this study is summarised in Table 2.

Table -2: Tools and software used

Tool	Purpose
CATIA	CAD modelling of the wing and winglet geometry
ANSYS Workbench	Simulation setup and project integration
ANSYS Meshing	Generation of the computational mesh for the flow domain
ANSYS Fluent	CFD analysis and flow-field simulation

8. SIMULATION SETUP

The boundary conditions and flow parameters used for the CFD model are summarized in Table 3.

Table -3: CFD simulation setup and boundary conditions

Parameter	Value / Setting
Free-stream velocity	30 m/s
Fluid	Air
Flow type	Steady, external flow
Flow regime	Subsonic
Mach number	< 0.3
Inlet condition	Velocity inlet (uniform flow)
Outlet condition	Pressure outlet (atmospheric)
Jet velocity	High-velocity jet through trailing-edge slot (curved-tip winglet)
Turbulence model	k-ε model
Operating pressure	Atmospheric pressure
Solver type	Pressure-based solver

Figures 3 and 4 show the computational mesh generated around the wing-winglet section in ANSYS Meshing, with a refined mesh near the aero foil surface and wake region to accurately capture the boundary layer and tip-vortex flow features.

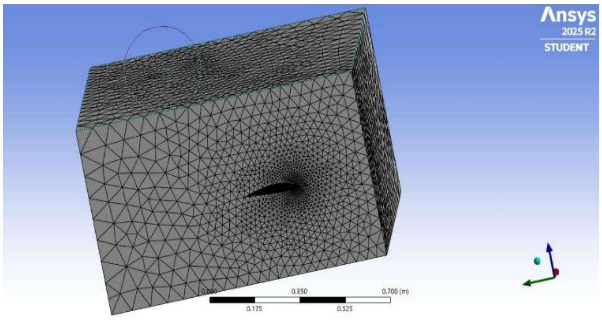

Fig -3: Computational mesh around the wing section (side view, ANSYS Meshing)

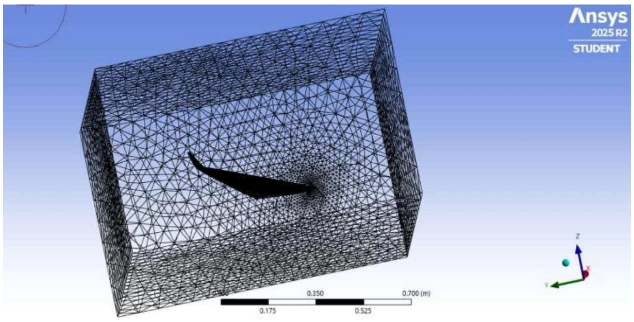

Fig -4: Computational mesh around the wing section (isometric view, ANSYS Meshing)

9. RESULTS AND DISCUSSION

The lift, drag and lift-to-drag (L/D) ratio obtained from the CFD simulations for the blended, canted and proposed curved-tip winglet configurations are summarised in Table 4.

Table -4: Comparison of aerodynamic performance

Winglet Type	Lift (N)	Drag (N)	L/D
Blended Winglet	24.836	1.535	16.17
Canted Winglet	24.316	1.5621	15.566
Curved-Tip Winglet (Proposed)	**26.498**	**1.564**	**16.942**

As summarised in Table 4, the proposed curved-tip winglet achieves the highest lift-to-drag ratio of 16.942, outperforming both the blended winglet (16.17) and the canted winglet (15.566). It also generates the highest lift (26.498 N) among the three configurations while maintaining a drag value (1.564 N) comparable to the other two configurations, indicating a superior overall aerodynamic efficiency.

Figure 5 shows the pressure contour around the curved-tip winglet configuration (the proposed design), Figure 6 shows the corresponding contour for the blended winglet, and Figure 7 shows the contour for the canted winglet, as obtained from ANSYS Fluent.

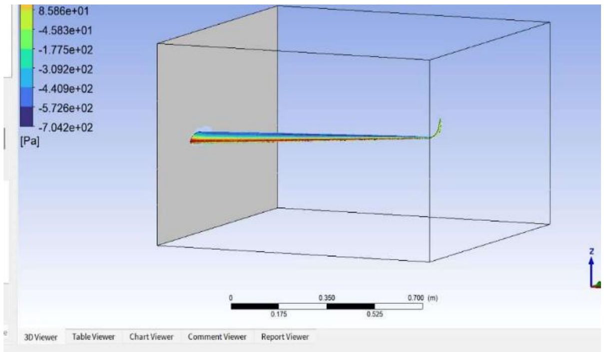

Fig -5: Pressure contour-Curved-Tip Winglet (ANSYS Fluent)

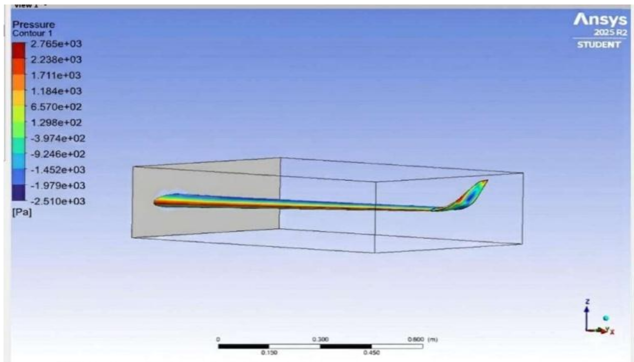

Fig -6: Pressure contour-Blended Winglet (ANSYS Fluent)

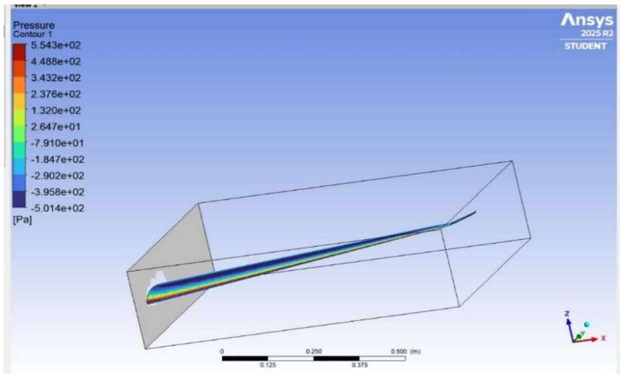

Fig -7: Pressure contour-Canted Winglet (ANSYS Fluent)

The pressure and velocity contours illustrate how the flow structure near the wingtip changes for each configuration. For the curved-tip geometry, the curvature combined with the trailing-edge jet visibly alters the pressure distribution near the tip in a manner consistent with a weakened wingtip vortex, supporting the higher lift-to-drag ratio obtained for this configuration. These results confirm that wingtip geometry significantly affects induced drag, and that the curved-tip winglet with a trailing-edge jet achieves the best lift-to-drag ratio among the configurations studied, suggesting effective mitigation of the wingtip vortex. The findings support winglet shape as an effective lever for improving aircraft fuel efficiency under subsonic cruise conditions.

10. ADVANTAGES

- **Higher L/D Ratio**-Improved lift-to-drag ratio compared to the blended and canted winglets, indicating better aerodynamic efficiency.
- **Reduced Induced Drag**-The curved geometry helps weaken wingtip vortices, directly addressing the primary source of induced drag.
- **Fuel and Emission Savings**-Lower drag translates into reduced fuel consumption and associated emissions over a flight.
- **CFD-Driven Design**-A systematic CAD + CFD workflow (CATIA + ANSYS Fluent) enables rapid, repeatable evaluation of design variants.
- **Flexible Geometry**-The trailing-edge jet and curved-tip concept can be tuned for different operating conditions and aircraft types.
- **Benchmarked Performance**-The proposed design is directly compared against established blended and canted winglet designs for credible, contextual results.

11. FUTURE SCOPE

- **Experimental Validation**-Wind-tunnel testing and comparison with real flight data to validate CFD predictions.
- **Higher-Fidelity CFD**-Use of RANS/LES turbulence models to capture flow separation and viscous effects more accurately.
- **Varied Flight Conditions**-Evaluation across different angles of attack, Mach numbers and flight phases (take-off, climb, etc.).
- **Advanced Winglet Designs**-Exploration of multi-element, blended and bio-inspired winglet geometries beyond the configurations studied here.
- **Structural Analysis**-Coupling of aerodynamic optimisation with structural (FEM) analysis for weight and load constraints.
- **Multi-Objective Optimisation**-Application of genetic algorithms / response-surface methods to jointly optimise lift, drag and pitching moment.

12. CONCLUSIONS

The present CFD-based study on the optimisation of winglet shape demonstrates that wingtip geometry has a significant effect on induced drag and overall aerodynamic efficiency. Among the blended, canted and curved-tip winglet configurations studied, the proposed curved-tip winglet with a trailing-edge jet delivered the highest lift-to-drag ratio of 16.942, compared with 16.17 for the blended winglet and 15.566 for the canted winglet. The CATIA and ANSYS Fluent based workflow developed in this project provides a repeatable framework for evaluating future winglet designs. Further work involving experimental validation and the exploration of advanced winglet geometries can build directly on these results to develop more fuel-efficient wing configurations.

ACKNOWLEDGEMENT

The authors gratefully acknowledge the support and guidance provided by Mr. Jobin John, Head of the Department of Aeronautical Engineering, and Mr. Kiran K Murali, Project Coordinator, Toms College of Engineering, throughout the course of this project.

REFERENCES

[1] S. A. Ning and I. Kroo, “Tip Extensions, Winglets, and C-wings: Conceptual Design and Optimization,” 2010.

[2] S. Reddy et al., “Multi-Element Winglets: Multi-Objective Optimization of Aerodynamic Shapes,” 2016.

[3] K. Takenaka et al., “Multi-Disciplinary Design Exploration for Winglet,” 2008.

[4] R. Whitcomb, “A Design Approach and Selected Wind Tunnel Results at High Subsonic Speeds for Winglets,” NASA Technical Note D-8260, 1976.