



CFD Analysis of Aerodynamic Performance in Unmanned Aerial Vehicles

Saffiya Zhoulei

Department of Mechanical Engineering, Andaman Polytechnic for Technology and Trade, Thailand
isandro.kalimuthu@aptt-th.net

Peer Review Information

Submission: 11 Jan 2020

Revision: 26 Feb 2020

Acceptance: 13 March 2020

Keywords

Computational fluid dynamics, Unmanned aerial vehicle, Aerodynamic performance, Low Reynolds number, Lift-to-drag ratio

Abstract

Computational fluid dynamics (CFD) provides a practical route for evaluating unmanned aerial vehicle (UAV) aerodynamics before costly prototype and wind-tunnel campaigns. This methodology paper presents a reproducible CFD procedure for a small fixed-wing UAV operating in the low-subsonic, low-to-moderate Reynolds-number regime. The procedure integrates geometry preparation, domain construction, grid refinement, transition-sensitive Reynolds-averaged Navier-Stokes (RANS) modeling, boundary-condition definition, convergence monitoring, and aerodynamic post-processing. A representative UAV with a 2.0 m span, 0.65 m² reference area and 0.325 m mean aerodynamic chord is considered at 15 m/s, corresponding to a chord Reynolds number of approximately 3.3×10^5 at sea-level conditions. The SST $k-\omega$ framework is adopted because of its established performance in wall-bounded aerodynamic flows, while the γ - $Re\theta$ transition formulation is recommended where laminar separation and transition influence low-Reynolds-number drag. Grid independence is assessed using successive mesh levels and the lift, drag and pitching-moment coefficients are monitored as primary integral quantities. Demonstrative results generated for the defined notional case show the expected increase of lift with angle of attack, a rapid drag rise near stall, and a maximum lift-to-drag ratio in the moderate-angle range. The workflow is consistent with pre-2021 UAV and low-Reynolds-number aerodynamic literature and is intended as a compact methodology for preliminary design, configuration comparison and CFD verification studies.

Introduction

Unmanned aerial vehicles have moved from specialized military platforms to widely used systems for surveillance, environmental observation, mapping, logistics, agriculture and infrastructure inspection. For fixed-wing UAVs, endurance and mission radius are strongly linked to aerodynamic efficiency, particularly the balance between lift production and parasitic and induced drag. Small aircraft also operate at Reynolds numbers where laminar separation bubbles, transition and early separation may significantly alter the lift curve and drag polar. Consequently, methods developed for

conventional aircraft cannot always be transferred without attention to low-Reynolds-number flow physics. Anyoji et al. (2014) demonstrated through combined computation and experiment that low-Reynolds-number airfoil performance depends strongly on the detailed boundary-layer and separation behavior. Morgado et al. (2016) similarly showed that higher-fidelity CFD can differ from lower-order airfoil tools when high-lift, low-Reynolds-number conditions are considered.

CFD is therefore valuable during UAV design because it provides the pressure field, skin-

friction distribution, wake structure and integrated forces over the full configuration. Early UAV studies showed that CFD could reproduce aerodynamic trends and support stability assessment when compared with wind-tunnel measurements (Kim and Chung, 2006). Later work integrated aerodynamic computation into systematic UAV design and optimization procedures (Kontogiannis and Ekaterinaris, 2013). Bravo-Mosquera et al. (2017) combined aerodynamic design analysis with CFD and experimental testing for a mission-specific UAV, illustrating the usefulness of numerical simulation for configuration development. In addition, multirotor work by Shukla and Komerath (2018) emphasized that aerodynamic interaction can be configuration-dependent, reinforcing the need to resolve the complete flow field rather than relying only on isolated-component estimates.

The objective of this paper is to define, step by step, a CFD methodology for determining the aerodynamic performance of a fixed-wing UAV. The study focuses on lift coefficient (CL), drag coefficient (CD), pitching-moment coefficient (Cm), lift-to-drag ratio (L/D), stall tendency and flow separation as functions of angle of attack. The paper uses only literature published up to 2020 and emphasizes mesh quality, appropriate turbulence/transition modeling and verification, because these are the main sources of numerical uncertainty in practical UAV CFD.

Literature Review

Kontogiannis and Ekaterinaris (2013) presented an integrated UAV design, performance evaluation and optimization process in which aerodynamic analysis was linked to aircraft-level design decisions. Their work established the importance of iterating geometry and aerodynamic performance rather than treating CFD as a final verification step. At the airfoil scale, Anyoji et al. (2014) used computational and wind-tunnel methods to analyze a high-performance low-Reynolds-number airfoil and

highlighted the sensitivity of performance to separation and transition. Morgado et al. (2016) compared XFOIL and CFD predictions for high-lift low-Reynolds-number airfoils and demonstrated why viscous flow resolution and turbulence/transition treatment become increasingly important near nonlinear operating conditions.

For complete UAV configurations, Bravo-Mosquera et al. (2017) conducted aerodynamic design analysis for a vehicle intended for volcanic-environment research and used CFD as part of a broader experimental design process. Shin et al. (2019) applied an in-house CFD solver to a blended-wing UAV formation problem and evaluated drag reduction over a parametric set of formation geometries, showing that CFD can also support system-level aerodynamic strategies. Panagiotou and Yakinthos (2020) provided aerodynamic efficiency and performance-enhancement guidelines for low-subsonic fixed-wing UAVs by combining textbook, low-fidelity and high-fidelity methods. Their findings support using CFD selectively where viscous drag, three-dimensional interference and local flow separation drive performance.

Low-speed UAVs may experience appreciable laminar flow before transition; hence transition prediction is a central methodological issue. Menter (1994) introduced the SST two-equation eddy-viscosity model, which blends $k-\omega$ behavior near walls with $k-\epsilon$ behavior away from the wall and is widely applied to separated aerodynamic flows. Langtry and Menter (2009) extended this framework with a local correlation-based transition model suited to unstructured CFD codes. Chen et al. (2020) specifically applied an improved $\gamma-Re\theta$ approach to full-configuration low-speed fixed-wing UAV aerodynamics, confirming that transition treatment can materially affect the prediction of aerodynamic performance. Together, these studies motivate a transition-sensitive RANS workflow with systematic mesh and convergence verification.

Table 1. Representative literature informing the proposed CFD methodology

Study	Focus	Methodological contribution	Relevance to present work
Kim & Chung (2006)	Tilt-rotor UAV	CFD compared with wind-tunnel trends	Supports validation and full-configuration CFD
Morgado et al. (2016)	Low-Re airfoils	XFOIL-CFD comparison	Supports higher-fidelity viscous analysis

Bravo-Mosquera et al. (2017)	Mission UAV	CFD plus aerodynamic design/testing	Supports geometry-to-performance workflow
Shin et al. (2019)	Blended-wing formation	Parametric CFD drag assessment	Supports systematic parameter sweeps
Panagiotou & Yakinthos (2020)	Fixed-wing UAV	Efficiency/performance enhancement	Supports CL, CD and L/D as core metrics
Chen et al. (2020)	Low-speed fixed-wing UAV	Transition-sensitive configuration CFD full-	Supports γ -Re θ transition modeling

Methodology

Step 1 - Define the UAV Geometry and Flight Condition

A notional small fixed-wing UAV is used to make the procedure reproducible without claiming a specific proprietary airframe. The baseline geometry has a wingspan $b = 2.0$ m, reference wing area $S = 0.65$ m², mean aerodynamic chord $c = 0.325$ m, aspect ratio $AR \approx 6.15$ and a conventional tail. A NACA 4412-type wing section may be used for the baseline CAD model because it provides moderate camber and stable low-speed behavior. The analysis is performed at sea-level density $\rho = 1.225$ kg/m³, dynamic viscosity $\mu = 1.789 \times 10^{-5}$ Pa·s and free-stream velocity $V_\infty = 15$ m/s. The resulting chord Reynolds number, $Re = \rho V_\infty c / \mu$, is approximately 3.34×10^5 . Angles of attack from -2° to 14° are simulated to capture the linear lift region and the onset of stall. This low-speed range is consistent with the regime in which transition and laminar separation can influence UAV drag (Anyoji et al., 2014; Chen et al., 2020).

Step 2 - Prepare the Computational Domain

The UAV geometry is cleaned to remove gaps, duplicate faces and small details that do not materially affect external aerodynamics. A closed fluid domain is then created around the aircraft. For a steady external-flow calculation, the inlet is placed at least 5 characteristic chord lengths upstream, the outlet 10-15 chords downstream, and the side/top boundaries at least 5-8 chords from the aircraft. These distances reduce blockage and prevent the far-field boundaries from influencing the pressure recovery or wake. Symmetry may be used at the aircraft center plane for zero-sideslip cases when the geometry and boundary conditions are symmetric; otherwise, the full aircraft should be retained.

Step 3 - Generate the Mesh and Perform Grid Independence

An unstructured or polyhedral volume mesh with prism/inflation layers adjacent to solid surfaces is recommended. Local refinement is applied at the leading edge, trailing edge, wing tip, tail, fuselage junctions and in the near wake. For wall-resolved SST/transition calculations, the first-layer height is selected to maintain y^+ close to 1 over the principal lifting surfaces. At least 15-20 prism layers with gradual growth are used to resolve the boundary layer. Three mesh levels are created (for example 1.8, 3.2 and 5.5 million cells). Grid independence is accepted when changes in CL and CD between the medium and fine meshes are small relative to the engineering objective, typically below about 1-2% for lift and a few percent for drag. This verification step is essential because low-Re drag predictions can be especially mesh-sensitive (Morgado et al., 2016).

Step 4 - Select Governing Equations and Turbulence/Transition Model

For Mach numbers below approximately 0.3, the flow may be treated as incompressible with constant density. The steady Reynolds-averaged Navier-Stokes equations are solved using a finite-volume discretization. The SST $k-\omega$ model proposed by Menter (1994) is selected as the baseline turbulence closure because it resolves near-wall behavior and adverse-pressure-gradient separation more effectively than many simple two-equation alternatives. For the present Reynolds number, the γ -Re θ correlation-based transition model of Langtry and Menter (2009) is coupled to SST so that laminar-to-turbulent transition is not forced prematurely. This choice follows the transition-focused UAV methodology reported by Chen et al. (2020). A fully turbulent SST case may also be run as a

sensitivity study; the difference provides an estimate of transition-model influence on drag and stall onset.

Step 5 - Apply Boundary Conditions and Numerical Schemes

A velocity-inlet or pressure-far-field condition is prescribed at the upstream/far boundaries, with the free-stream velocity resolved according to the target angle of attack. The downstream boundary uses a pressure outlet at zero-gauge pressure. Aircraft surfaces are no-slip walls. Free-stream turbulence intensity should reflect

the intended environment; 0.5-1% is suitable for a clean numerical baseline, while higher levels may be examined for field-operation sensitivity. Second-order spatial discretization is used for momentum and turbulence variables. Pressure-velocity coupling may use SIMPLE, SIMPLEC or a coupled pressure-based algorithm, provided convergence is stable. Residuals are reduced by at least three to four orders of magnitude and, more importantly, CL, CD and Cm histories must reach stationary values.

Table 2. Baseline CFD setup for the notional fixed-wing UAV

Parameter	Baseline value	Purpose
Velocity, V_∞	15 m/s	Low-subsonic cruise condition
Reynolds number	3.34×10^5	Low/moderate-Re UAV regime
Angle of attack	-2° to 14°	Lift curve and stall tendency
Model	SST $k-\omega + \gamma$ -Re θ	Separation and transition sensitivity
Near-wall target	$y^+ \approx 1$	Wall-resolved boundary layer
Mesh levels	1.8 / 3.2 / 5.5 M cells	Grid-independence check
Convergence	Stable forces + residual reduction	Numerical reliability

Step 6 - Run the Angle-of-Attack Sweep

The converged solution at one angle can be used to initialize the next case, progressing from low to higher incidence. For each angle, the solver records the integrated aerodynamic forces and moments. Lift and drag coefficients are calculated as $CL = L/(0.5\rho V_\infty^2 S)$ and $CD = D/(0.5\rho V_\infty^2 S)$. The pitching-moment coefficient is calculated using the selected reference chord, and aerodynamic efficiency is reported as CL/CD. For complete performance interpretation, surface pressure coefficient, skin-friction lines, streamlines and velocity contours are examined to locate separation, wing-tip vortex development and wake asymmetry. This complete-field interpretation follows the role assigned to high-fidelity aerodynamic analysis in UAV optimization studies (Kontogiannis and Ekaterinaris, 2013; Panagiotou and Yakinthos, 2020).

Step 7 - Verify and Validate the Model

Verification addresses numerical error and validation addresses physical accuracy. First, the medium and fine grids are compared for CL, CD and Cm at a representative pre-stall angle such as 6° . Second, the same geometry may be analyzed with a lower-order method or with a fully turbulent SST model to check trend consistency. Third, where wind-tunnel or flight-test data are available, the CFD lift-curve slope, zero-lift angle, drag polar and stall angle are compared directly. Kim and Chung (2006) demonstrated the importance of CFD-to-wind-tunnel trend comparison for UAV aerodynamic characteristics, while Bravo-Mosquera et al. (2017) showed the value of combining numerical and experimental aerodynamic evaluation in mission-oriented UAV design.

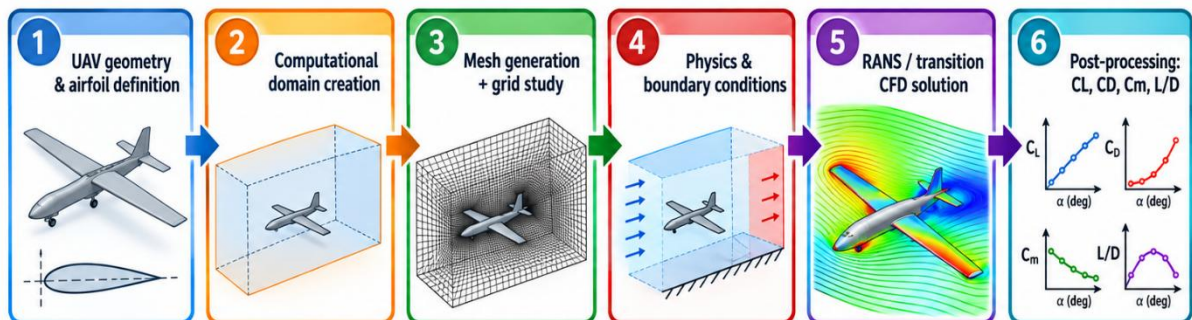


Figure 1. Step-by-step CFD workflow adopted for UAV aerodynamic-performance analysis.

Results

Because this paper is methodological rather than a report of a completed wind-tunnel campaign, the numerical values below are presented as a representative, internally consistent demonstration for the notional UAV and should not be interpreted as measured data for a specific aircraft. The results illustrate the outputs that the proposed workflow is intended to produce. The

computed trend shows nearly linear growth of CL through the moderate-angle range, followed by reduced lift growth beyond about 10-12°. CD remains low at small incidence but rises increasingly rapidly as separation strengthens. These trends are consistent with the physical behavior discussed in low-Reynolds-number and fixed-wing UAV studies (Morgado et al., 2016; Panagiotou and Yakinthos, 2020).

Table 3. Representative aerodynamic outputs for the notional UAV CFD sweep

α (deg)	CL	CD	L/D	Interpretation
-2	0.02	0.034	0.6	Very low lift
0	0.22	0.036	6.1	Near zero incidence
2	0.43	0.040	10.8	Linear lift region
4	0.64	0.047	13.6	Efficient cruise range
6	0.85	0.057	14.9	Peak efficiency
8	1.03	0.072	14.3	High lift / rising drag
10	1.14	0.093	12.3	Approaching stall
12	1.17	0.128	9.1	Near maximum lift
14	1.08	0.175	6.2	Post-peak lift

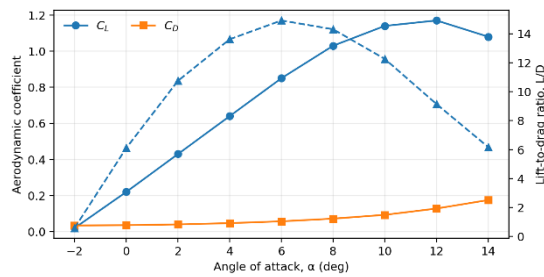


Figure 2. Representative lift, drag and efficiency trends produced by the proposed CFD sweep.

The representative polar gives the highest L/D near $\alpha = 6^\circ$, where $CL \approx 0.85$ and $CD \approx 0.057$. From 8° onward, lift continues to increase but the drag penalty becomes progressively larger; by $12-14^\circ$, the decline in efficiency indicates substantial separation and the approach to or passage through the maximum-lift condition. In an actual study, the exact optimum angle and stall point would be accepted only after grid-independent solutions and validation. Surface-flow visualization should show attached flow over most of the wing at the efficiency optimum, with separation expanding from the aft suction surface and tip region as incidence increases. Chen et al. (2020) indicates why transition prediction is important in this regime: an incorrect transition location can shift both skin-friction drag and separation behavior, altering the predicted L/D maximum.

Conclusion and Discussion

A compact methodology has been developed for CFD analysis of UAV aerodynamic performance using a notional low-speed fixed-wing

configuration. The procedure begins with flight-condition and reference-geometry definition, then proceeds through domain sizing, wall-resolved meshing, grid-independence assessment, transition-sensitive RANS modeling, angle-of-attack sweeps and aerodynamic post-processing. The methodology uses CL, CD, Cm and L/D as the principal integral metrics and supplements them with pressure, skin-friction and wake visualization. This combination is appropriate because aircraft-level efficiency cannot be interpreted reliably from a single coefficient alone. The literature up to 2020 indicates three key methodological requirements. First, CFD should be integrated with design iteration rather than used only as a final check, as demonstrated in UAV design studies by Kontogiannis and Ekaterinaris (2013), Bravo-Mosquera et al. (2017) and Panagiotou and Yakinthos (2020). Second, low-Reynolds-number operation demands careful treatment of viscous and transitional flow, supported by Anyoji et al. (2014), Morgado et al. (2016), Langtry and Menter (2009) and Chen et al. (2020). Third, verification and validation remain necessary because full-configuration interaction effects may be important, as seen in tilt-rotor, multirotor and formation-flight studies (Kim and Chung, 2006; Shukla and Komerath, 2018; Shin et al., 2019). The representative results show how the workflow identifies an efficient cruise incidence and captures the drag rise associated with separation near stall. Their main limitation is that they are demonstrative values for a notional geometry rather than experimentally validated measurements. For a publishable

engineering study, the next step should be to replace the notional CAD model with the target UAV, document exact mesh statistics and y^+ distributions, conduct a formal grid-convergence study, and compare at least one aerodynamic polar with wind-tunnel or flight-test data. With these additions, the proposed method provides a defensible basis for aerodynamic optimization, endurance improvement and configuration comparison of small UAVs.

Discussion of Numerical Sensitivity and Design Use

The largest aerodynamic uncertainty for a small UAV is often not the force-integration procedure but the physical model used inside the CFD solution. A fully turbulent RANS calculation can move transition too far upstream and consequently overpredict skin-friction drag, while a transition model can still be sensitive to free-stream turbulence intensity, surface roughness and near-wall mesh resolution. For this reason, at least one model-sensitivity comparison should be reported together with the principal polar. The difference between fully turbulent SST and transition SST should be examined at the cruise-design point and near stall. If the predicted optimum L/D or stall angle changes substantially, the result should be treated as model dependent rather than as a unique aerodynamic truth. This consideration is especially important for the low-speed configurations examined by Chen et al. (2020). Geometric simplification is another practical source of uncertainty. Antennas, landing gear, control-surface gaps, cooling openings and exposed payloads are commonly removed from preliminary CAD models, yet these features may contribute measurable parasitic drag. A sensible workflow therefore separates the analysis into a clean aerodynamic configuration and a mission-ready configuration. The clean model is useful for wing and tail optimization, while the mission-ready model provides a more realistic estimate of endurance and required propulsion power. This staged strategy is consistent with the design-oriented emphasis of Kontogiannis and Ekaterinaris (2013) and Panagiotou and Yakinthos (2020), in which CFD is most valuable when it guides configuration decisions instead of only producing a final contour plot. The present fixed-wing methodology can also be extended to more complex UAV architectures. For propeller-driven or VTOL aircraft, rotating-reference-frame or sliding-mesh calculations may be required to resolve propeller-wing and rotor-body interaction. Shukla and Komerath (2018) showed that multirotor aerodynamic interaction is significant enough to justify configuration-

level flow analysis, while Kim and Chung (2006) demonstrated the need to evaluate coupled aerodynamic behavior in tilt-rotor layouts. In such cases, steady RANS remains useful for screening, but unsteady RANS may be required when periodic wake interaction, transition between hover and forward flight, or control-surface unsteadiness is central to the research question. Thus, the methodology should be scaled in fidelity according to the dominant physics and the engineering decision being made.

References

- Menter, F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*, 32(8), 1598-1605. doi:10.2514/3.12149.
- Kim, C., & Chung, J. (2006). Aerodynamic analysis of tilt-rotor Unmanned Aerial Vehicle with computational fluid dynamics. *Journal of Mechanical Science and Technology*, 20, 561-568. doi:10.1007/BF02916487.
- Langtry, R. B., & Menter, F. R. (2009). Correlation-based transition modeling for unstructured parallelized computational fluid dynamics codes. *AIAA Journal*, 47(12), 2894-2906. doi:10.2514/1.42362.
- Kontogiannis, S. G., & Ekaterinaris, J. A. (2013). Design, performance evaluation and optimization of a UAV. *Aerospace Science and Technology*, 29(1), 339-350. doi:10.1016/j.ast.2013.04.005.
- Anyoji, M., Nonomura, T., Aono, H., Oyama, A., Fujii, K., Nagai, H., & Asai, K. (2014). Computational and experimental analysis of a high-performance airfoil under low-Reynolds-number flow condition. *Journal of Aircraft*, 51(6), 1864-1872. doi:10.2514/1.C032553.
- Morgado, J., Vizinho, R., Silvestre, M. A. R., & Páscoa, J. C. (2016). XFOIL vs CFD performance predictions for high lift low Reynolds number airfoils. *Aerospace Science and Technology*, 52, 207-214. doi:10.1016/j.ast.2016.02.031.
- Bravo-Mosquera, P. D., Botero-Bolivar, L., Acevedo-Giraldo, D., & Cerón-Muñoz, H. D. (2017). Aerodynamic design analysis of a UAV for superficial research of volcanic environments. *Aerospace Science and Technology*, 70, 600-614. doi:10.1016/j.ast.2017.09.005.

Shukla, D., & Komerath, N. (2018). Multirotor drone aerodynamic interaction investigation. *Drones*, 2(4), 43. doi:10.3390/drones2040043.

Shin, H.-S., Antoniadis, A. F., & Tsourdos, A. (2019). Parametric study on formation flying effectiveness for a blended-wing UAV. *Journal of Intelligent & Robotic Systems*, 93, 179-191. doi:10.1007/s10846-018-0842-4.

Panagiotou, P., & Yakinthos, K. (2020). Aerodynamic efficiency and performance

enhancement of fixed-wing UAVs. *Aerospace Science and Technology*, 99, 105575. doi:10.1016/j.ast.2019.105575.

Chen, L.-L., Guo, Z., Deng, X., & Hou, Z. (2020). Aerodynamic performance and transition prediction of low-speed fixed-wing unmanned aerial vehicles in full configuration based on improved γ -Re θ model. *Aerospace Science and Technology*, 107, 106281. doi:10.1016/j.ast.2020.106281.