

Comparison of Aerodynamic Analysis Tools Applied to a Propeller-Blown Wing

Vivek Ahuja*

Research in Flight, Austin, TX 78750, U.S.A.

Brandon L. Litherland†

NASA Langley Research Center, Hampton, VA 23681, U.S.A.

This paper describes initial results from an ongoing activity comparing two aerodynamic analysis tools: OVERFLOW, a Navier-Stokes flow solver, and FlightStream, a viscous surface-vorticity, unstructured panel method flow solver. Both tools are applied to analyses of an unblown wing and a propeller-blown wing, and then comparisons are performed between the predicted aerodynamic loads and moments from each tool. Computational performance and efficiencies of each tool and their usability in conceptual and preliminary air vehicle design are also discussed. A description of the workflow, underlying theoretical foundations, and best practices are provided. In comparing FlightStream and OVERFLOW, reasonable agreement between predicted performance is observed for flow regimes of interest at high angle of attack prior to stall, including blown wing aero-propulsive effects on lift augmentation and stall delay. In the post-stall regions, FlightStream appears to underestimate the effects of turbulent flow separation that contribute heavily to drag. However, as expected for a high-fidelity, time-accurate solver, the setup and computation time are substantially greater for OVERFLOW than for FlightStream.

Nomenclature

c	=	reference mean aerodynamic chord
C_D	=	drag coefficient
CF_X	=	force coefficient in X-direction (aft)
CF_Y	=	force coefficient in Y-direction (right)
CF_Z	=	force coefficient in Z-direction (up)
C_L	=	lift coefficient
C_M	=	pitching moment coefficient
KTAS	=	knots true air speed
R	=	propeller radius
r	=	local blade radial distance
r_0	=	blade root radial distance
S_{ref}	=	reference area (wing planform area)
T	=	thrust
V_{inf}	=	freestream air speed
α	=	angle of attack
β	=	local blade twist
$\beta_{3/4}$	=	blade pitch measured at 3/4 radius
η	=	propeller efficiency
ρ	=	air density

* CEO, Research in Flight, Austin, Texas 78641, AIAA Senior Member

† Aerospace Engineer, Aeronautics Systems Analysis Branch, 1 N. Dryden St. MS 442.

I. Introduction

Over the last decade or so, the emergence of distributed electric propulsion (DEP) in aviation has added a new paradigm to the aircraft design problem. DEP enables flexibility in propulsor location, thereby enabling highly unconventional aircraft designs, which may be suited to new concepts of operation such as Advanced Air Mobility (AAM). Some DEP designs augment airflow over wings with a collection of electric motor-driven high-lift propellers (HLP) for lift augmentation in low-speed flight conditions, such as on the NASA X-57 Maxwell flight demonstrator [1]. Other designs can provide additional thrust for vertical or short takeoff and landing (V/STOL) [2]. Several DEP-enabled concepts aim to realize favorable interactions between the airframe aerodynamics and strategically integrated propulsors, achieving hitherto unrealizable performance benefits.

However, the same highly coupled aero-propulsive interactions are difficult to model with sufficient levels of accuracy in early design stages (conceptual and preliminary). Commercial and open-source low- and mid-fidelity tools implementing vortex lattice methods, surface vorticity methods, seeded particle methods, or lifting line theory have emerged to fill this capability gap. At issue with the various approaches to modeling the aero-propulsive interactions is not necessarily the numerical simulation capability, but rather validation of said capability on the spectrum of computational fidelity and computational performance on representative configurations. Further, in the high-fidelity spectrum, there is a dearth of best practices for the modeling of such scenarios. Underlining this problem is the general lack of well-defined geometries with sufficient validation data in the public domain.

This paper supports multiple efforts at NASA and in the industry that seek to address the lack of validated conceptual and preliminary design tools applicable to analysis of propeller-blown wings. The primary objective of this work is to compare two established computational tools at low to moderate angles of attack: (1) the high-fidelity Navier-Stokes flow solver, OVERFLOW (OVERset grid FLOW solver) [3], and (2) a low-to-mid fidelity unstructured panel method flow solver with an integrated viscous-module, FlightStream [4]. A secondary objective is to evaluate simulations of high angles of attack from both solvers to determine if stall behavior is similarly captured. The second objective of this study extends well beyond the range of wing angles of attack that exhibit linear aerodynamics to investigate nonlinear aerodynamics near stall conditions. Eventually, the ability to model higher angles of attack will enable the study of transition between forward flight and vertical flight in UAM vehicles.

The OVERFLOW data presented in this work are a continuation of the analyses discussed in a prior comparison of two vortex lattice methods, VSPAERO and RoBIN, to OVERFLOW [5]. In the prior comparison study, the OVERFLOW simulations showed expected behaviors up to 10 degrees angle of attack but failed to exhibit the expected lift losses at higher angles of attack. This was due to an insufficient number of steps in the solution to effectively mature the total flow around the wing. Therefore, significantly longer simulations were performed in this new study by running both the wing in isolation and the blown wing configuration to a relatively large number of iterations in an attempt to resolve the total flow. For example, the approach in earlier X-57 analyses was to simulate the HLP by running 8 revolutions at 1° per step for a total of 2880 steps [6, 7]. At reasonable freestream speeds and low angles of attack, this was sufficient to capture the propeller wake interacting with the blown wing. For high angle of attack cases in this study, the propeller was run to 32 revolutions at 1° per step.

This paper summarizes the model geometry analyzed, a derivative of the NASA X-57 wing and HLP, followed by a discussion of the tools and methods implemented in gridding and analysis. The paper will also highlight the emerging best practices for OVERFLOW simulations for this class of aero-propulsive problems. Results from simulations in FlightStream and OVERFLOW are presented along with a brief discussion of future work.

II. Model Geometry

This work is a continuation of prior analyses and comparisons performed in 2020-2021 [5], and, therefore, the wing and propeller geometry is the same as that described in Ref. [5]. The wing, illustrated in Fig. 1 along with the HLP, is a rectangular section having no twist, dihedral, or taper with a constant airfoil based on the X-57 main wing root airfoil [8]. Table 1 describes the wing dimensions and placement relative to the global origin, which is at the propeller center of rotation for both the blown and unblown analyses.

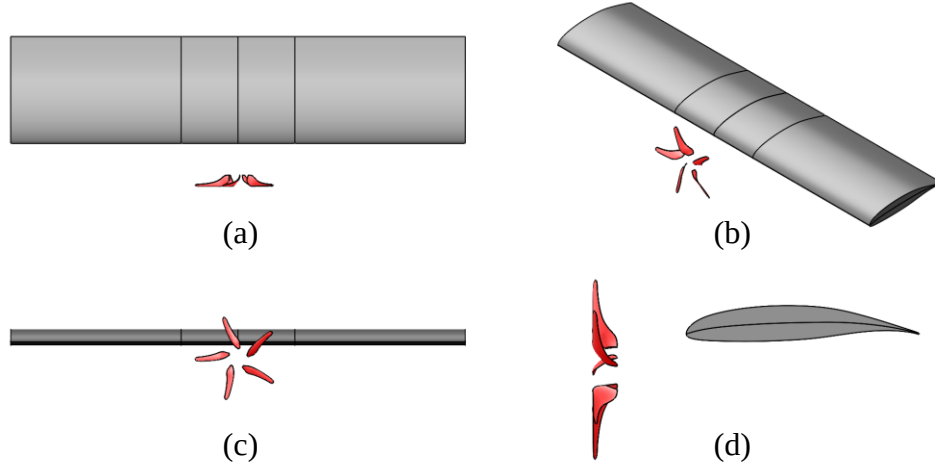


Fig. 1 Geometry of wing and propeller used as the basis for analysis. Views include (a) top, (b) isometric, (c) front, and (d) left side.

Table 1 Wing Parameters

Parameter	Value
Wing Chord	0.7141 m
Wingspan	3.048 m
Wing Area	2.177 m ²
Wing Leading Edge Center, XYZ	0.2588 m, 0.0000 m, 0.1241 m
Wing Quarter Chord Location, XYZ	0.4376 m, 0.0000 m, 0.1241 m

The X-57 HLP is designed to have a relatively constant induced velocity aft of the propeller according to the methods described by Patterson, Borer, and German [9]. The propeller root chord was modified from the original design to accommodate folding and stowage along the nacelle by removing material from the root leading edge to avoid cutting into the spinner [10]. The blades have a constant MH114 airfoil from root to tip, which was modified to include a 0.020 inch (0.508 mm) blunt trailing edge thickness [6]. Table 2 describes the HLP dimensions and design flight conditions set for the analyses in this study.

Table 2 High-Lift Propeller Design Parameters

Parameter	Value
Radius (R)	0.28769 m
Blade Root Radius (r_0)	0.0583 m
3/4 Pitch ($\beta_{3/4}$)	27.4°
Number of Blades	5
Speed	4549 RPM
Shaft Power	10.5 kW
Thrust	220 N
Flight Velocity	29.9 m/s (58 KTAS), Sea Level
Center Location, XYZ	0.000 m, 0.000 m, 0.000 m

The distribution of chord, twist, rake, and skew along the propeller radius is shown in Fig. 2. Rake and skew are defined relative to the local blade chord* as shown in Fig. 3. Positive rake is in the direction perpendicular to the local chord line in the airfoil plane and points away from the blade lower surface. Positive skew is in the direction parallel to the local chord line in the airfoil plane and points toward the trailing edge.

* Rake and skew differ from dihedral and sweep in that rake and skew are translations relative to the local chord while dihedral and sweep are angular deflections relative to the propeller plane of rotation, typically measured at the blade leading edge.

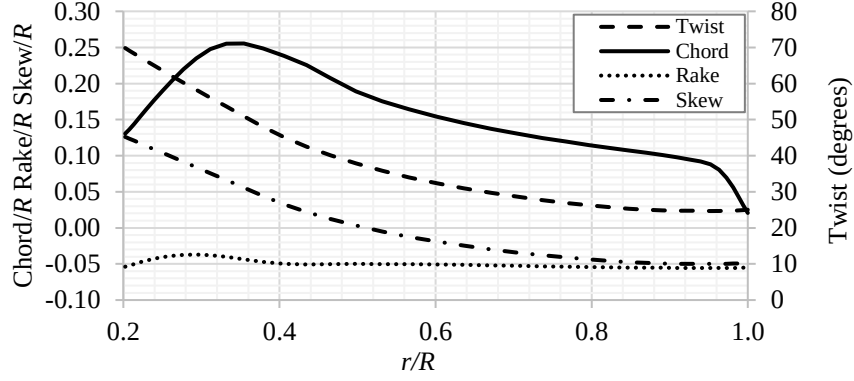


Fig. 2 High-lift propeller blade geometry.

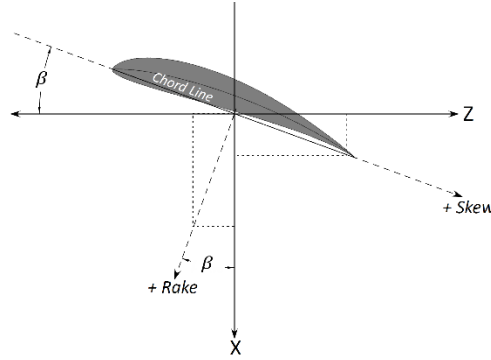


Fig. 3 Rake and skew directions relative to a local blade airfoil section with twist angle of β .

III. Modeling Tools and Methods

The model geometry for both FlightStream and OVERFLOW was created in Open Vehicle Sketch Pad (OpenVSP) [11, 12]. The geometry was then exported to FlightStream for refinement and analysis or to the Chimera Grid Tools overset grid generation software. After creating the volumetric grids for analysis in OVERFLOW, the required input files were written for the case being simulated. The OVERFLOW simulations were run on the NASA Langley Research Center K-Cluster high-performance computing (HPC) resource. The OpenVSP model of this configuration is publicly available on the OpenVSP Hangar model library [13]. The following subsections describe these tools and the methods in greater detail.

A. OpenVSP

The wing geometry, propeller geometry, and the associated files needed for export to grid generation and aerodynamic analysis tools were created in OpenVSP. OpenVSP is an open-source, parametric-geometry-based computer modeling environment originally funded by NASA [11, 12]. The software enables users to rapidly generate a variety of designs with aircraft-native components, such as wings, fuselages, ducts, and propellers. OpenVSP facilitates the initial creation of conceptual models as well as detailed approximations of existing vehicles and objects. OpenVSP can also export geometry in a variety of formats, including PLOT3D and TRI, which served as a basis for grid creation in this study.

B. Chimera Grid Tools and OVERFLOW

1. Grid Parameters

The overset grid systems required to perform OVERFLOW CFD analyses were created in Chimera Grid Tools (CGT) according to the best practices established by Chan, Gomez, Rogers, and Buning [14]. CGT software contains several pre- and post-processing tools for the Chimera overset approach to gridding for CFD analyses on complex geometries [15]. The wing and blades were modeled as O-type grids wrapping around the geometry in the chordwise direction, and several planar edges were removed from the wing grid ends to avoid coincident edges with the square wing cap grids. The caps for each geometry were also modeled as O-type grids generated from the leading edges,

camber lines, and trailing edges. All surface and volume grids were generated with hyperbolic growth ratios of approximately 1.1 or less. A breakdown of the grid parameters for both the blown and unblown geometries is provided in Table 3. A view of the total near-body blown wing grid system is provided in Fig. 4, with a detailed view of the blade grids shown in Fig. 5.

Table 3 Grid Parameters for Blown and Unblown Geometries

Parameter	Wing	Blade
Number of Chordwise Points	221	201
Number of Spanwise Points	393	181
Near-Body Distance (Number of Points)	0.9144 m (181)	0.0914 m (91)
Parameter	Value	
Viscous y^+ spacing (y distance)	0.5 (3.048×10^{-6} m)	
Off-Body Grid Distance	60.96 m	
Minimum Off-Body Grid Spacing	0.0209 m	
Unblown Total Vertex Count	24,296,367	
Blown Total Vertex Count	44,457,872	

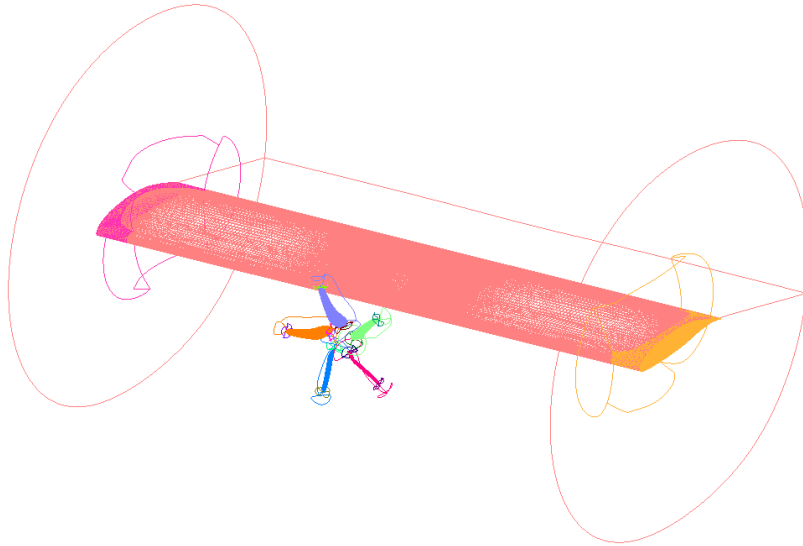


Fig. 4 Near-body grids for blown wing configuration. Volume grid boundary edges shown.

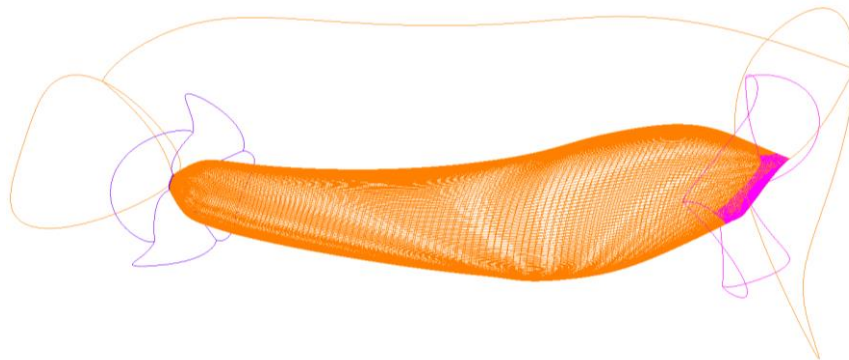


Fig. 5 Near-body grids for HLP blade. Volume grid boundary edges shown.

2. OVERFLOW Setup

Unsteady Reynolds-Averaged Navier-Stokes (URANS) cases and Delayed Detached Eddy Simulations (DDES) were performed with OVERFLOW, a structured, overset grid, computational fluid dynamics flow solver developed by NASA [3, 16]. A fifth-order WENO5M (Weighted Essentially Non-Oscillatory) method [17] with a HLLE++

(Harten, Lax, van Leer and Einfeldt) flux scheme [18] was implemented to calculate inviscid flux contributions, and second-order central differences were used to approximate the viscous flux. For the blown wing cases, a $k-\omega$ SST-RC-QCR2000 (rotation correction with quadratic constitutive relationship) turbulence model [19, 20] coupled with the Langtry-Menter CFX-v-1.1 transition model [21] was used to maintain commonality with previous X-57 studies [6, 10]. This method was successfully demonstrated in other rotor simulations with rotational reference frames, e.g., Refs. [22, 23]. Dual time-stepping with 20 subiterations and a physical time step size corresponding to 1° of propeller rotation were used to predict the time accurate flow field, with at least eight revolutions of travel being simulated at lower angles of attack (as with prior X-57 propeller simulations [6, 7]) and up to 32 revolutions for high angles of attack. For the unblown wing cases, a $k-\omega$ SST-RC-QCR2000 turbulence model was applied along with dual time-stepping with 20 subiterations and a physical time step size ten times larger than that of the blown cases, which is approximately 1.5% of wing chord per step. DDES was leveraged to capture stalled wake behavior at high angles of attack in both the blown and unblown configurations.

The methods discussed here and the results that follow should not be considered the definitive solution for this configuration. Rather, the methods are a significant progression of the numerical simulations intended to capture blown-section wing stall and post-stall effects. Time-accurate stall behavior presents a significant challenge in simulation. Although best practices and multiple successes have been established [24, 25, 26, 27, 28], combining wing stall with rotating propeller blades is generally discouraged for exploratory CFD. Without a large number of HPC processors (between 500 to 1000 for highly complex simulations), the disproportionate physical time scales and highly complex flow interactions between the propeller and wing flow fields require hundreds of hours to simulate to convergence. As an illustration, a conventional propeller rotating at 2,500 RPM attached to a wing flying at 30 m/s with 1° of propeller rotation per time step results in a step size that is nearly half of that necessary to resolve eddy behaviors on the wing. In other words, nearly twice as many steps are required for the flow to propagate the same distance downstream as a solution with time steps set to capture wing trailing edge eddies alone.

In the case of the X-57 HLP where propeller speeds commonly exceed 4000 RPM, the problem of disproportionate time scales is more extreme than conventional aircraft. For example, at the 4550 RPM condition with 1° rotation per time step, the simulation would require 27,300 steps for one second of real time, or, considered another way, the first 8 revolutions happen in about 0.1 seconds of real time. Meanwhile, the unblown flow over the wing at 29.9 m/s (58 KTAS) has progressed only 3.13 m, about 1 wingspan or 4.4 chord lengths, in the same 0.1 seconds. At low angles of attack, this progression is sufficient for the flow to become steady and may be used for engineering estimates of performance and behavior. However, higher angles of attack close to and beyond stall require greater flow propagation and computation time to capture the oscillatory flow behavior caused by separation bubbles forming then collapsing and shedding trailing edge vortices. This amount of flow propagation is considerably longer than that required to “start” the initial flow solution. Unfortunately, there was inadequate calendar time available to conduct dynamic post-stall simulations of either configuration on the available HPC resources, so compromises were made, such as limiting the angle of attack range to capture relatively mature flow solutions.

For the unblown wing, the simulations were started at 10° angle of attack, which is well below the stall angle of attack, and then the angle was incrementally increased, restarting from each previous run, until a sudden drop in the lift force history presented in the solution indicating stall. At angles of attack of 18° or greater, the solver was restarted at the same angle at least twice to determine if any stall onset behavior presented in the solution history before increasing the angle and starting the next run. For example, 25,000 steps were run for the 22° case (5 restarts at 5,000 steps each) compared to 6,000 steps for the 18° case (2 restarts at 3,000 steps each).

The blown wing cases were run at 4550 RPM (i.e., physical time step size, $DTPHYS = 0.003586$). Each angle of attack for the blown wing configuration required at least 16 propeller revolutions to exhibit steady, periodic trends, and, in some cases, considerably more revolutions were required to establish a relatively periodic force and moment history. Because an eight-revolution simulation took approximately 50 to 60 hours of wall clock time on the available HPC resources, the decision was made to submit each case to the HPC queue at once rather than completing one case before starting the next. Running simultaneous cases in this way reduced the wall clock time needed to run a limited drag polar near the stall angle of attack. Cases from 10° to 30° in 5° increments were submitted to roughly determine if the wing was stalled at a particular angle. Although starting a flow solution at high angles of attack tends to produce errors in the solution that may only be damped by long simulation times, if at all, the indicators of a stalled wing will generally present in the solution regardless. At angles greater than or equal to 20° , where stall onset was expected, the simulations were run to 32 propeller revolutions (11,520 steps) such that the unblown flow over the outboard wing sections would have progressed approximately 4 wingspans. The unblown simulations required less computing time to achieve sufficient flow progression than the blown wing cases because the physical time step could be increased to approximately 10 times larger than the blown wing step size.

3. OVERFLOW Performance and Convergence

The subcluster used for the OVERFLOW simulations consists of dual socket 20 core 2.40 GHz Intel Gold 6148 Skylake nodes with 96 GB RAM per node. Based on mid-range HPC common practice for the K-Cluster, a ratio of 250,000 vertices per processor was used as guidance to select 120 processors for the unblown wing cases and 160 processors for the blown wing cases. With these resources, an unblown wing case would typically complete 5000 steps in about 40 hours, whereas the blown wing case would complete 2880 steps (8 propeller revolutions) in 50 to 60 hours.

At the onset of these analyses, attempts were made to leverage the improved Successive Symmetric Over Relaxation (SSOR) scheme (ILHS = 26) introduced to OVERFLOW in 2020 [29]. However, multiple failures to start the solution resulted in reverting to the original SSOR algorithm (ILHS = 6). Discussion of this issue with OVERFLOW developers led to the conclusion that the difference in freestream and boundary conditions at the start of the simulation, while also splitting the grid system for parallel computing, was sufficient to cause a destabilizing effect and halt the solution progression. Limiting the initialization of the simulation to linear progression using two left-hand side SSOR subiterations (ILHSIT = 2) and the original SSOR algorithm for the first several hundred steps produced a relatively resolved flow solution. This solution was used to restart subsequent analyses with the improved SSOR scheme and ten SSOR subiterations (ILHSIT = 10). Unfortunately, this was not discovered until after most of the cases had completed, but it is recommended for future analyses. Although the simulations for this work were already complete at the time, the authors wish to note that an update to the improved SSOR scheme, dubbed “parallel SSOR or PSSOR,” was published in 2021 in which the destabilizing effects of splitting grids for parallel computing were greatly alleviated [30].

The convergence behavior of both the unblown and blown wing cases was acceptable for this study but could be improved. In the unblown cases, particularly near and following stall onset, the Log of the residuals (L2 Norm RHS) hovered near -4.5 as shown in Fig. 6 but with only a -0.23 average drop in the residual Log per subiteration. The blown wing cases, however, averaged only around -3.75 in the residual Log but showed a -0.93 average drop in the residual Log per subiteration at 15° to 25°. The relatively small drop per subiteration for both cases can be attributed to starting flow simulations at a high angle of attack without first initializing the flow solution downstream, a method identified after the simulations were complete. For the unblown cases, the improved SSOR scheme was leveraged near- and post-stall to improve the speed and convergence. At the time, it was incorrectly believed that the solution would be sufficiently unstable over time due to oscillatory separation to prohibit increasing the number of SSOR subiterations, which unfortunately negated any potential gains from using the improved scheme. Subsequent to the simulations in this paper, updated tests of the unblown wing geometry with the improved SSOR scheme and ten subiterations showed an improvement in convergence and simulation runtime.

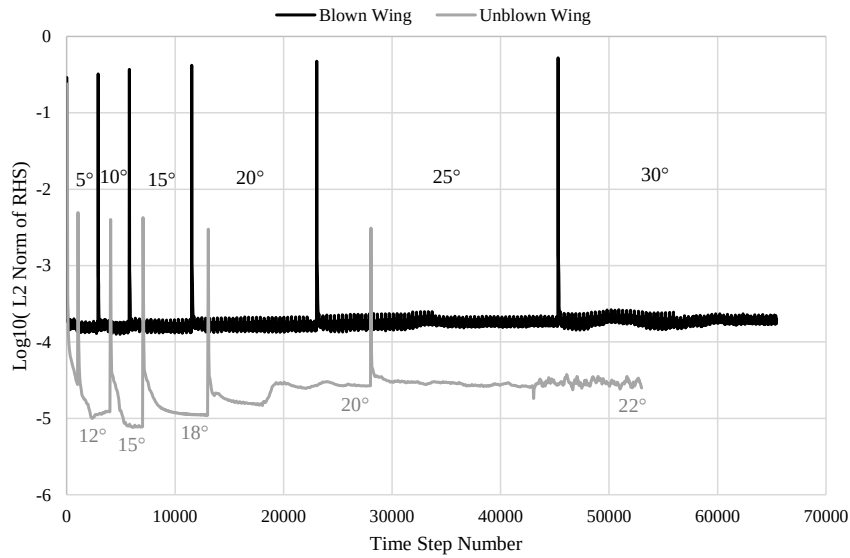


Fig. 6 Residuals for the blown and unblown wing simulations where peaks indicate a change in angle of attack. Angles of attack are labeled near corresponding time step ranges for both configurations.

C. FlightStream

1. Introduction

The mid-fidelity flow solutions for this effort were performed using the FlightStream viscous surface-vorticity flow solver. As part of prior NASA SBIR 2018-20 funded activities, the Research in Flight company has developed the FlightStream flow solver to make use of surface vorticity on an unstructured surface mesh to predict loads with attached flow [4, 31]. Aerodynamic loads are computed by shedding vorticity from the geometry. The solver utilizes the Fast Multipole Method (FMM) [4]. The general implementation of FMM in any problem revolves around the coalescing of far-field potential inviscid source/doublet bodies into a single body for evaluation of the net effect on a near-field body [4]. The algorithm evaluates the inductive effect of near-field bodies identically to the $O(N^2)$ implementation, where N is the number of mesh surface faces, but converts the large number of far-field bodies into a handful of clustered bodies inducing an effect based on the sum total of their field strengths. In FlightStream, FMM implementation coalesces far-field vorticity into localized three-dimensional point doublets encompassing the net vorticity strengths of all the coalesced far-field vorticity panels. Once the FMM near-field and far-field thresholds have been established, a spatial tree is constructed around the three-dimensional geometry to allow for the demarcation of these thresholds and to identify the near-field neighbors. The ideal spatial-tree for such an implementation is a spatial-octree. FlightStream already uses an octree-based Cartesian mesh for propagating the wake strands. A similar implementation holds for the viscous formulation of surface vorticity. This allows extremely fast near-field sorting of points closest to, or inside of, a viscous flow region.

Two models for the boundary layer have been implemented: laminar and turbulent. A boundary layer transition model has also been implemented. These models are two-dimensional methods implemented along the on-body surface streamlines. These integral methods are computationally efficient and can be applied to any general three-dimensional wall boundary with the exception of regions involving three-dimensional crossflows. The laminar boundary layer analysis method used in FlightStream is the standard two-parameter model of the Thwaites [4] integral method with the momentum integral equation. An integral boundary layer model for compressible turbulent flows has been implemented into the inviscid flow solver [4]. The turbulent boundary layer model implemented has numerical improvements and optimizations over that of the original model developed by Standen, but philosophically follows the work of Standen quite closely, with the application to subsonic, turbulent, compressible flows along on-body streamlines. The method is essentially two-dimensional (similar to the laminar model described previously) but is implemented in a quasi-three-dimensional manner inside FlightStream. For the fluid properties outside the boundary layer, the flow is assumed to be isentropic and subsonic, but compressible. Solving these equations numerically (using 4th order Runge-Kutta methods) along the on-body streamline generates all relevant turbulent boundary layer data. This data is then used to compute the boundary layer thicknesses.

With the addition of advanced viscous models, additional computational enhancements have been added to the solver to maintain an $O(N \log(N))$ scalability to the performance of the boundary layer algorithms. To this end, spatial octree algorithms that mimic the existing Fast Multipole Method implementation (for the inviscid solver) have been implemented for the viscous flow near-wall regions. This allows extremely fast near-field sorting of points closest to, or inside of, a viscous flow region. This spatial tree increases the memory footprint of the solver by roughly 5% for all cases. However, the performance benefits are substantial with an otherwise $O(N \times M)$ algorithm (where M is a very large number equal to solver surface mesh faces) being reduced to $O(N \log(N))$.

A robust model for computing attachment lines and critical points on the inviscid surface velocity fields generated in the solver was added. This is an enabling requirement for robust coupling with the viscous boundary layer models described previously. The laminar and turbulent boundary layer models were integrated with the inviscid solver via displacement of the inviscid flow boundary equal to the displacement thicknesses of the local boundary layers. For boundary layers, the displacement thickness indicates the extent to which the surface would have to be displaced in order to match the flow rate of the viscous flow, but with an inviscid velocity profile. Therefore, the displacement thickness is suited for the measure of displacement needed for the inviscid flow boundary. The inviscid boundary displacement is simulated by blowing a velocity normal to the mesh face. The magnitude of the blowing velocity can be computed by using the momentum flux equations. This transpiration velocity can be computed along the on-body streamlines for each mesh face using standard finite-difference numerical marching schemes. The transpiration (blowing) velocity is then used to modify the inviscid Neumann boundary condition [4].

2. Grid setup

The FlightStream solver requires a surface grid only and this grid was generated using the surface gridding tools available within FlightStream. Primarily, the aligned meshing tool was used to generate a structured, anisotropic quad surface grid (Fig. 7) using directional refinements in the chordwise and spanwise directions, according to the best practices established by Ahuja et al. [4]. Anisotropic ratios are defined as the mid-chord spanwise-to-chordwise

parameter, and values between 2.5 and 3.5 were chosen for both the blown and unblown components. The blown section of the wing was provided a spanwise refinement with anisotropy ratios of 1.0 for $1.5R$ spanwise distance on either side of the propeller centerline. The blades were modeled using the solid body representation similar to the OVERFLOW setup described previously. Surface growth ratios were typically limited to 1.1 or less for all surface grids. A breakdown of the grid parameters for both the blown and unblown geometries is provided in Table 4.

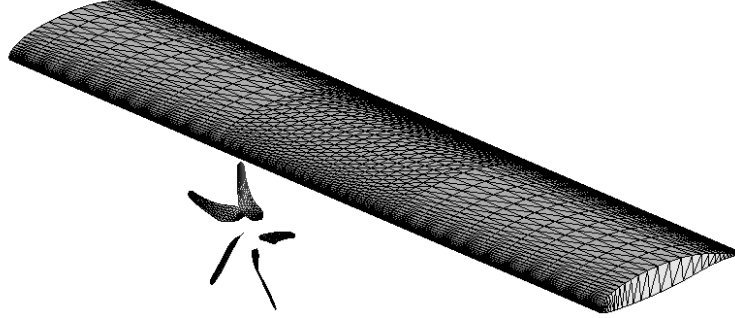


Fig. 7 FlightStream surface grids for blown wing configuration.

Table 4 FlightStream Grid Parameters for Blown and Unblown Geometries

Parameter	Wing	Blade
Chordwise Points	82	30
Spanwise Points	40	25
Surface directional growth rate	1.1 (Dual-sided)	1.1 (Dual-sided)
Parameter	Value	
Unblown Total Face Count	6,600	
Blown Total Face Count	8,100	

3. FlightStream Performance and Convergence

The FlightStream unsteady solver was run on a quad-core CPU with the Intel Core i5 processors, utilizing 4 cores of the solution. Each unsteady solver run was performed out to 50 time-steps of 0.001 seconds each. The solution run time was measured to be roughly 10 minutes per angle of attack for both the blown and unblown wing scenarios. Solver convergence was driven to 10^{-5} for aerodynamic loads and moments per time step. Viscous coupling was initiated for these unsteady solutions. Given the mid-fidelity range of the solutions provided by FlightStream, this was considered a reasonable balance of solution fidelity and computational expense.

IV. Results

The results from simulations of the unblown wing, the isolated propeller, and the propeller-blown wing are covered in the following sections. In each case, FlightStream is compared to OVERFLOW over a range of angle of attack. The predictions obtained from both tools were normalized into coefficients (except where highlighted) as described here:

$$C_F = \frac{Force}{\frac{1}{2}\rho S_{ref} V_{inf}^2}, C_M = \frac{Moment}{\frac{1}{2}\rho S_{ref} V_{inf}^2 c}$$

$$\rho = 1.225 \text{ kg/m}^3, S_{ref} = 2.1767 \text{ m}^2, V_{inf} = 29.9 \text{ m/s}, c = 0.7141 \text{ m}$$

A. Unblown Wing

Initial tests were performed on the unblown wing in isolation (no propeller). Predictions are summarized in Fig. 8—Fig. 10 in which the OVERFLOW points are the result of averaging the last 1000 steps per case, except for post stall at 22° where 5000 steps are averaged to capture the data spread (Fig. 10). Nonlinear lift forces at low flow incidence angles are apparent in Fig. 8a. This is not unusual for an airfoil derived from the X-57 Maxwell aircraft with a large amount of natural laminar flow. OVERFLOW predicts stall at 20° to 22° angle of attack whereas FlightStream

is delayed in predicting stall at 25° . Capturing accurate transition effects with URANS simulations is difficult, and most such simulations tend to lean more towards fully turbulent conditions than the true natural transition condition. FlightStream utilizes an empirical model to capture transition locations; it is probable that FlightStream predicts a higher region of laminar flow as a result of this model relative to OVERFLOW and, consequently, underpredicts the viscous boundary layer effect. Drag predictions (Fig. 8b) are generally in good agreement up to 22° angle of attack, at which point substantial turbulent flow separation effects are dominant in the OVERFLOW and FlightStream simulations.

The pitching moment data from the two tools are summarized in Fig. 8c. Both tools are consistent in the range of 12° to 18° and the C_M curve slope is consistently captured, despite its nonlinearity. Beyond 20° , both FlightStream and OVERFLOW predict the change in sign for the slope of the C_M data, but FlightStream predicts a shallower slope. This is expected given the likely under-captured viscous coupling in FlightStream relative to OVERFLOW.

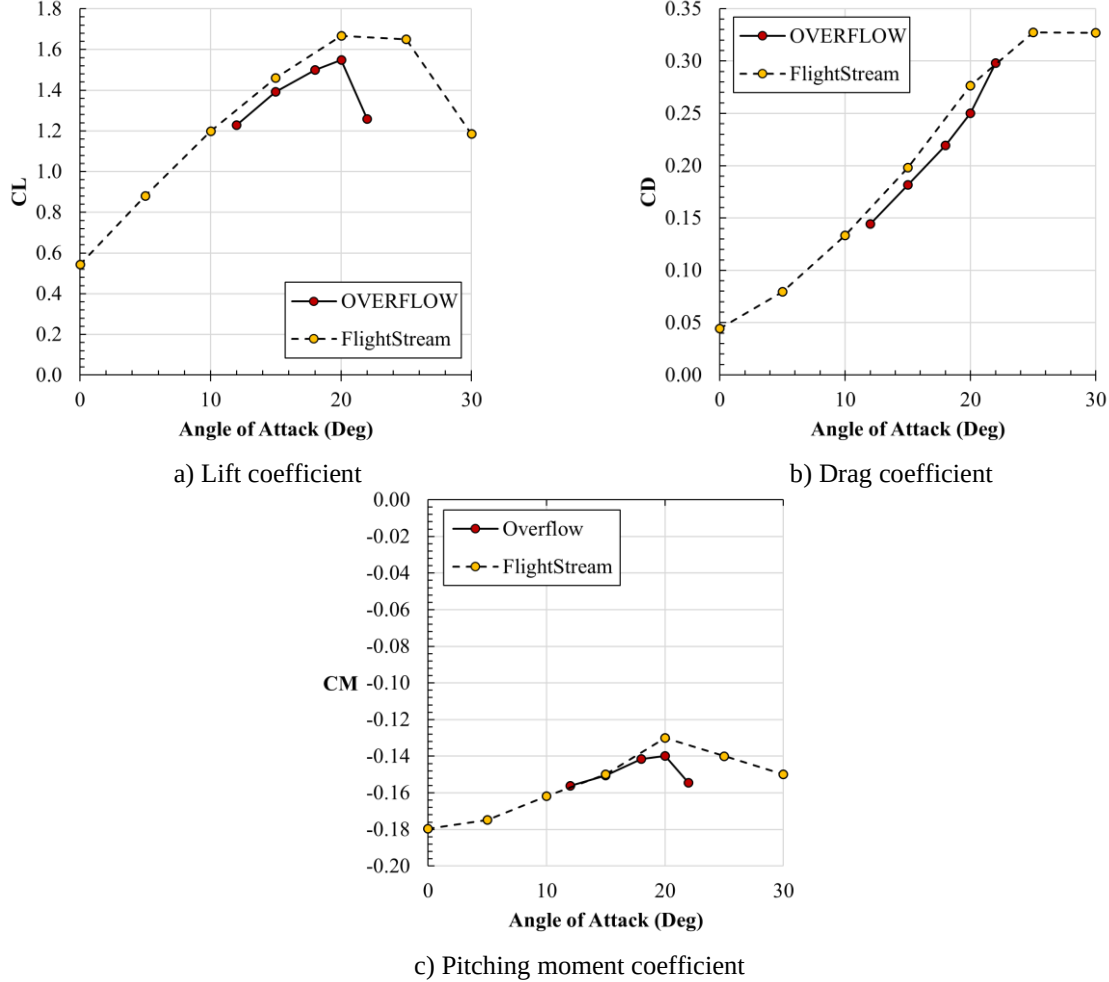


Fig. 8 Lift, drag, and moment coefficients for the unblown wing.

CF_x , CF_y , and CF_z results are shown in Fig. 9. CF_x results are consistently captured for the linear region and the CF_z results are very similar to the lift results previously discussed. It is noted that the $CF_y = 0$ condition applies for the symmetric airflow over the wing, and FlightStream and OVERFLOW both capture this prior to stall. At stall, there is a very slight asymmetric load predicted in the OVERFLOW solution that is suspected to be a volumetric behavior of the unsteady, spanwise movement of the detached flow as the flow attempts to reattach immediately after stall onset. The solution history for CF_y , illustrating the onset of oscillatory behavior, is provided in Fig. 10 where the increasing angle of attack is labeled along the top, corresponding to boundaries that illustrate the number of averaged time steps to produce mean forces and pitching moment for that case. The spanwise differential pressure across the wing likely leads to asymmetric wingtip vortex strengths, which impart a net side-load on the squared wingtip surfaces as well as a strong oscillating rolling moment. Without a separating barrier, such as a fuselage, between the left- and right-wing

sections, this separation bubble can pass unimpeded spanwise across the wing. However, it is important to keep in mind that these forces are minimal relative to the lift and drag: the largest CF_Y at 22° was comparatively only 0.2% of the drag or -0.76 N. Interestingly, FlightStream predicts a small artifact of this asymmetric behavior at 25° , which may be indicative of similar unsteadiness to that of OVERFLOW at high angles of attack. Additional investigation of the time histories of both tools is ongoing for this condition.

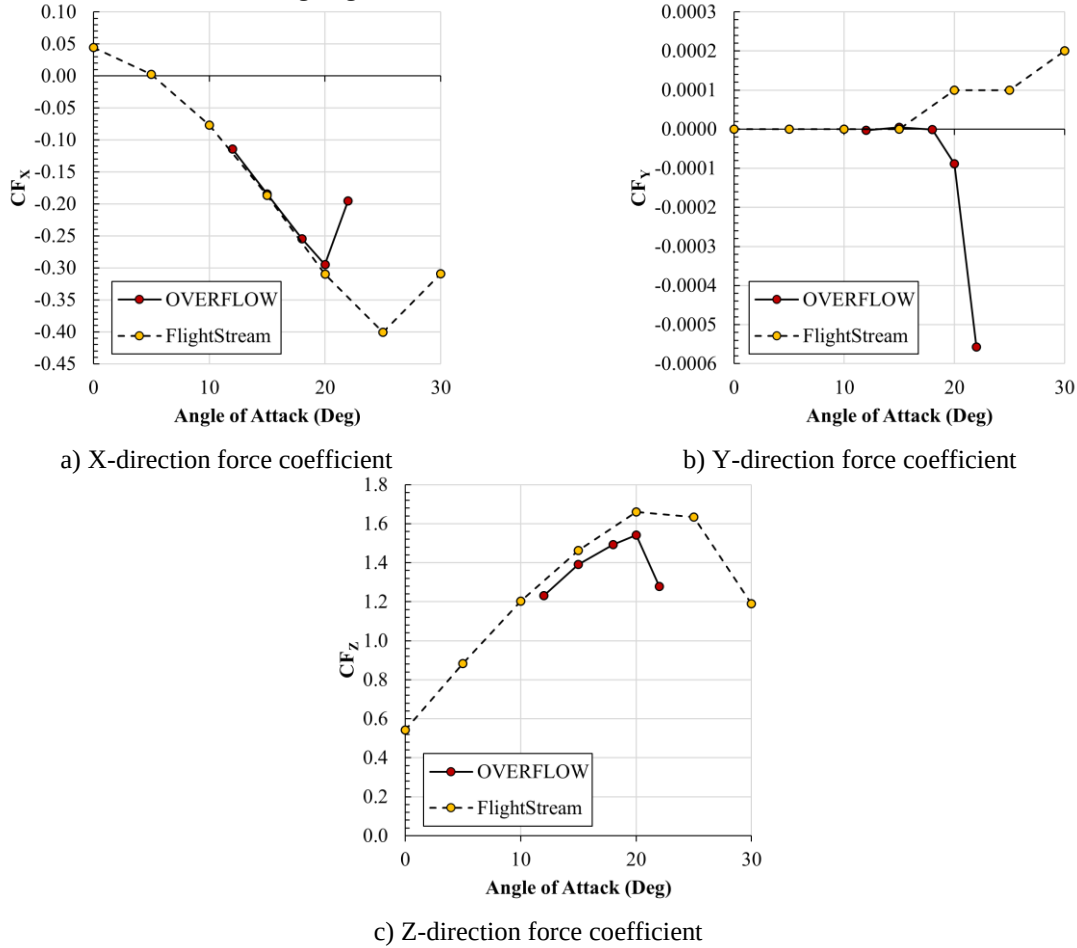


Fig. 9 Aerodynamic X-, Y-, and Z-component of force coefficients for the unblown wing.

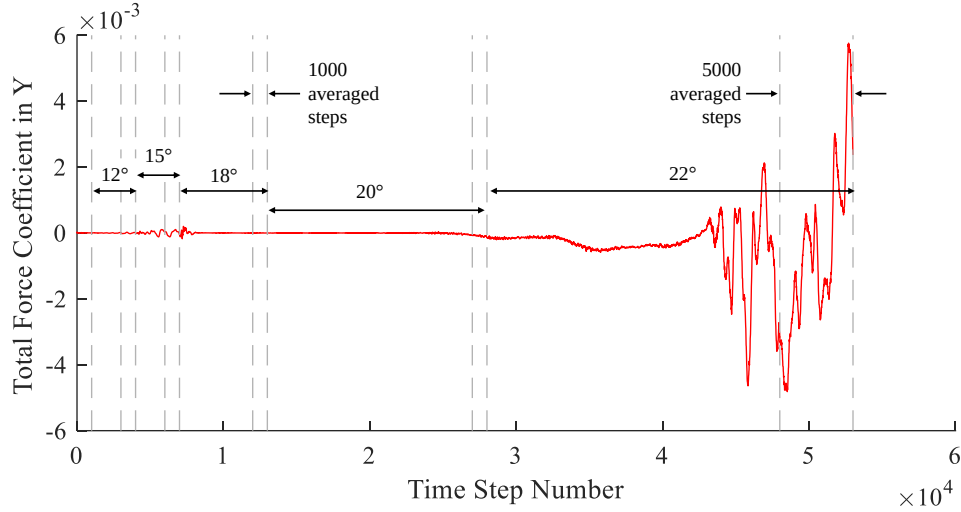


Fig. 10 Unblown CF_y history with regions illustrating averaged steps at angles of attack.

B. Installed Propeller

The next analyses involved the blown condition with the installed propeller turning in both the OVERFLOW and FlightStream unsteady flow solvers. The airspeed was maintained at 29.9 m/s (58 KTAS) and the operating RPM of the propeller was maintained at 4549 RPM for both tools. The predicted installed propeller performance from both tools for a range of angles of attack is summarized in Fig. 11. It is noted that both tools are generally in agreement with regard to the magnitude and trend of the thrust and torque curves. Relative to OVERFLOW, FlightStream tends to underpredict the thrust slightly at incidence angles below 10° and overpredicts it thereafter. FlightStream also consistently underpredicts the torque compared to OVERFLOW. The reason for this is identical to that for the overprediction of stall incidence in FlightStream: the boundary layer transition results from a free transition are likely underrepresenting the viscous effects in the FlightStream prediction relative to OVERFLOW.

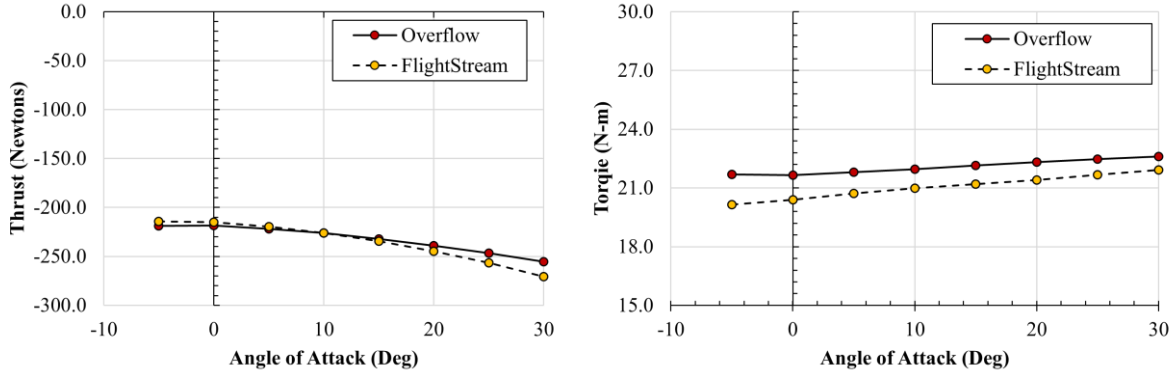


Fig. 11 FlightStream & OVERFLOW thrust and torque predictions for the installed propeller.

C. Blown Wing

The results obtained from OVERFLOW and FlightStream for the blown wing are shown in Fig. 12 and Fig. 13. Lift predictions are generally in agreement, though FlightStream generates an overprediction compared to OVERFLOW in lift force, up to 15° angle of attack, on account of underpredicting the viscous coupling. Of note is the approximate prediction of $C_{L_{max}} = 1.78$ by FlightStream compared to OVERFLOW's prediction of $C_{L_{max}} = 1.94$ at a stall angle of 20°. Note also that both OVERFLOW and FlightStream capture similar C_L vs. α curve slopes in the linear regime of the flow. Trends in the post-stall lift force are similar between both tools except for the highest angle of attack, which shows some divergence. Similarly, drag predictions were found to be in good agreement up to 10° flow incidence, at which point substantial flow separation effects are dominant in the OVERFLOW simulation and likely underrepresented in FlightStream.

Pitching moment results in Fig. 12c shows close agreement between the two methods up to the onset of stall. Drag predictions are also consistent up to 25° for both methods. Post stall, FlightStream drag predictions are measurably lower than OVERFLOW, which contributes to the difference in slope of the C_M vs. α curve after stall. The consistency between the two methods for the linear regime makes for useful conclusions regarding the early design stability & control requirements but illustrates the difficulty in capturing post-stall behaviors, particularly with low- to mid-fidelity potential flow solvers. Similar consistency is observed for the force coefficients in Fig. 13.

As with the wing-only case, asymmetric flow separation presents after stall in the unsteady OVERFLOW simulations shown in Fig. 13b, but the flow behavior exhibits a significant difference from the unblown scenario. Just after stall, rather than repeatedly moving spanwise across the wing as in the unblown case, the flow seems encouraged to remain relatively attached on the right side of the wing where the propeller swirl is washing down onto the wing. This is reflected in the Mach contours shown in Fig. 14 for an angle of attack of 25° , coloring a planar slice 0.457 m aft of the propeller, near the wing quarter-chord, where the flow is detached and decelerated (or even reversed) on the propeller upwash side (right side of Fig. 14). Examining flow vortex structures in Fig. 15 for Q Criterion = 0.01 (the ratio of vorticity to viscous stresses), the higher pressure on the left side of the wing (right side of the figure) appears to be forcing the propeller wake toward the right wingtip (left side of the figure), increasing the asymmetric load. However, whether the magnitude of this imbalance is representative of actual flight or the simulation requires additional runtime to detach the flow on the right side of the wing was inconclusive at the completion of this study. Like the unblown wing case, the load is relatively small compared to the lift and drag forces where the peak CF_Y represents only 20 N of side load at 25° compared to 2400 N of lift. FlightStream also shows a small asymmetric load in CF_Y , which is still being investigated.

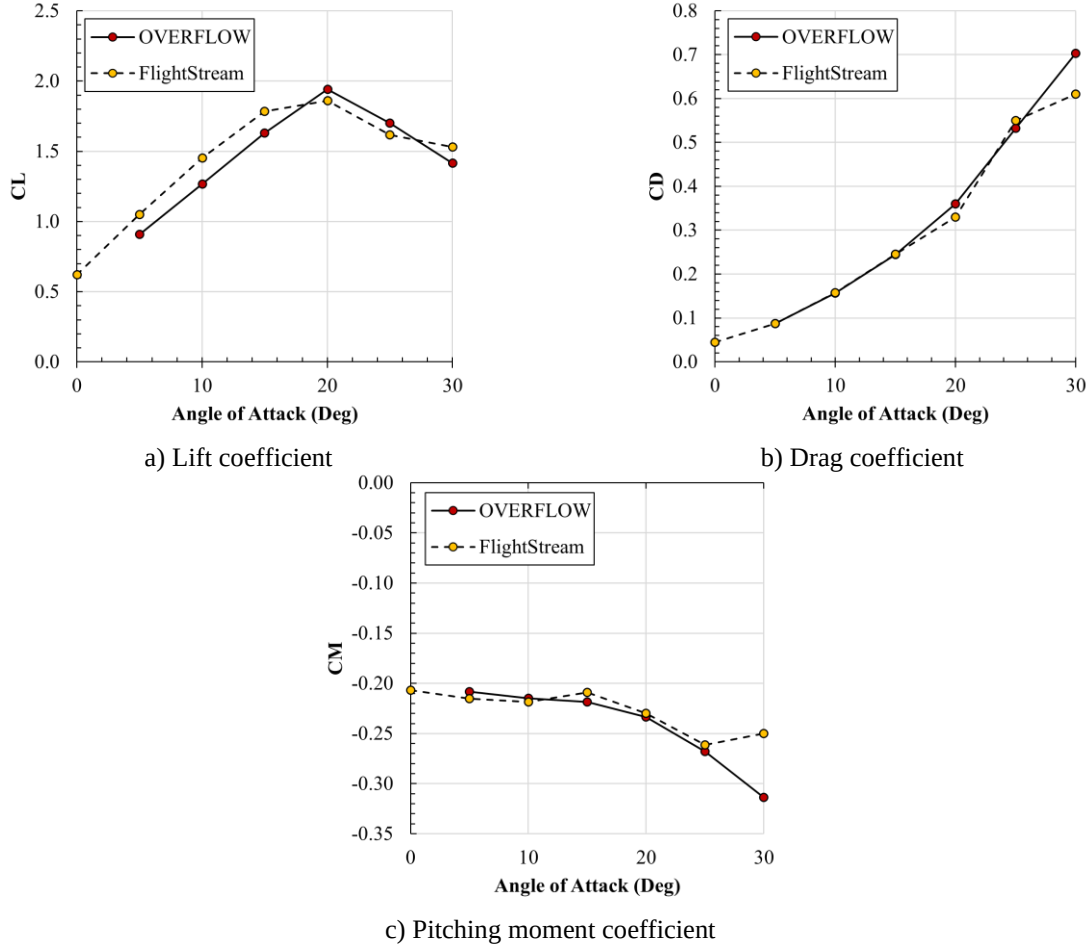
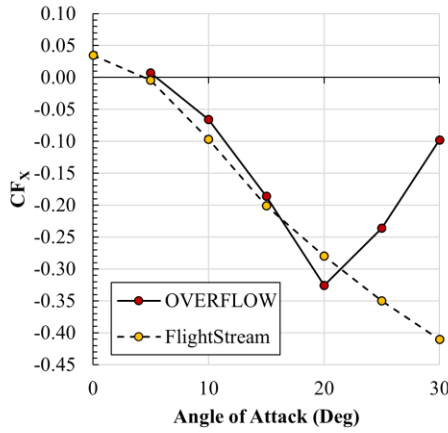
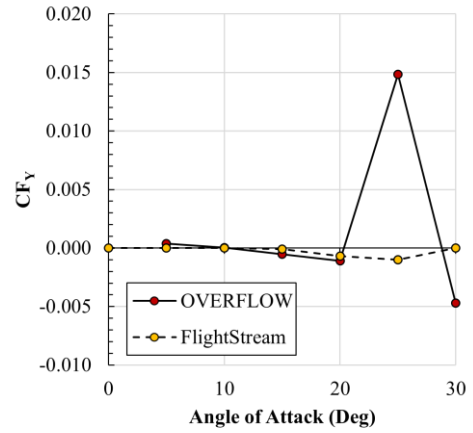


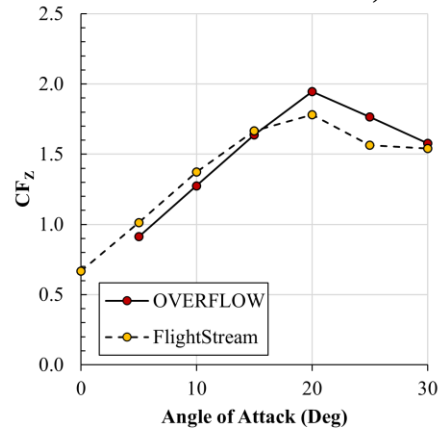
Fig. 12 FlightStream & OVERFLOW aerodynamic lift and drag forces for the blown wing.



a) X-direction force coefficient



b) Y-direction force coefficient



c) Z-direction force coefficient

Fig. 13 FlightStream & OVERFLOW aerodynamic X-, Y-, and Z-component of force coefficients for the blown wing.

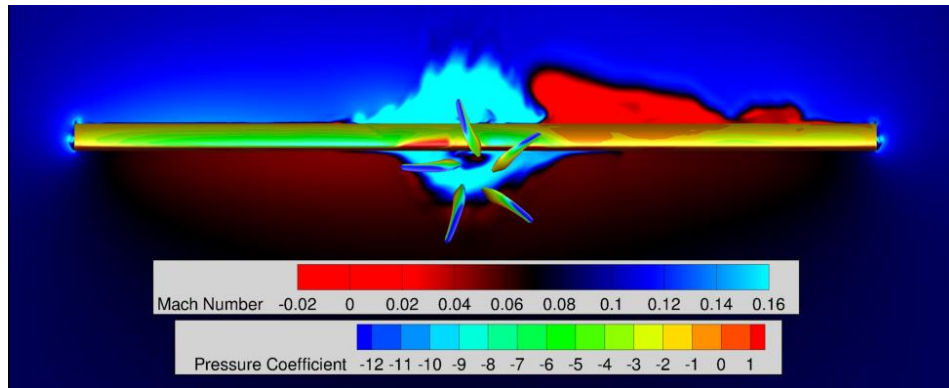


Fig. 14 Blown wing asymmetric stall at 25° angle of attack as predicted by OVERFLOW. Planar slice 0.457 m aft of the propeller showing Mach number contours over the wing near the wing quarter chord. The wing and propeller surfaces are colored by pressure coefficient at the last time step.

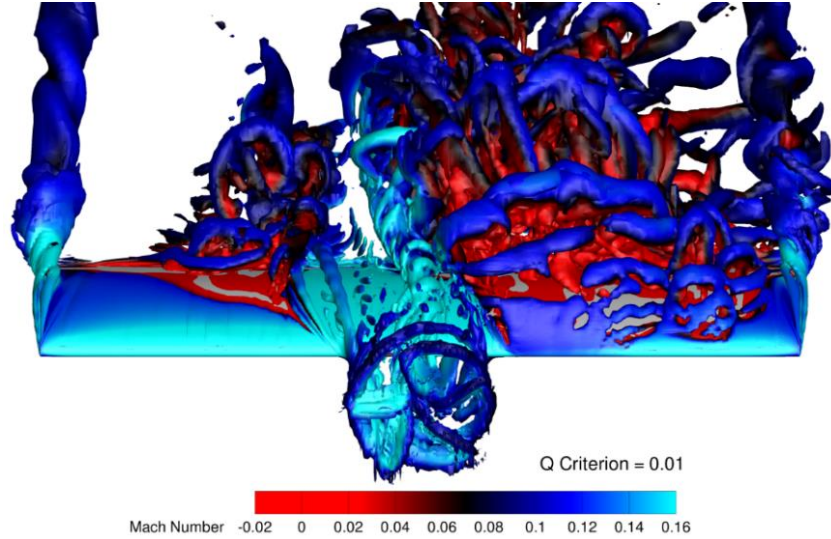


Fig. 15 Blown wing asymmetric stall at 25° angle of attack as predicted by OVERFLOW showing Q Criterion contours at 0.01 colored by Mach number. Last time step shown (32 propeller turns complete).

V. Conclusions and Future Work

This work described an ongoing effort to compare OVERFLOW and FlightStream simulation results for unblown and propeller-blown wing configurations at high angles of attack. Overall, the predictions of OVERFLOW and FlightStream agreed for angles of attack up to and approaching stall for the unblown and blown wing cases. Post-stall, blown wing lift continued to agree relatively well, but FlightStream was observed to underestimate drag forces and pitching moments compared to OVERFLOW due to a relative underrepresentation of the turbulent separation effects. Furthermore, FlightStream predicted a higher stall angle of attack for the unblown wing compared to OVERFLOW simulations. Additional corrections to the empirical model in FlightStream may be required to model the effects of turbulent separation post-stall. However, for the purposes of early design analysis, the rapid setup and short execution times that FlightStream enables should not be discounted. Although OVERFLOW may provide a more accurate representation of the volumetric flow behavior, particularly post-stall, the time and effort to set up and run such cases is orders of magnitude larger than in FlightStream. Therefore, FlightStream may be very useful for analyzing early designs of blown-wing aircraft, such as tiltwings and DEP lift-augmented configurations, enabling many cases to be run on several design variants in relatively short order.

During this work, lessons were learned in the implementation of improved methods and best-practices for OVERFLOW simulations of highly complex propeller-blown wing aerodynamics. In situations where performing dynamic deep stall simulations of blown wings is infeasible with available computing resources or time, a buildup approach toward smaller time steps is recommended. Starting the simulation at larger time scales enables the flow to propagate farther downstream while subsequent reduced time stepping better resolves the propeller wakes. The size of these steps should be investigated for each case individually to determine an appropriate balance of speed vs accuracy. Also, although the new parallel SSOR scheme alleviates many of the issues resulting from splitting grids for parallel computing, some simulations may still benefit from initialization using linear progression with two left-hand side SSOR subiterations ($ILHSIT = 2$) for the first few hundred steps then restarting with greater subiteration to improve speed and convergence.

This paper is a summary of ongoing research comparing low-to-mid fidelity tools to higher fidelity numerical simulations. Future work is planned to focus on the comparison of flow data, in the form of surface pressure and velocity, a time history comparison of aerodynamic loads and moments, and the comparison of predicted flow separation characteristics in both tools. The OVERFLOW data are planned to be expanded to include a closer resolution of the near-stall incidence angle regimes, a broader range of angle of attack both pre- and post-stall, and a closer investigation of some of the observed separated flow characteristics. Finally, investigations of various methods for analyzing propeller-blown wings in OVERFLOW are planned to achieve improved computational efficiency and convergence of results.

Acknowledgments

The authors wish to acknowledge the cooperation and support of the NASA SBIR Office for this effort. The primary developmental work for FlightStream for this paper was performed within the context of the NASA SBIR NNX17CL12C Phase II and Phase II Extended contract activities and the NASA NRA Transformative Tools and Technologies Contract 80LARC19C0013. The OVERFLOW aspects of this work were supported by the NASA Langley Research Center Innovation Fund for Internal Research and Development, the Flight Demonstrations and Capabilities project's X-57 Maxwell subproject, the Transformational Tools and Technologies Project, and the Revolutionary Vertical Lift Technology Project. The authors also wish to express gratitude to the individuals that provided guidance and feedback supporting the OVERFLOW aspects, including Joseph Derlaga, Pieter Buning, Sally Viken, Karen Deere, Neal Chaderjian, and Patricia Ventura-Diaz, and to Siena Whiteside for coordination of this activity.

References

- [1] N. K. Borer, M. D. Patterson, J. K. Viken, M. D. Moore, J. Bevirt, A. M. Stoll and A. R. Gibson, "Design and Performance of the NASA SCEPTOR Distributed Electric Propulsion Flight Demonstrator," in *16th AIAA Aviation Technology, Integration, and Operations Conference*, American Institute of Aeronautics and Astronautics, AIAA 2016-3920.
- [2] National Aeronautics and Space Administration, "UAM Reference Vehicles - SACD," [Online]. Available: <https://sacd.larc.nasa.gov/uam-refs/>. [Accessed December 2021].
- [3] R. H. Nichols and P. G. Buning, "User's Manual for OVERFLOW 2.3, v.2.3d," February 2021. [Online]. Available: <https://overflow.larc.nasa.gov/users-manual-for-overflow-2-3/>. [Accessed 21 March 2022].
- [4] V. Ahuja, J. Burkhalter and R. J. Hartfield, Robust prediction of high lift using surface vorticity, Austin: NASA SBIR NNX17CL12C, Phase II final report, 2017.
- [5] C. N. Sheridan, D. D. Pham and S. Whiteside, "Evaluation of VSPAERO Analysis Capabilities for Conceptual Design of Aircraft with Propeller-Blown Wings," in *AIAA Aviation 2021 Forum*, American Institute of Aeronautics and Astronautics, AIAA 2021-2510.
- [6] B. L. Litherland and J. M. Derlaga, "A Performance Analysis of Folding Conformal Propeller Blade Designs," in *AIAA Aviation 2019 Forum*, American Institute of Aeronautics and Astronautics, AIAA 2019-3676.
- [7] B. L. Litherland, N. K. Borer and N. S. Zawodny, "X-57 Maxwell High-Lift Propeller Testing and Model Development," in *AIAA Aviation 2021 Forum*, American Institute of Aeronautics and Astronautics, AIAA 2021-3193.
- [8] J. K. Viken, S. A. Viken, K. A. Deere and M. B. Carter, "Design of the Cruise and Flap Airfoil for the X-57 Maxwell Distributed Electric Propulsion Aircraft," in *AIAA Aviation 2017 Forum*, American Institute of Aeronautics and Astronautics, AIAA 2017-3922.
- [9] M. D. Patterson, N. K. Borer and B. German, "A Simple Method for High-Lift Propeller Conceptual Design," in *54th AIAA Aerospace Sciences Meeting*, American Institute of Aeronautics and Astronautics, AIAA 2016-0770.
- [10] B. L. Litherland, M. D. Patterson, J. M. Derlaga and N. K. Borer, "A Method for Designing Conforming Folding Propellers," in *AIAA Aviation 2017 Forum*, American Institute of Aeronautics and Astronautics, AIAA 2017-3781.
- [11] "OpenVSP," [Online]. Available: <http://openvsp.org/>. [Accessed May 2020].
- [12] R. A. McDonald and J. R. Gloudemans, "Open Vehicle Sketch Pad: An Open Source Parametric Geometry and Analysis Tool for Conceptual Aircraft Design," in *AIAA SciTech 2022 Forum*, American Institute of Aeronautics and Astronautics, AIAA-2022-0004.
- [13] S. K. Whiteside, B. L. Litherland and X. Fei, "VSP Hangar | Rectangular Wing with High-Lift Propeller," [Online]. Available: <http://hangar.openvsp.org/vspfiles/526>. [Accessed 11 Feb. 2022].
- [14] W. M. Chan, R. J. Gomez III, S. E. Rogers and P. G. Buning, "Best Practices in Overset Grid Generation," in *32nd AIAA Fluid Dynamics Conference*, American Institute of Aeronautics and Astronautics, AIAA 2002-3191.
- [15] W. M. Chan, S. A. Pandya, S. E. Rogers, J. C. Jensen, H. C. Lee, D. L. Kao, P. G. Buning, R. L. Meakin, D. A. Boger and S. M. Nash, "Chimera Grid Tools User's Manual, v.2.2," June 2018. [Online]. Available: <https://www.nas.nasa.gov/publications/software/docs/chimera/index.html>. [Accessed 11 Feb 2022].
- [16] R. Nichols, R. Tramel and P. Buning, "Solver and Turbulence Model Upgrades to OVERFLOW 2 for Unsteady and High-Speed Applications," in *24th AIAA Applied Aerodynamics Conference, Fluid Dynamics and Co-located Conferences*, American Institute of Aeronautics and Astronautics, AIAA 2006-2824.
- [17] A. K. Henrick, T. D. Aslam and J. M. Powers, "Mapped Weighted Essentially Non-Oscillatory Schemes: Achieving Optimal Order near Critical Points," *Journal of Computational Physics*, vol. 207, pp. 542-567, 2005.
- [18] R. Tramel, R. Nichols and P. Buning, "Addition of Improved Shock-Capturing Schemes to OVERFLOW 2.1," in *19th AIAA Computational Fluid Dynamics*, American Institute of Aeronautics and Astronautics, AIAA 2009-3988.

- [19] F. R. Menter, "Two-equation eddy-viscosity turbulence models for engineering applications," *AIAA Journal*, vol. 32, pp. 1598-1605, 8 1994.
- [20] M. L. Shur, M. K. Strelets, A. K. Travin and P. R. Spalart, "Turbulence Modeling in Rotating and Curved Channels: Assessing the Spalart-Shur Correction," *AIAA Journal*, vol. 38, pp. 784-792, 2000.
- [21] R. Langtry and F. Menter, "Transition Modeling for General CFD Applications in Aeronautics," in *43rd AIAA Aerospace Sciences Meeting and Exhibit*, American Institute of Aeronautics and Astronautics, AIAA 2005-522.
- [22] R. Jain, "CFD Performance and Turbulence Transition Predictions on an Installed Model-scale Rotor in Hover," in *55th AIAA Aerospace Sciences Meeting*, American Institute of Aeronautics and Astronautics, AIAA 2017-1871.
- [23] R. Jain, "Hover Predictions on the S-76 Rotor with Tip Shape Variation Using Helios," *Journal of Aircraft*, vol. 55, pp. 66-77, 2018.
- [24] N. M. Chaderjian, "Navier-Stokes Simulation of UH-60A Rotor/Wake Interaction Using Adaptive Mesh Refinement," in *AHS International 73rd Annual Forum & Technology Display 2017*, ARC-E-DAA-TN40776.
- [25] N. M. Chaderjian, "Numerical Simulation of Dynamic Stall Using Near-Body Adaptive Mesh Refinement," in *Tenth International Conference on Computational Fluid Dynamics (ICCFD10) 2018*, ICCFD10-102, ARC-E-DAA-TN58516.
- [26] J. C. Duensing, S. Yoo, D. Maldonado, J. A. Housman, J. C. Jensen and C. C. Kiris, "Establishing Best Practices for X-57 Maxwell CFD Database Generation," in *AIAA SciTech 2019 Forum*, American Institute of Aeronautics and Astronautics, AIAA 2019-0274.
- [27] National Aeronautics and Space Administration, "4th AIAA CFD High Lift Prediction Workshop (HLPW-4)," March 2022. [Online]. Available: <https://hilftpw.larc.nasa.gov/>. [Accessed 08 April 2022].
- [28] J. G. Shum and S. Lee, "Computational Fluid Dynamics Best Practices for 2D Dynamic Stall Predictions," in *AIAA Aviation 2020 Forum*, American Institute of Aeronautics and Astronautics, AIAA 2020-2743.
- [29] J. M. Derlaga, C. W. Jackson and P. G. Buning, "Recent Progress in OVERFLOW Convergence Improvements," in *AIAA SciTech 2020 Forum*, American Institute of Aeronautics and Astronautics, AIAA 2020-1045.
- [30] J. M. Derlaga, C. W. Jackson and P. G. Buning, "A Parallel Symmetric Successive Overrelaxation Method for OVERFLOW," in *AIAA Aviation 2021 Forum*, <https://arc.aiaa.org/doi/pdf/10.2514/6.2021-2747>, AIAA-6.2021-2747.
- [31] V. Ahuja and R. Hartfield, "Aerodynamic Loads over Arbitrary Bodies by Method of Integrated Circulation," *Journal of Aircraft*, vol. 53, pp. 1719-1730, 2016.
- [32] K-Cluster Systems Team, "LaRC Center Cluster: Hardware Resources," NASA Langley Research Center, [Online]. Available: https://k-info.larc.nasa.gov/CCFresources_hardware.html. [Accessed 07 April 2022].