

Pre-Test CFD Analyses for The Tiltwing Test Rig Experiment

James C. Jensen*, Andrew M. Chuen^{†‡}, and William M. Chan[§].
NASA Ames Research Center, Moffett Field, CA 94035

In preparation for the Tiltwing Test Rig experiment at NASA Langley's 14- by 22-Foot Subsonic Tunnel, a series of CFD simulations are being run using OVERFLOW to better understand the physics involved. The first study focuses on determining the best actuator disk model to use for the proprotors. Using a model to approximate the proprotors significantly reduces convergence time and computational expense, offering a practical alternative to high-fidelity simulations. The two models being investigated are the Pressure Difference model that uses a uniform pressure jump and the Rotor Disk model which is based on blade-element theory. The Rotor Disk model proved to better approximate the flow physics, so it was used to perform three pre-test CFD studies. The first is a component build-up to better understand the effects of each geometric component on the aerodynamic performance. The second is a study of the aerodynamic effect of varying the proprotor turning direction, while the third study looks at the effects of changing the flap deflection angle.

I. Nomenclature

C_L	=	Lift coefficient	Re	=	Reynolds number
C_D	=	Drag coefficient	α	=	Angle of attack
C_M	=	Pitching moment coefficient	$ \Delta $	=	Difference between two quantities
C_p	=	Pressure coefficient	Δs_{max}	=	Max surface mesh spacing
M_∞	=	Freestream Mach number	np_{min}	=	Minimum number of points across a surface
V_∞	=	Freestream velocity	Δs_{wall}	=	Initial wall normal spacing
p_∞	=	Freestream pressure	y^+	=	Non-dimensionalized wall normal spacing
ρ_∞	=	Freestream density	C_T	=	Coefficient of Thrust
T_∞	=	Freestream temperature			

II. Introduction

NASA's Revolutionary Vertical Lift Technology Project (RVLT) is working to develop critical technologies that enable revolutionary new air travel options, such as passenger-carrying air taxis. One such concept vehicle is the Tiltwing[1]. This vehicle is designed as an air-taxi, with a payload of 1200 lbs or up to six passengers. It has enough endurance to complete two 37.5 nautical mile flights, with 20 minutes of reserve. The most unique feature of this aircraft is the tilting wing with six proprotors mounted that is capable of rotating 90°. Unlike a tilt-rotor aircraft, the entire wing is able to rotate as shown in Figure 1. The main benefit of this type of configuration is an improvement in both cruise speed and hover performance due to the ability of the wing and proprotors to be positioned optimally for both segments of flight.

To better understand the performance of the Tiltwing in cruise, hover, and in transition between the two, a wind tunnel test is being performed at NASA Langley's 14- by 22-Foot Subsonic Tunnel[2]. This tunnel has previously been used to run performance and acoustic testing of similar proprotor geometries[3]. The test article has a span of 78 inches, a constant chord of 18 inches, and rotors that are 2 feet in diameter. The wing also has a 30% fowler flap that runs along the entire span. This hardware is then mounted on top of a cylindrical splitter plate to minimize the impact of the boundary layer on the measured loads. The tunnel model with the splitter plate is referred to as the Tiltwing Test Rig (TWTR). The objective of the test is to create a large set of data that can be used to validate the numerical prediction of this type of geometry[4]. A non-comprehensive list of parameters that are being varied during the test to create this

*Research Aerospace Engineer, Computational Aerosciences Branch, M/S 258-2. Member AIAA.

[†]Pathways Student, Computational Aerosciences Branch, M/S 258-2. Student Member AIAA.

[‡]Graduate Student Researcher, Univ. of California, Davis, Department of Mechanical and Aerospace Engineering

[§]Computer Scientist, Computational Aerosciences Branch, M/S 258-2. Associate Fellow AIAA.

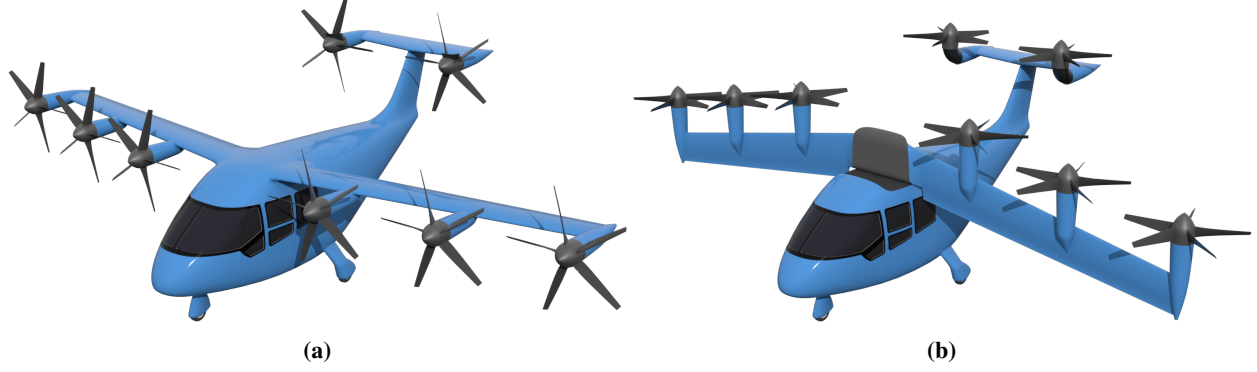


Fig. 1 Rendering of the Tiltwing concept vehicle. (a) Cruise Configuration. (b) Hover Configuration.

dataset includes: the location of the propellers along the span, the leading edge offset of the propellers, the vertical positioning of the propellers, the deployment angle of the flap, the disk loading of the propellers, and alpha sweeps at various free stream velocities. Table 1 provides a rough estimate of the number of cases that would be needed to cover the entire parameter space from this list. It is not feasible to run every permutation within the table in either wind tunnel tests or CFD simulations, so a subset of conditions is used to investigate the variation in each parameter. Figure 2 shows a visual representation for the clean wing and the Flap 20° versions of the TWTR geometry.

Table 1 TWTR Experimental Parameters

Parameter	Min Value	Max Value	Increment	Parameter Space
Prop Spanwise Offset	0 in	4 in	1 in	5
Prop Vertical Offset	4 in	6 in	0.5 in	5
Prop LE Offset	6 in	12 in	1.2 in	6
Flap Deflection	0°	60°	20°	4
α	0°	90°	10°	10
V_∞	0 knots	150 knots	-	4
Rotor Model	-	-	-	3
Turning Direction	-	-	-	4
# of Possible Cases	288,000			

Table 2 TWTR Pre-Test CFD Studies

Study Name	# of Free Stream Conditions	# of Geometries	# of Proprotor Models	# of Cases
Proprotor Modeling	6	2	3	18
Configuration Build-up	10	4	1	40
Proprotor Turning Direction	24	1	1	24
Flap Deflection	10	3	1	30
Total # of Cases	112			

The objective of this work is to utilize OVERFLOW[5] to assess the capability of current practices to simulate this type of vehicle under the same conditions that were being run during the wind tunnel tests. There are two general methods for modeling the propellers. The first is fully resolving each individual blade for the rotors and using prescribed

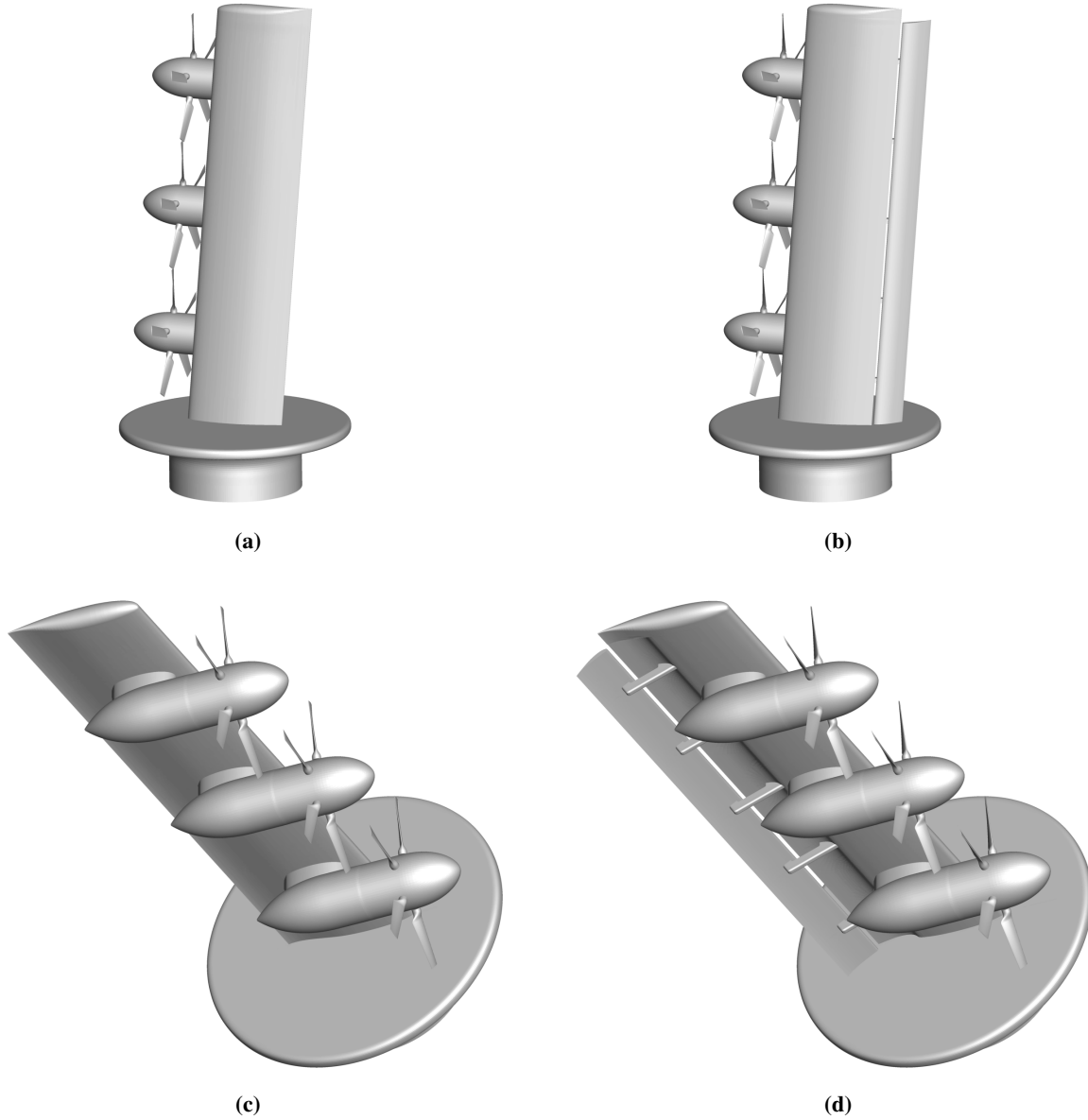


Fig. 2 Images showing wind tunnel geometry for the (a) upper surface of the Clean Wing, (b) upper surface of the Flap 20°, (c) lower surface of the Clean Wing, and (d) lower surface of the Flap 20° configurations

motion to spin them in the simulation. The second is to use an actuator disk model to negate the expense of having to both mesh the blades and run time-accurate simulations to capture the motion. Using an actuator disk is less accurate in terms of the physics being captured, but is significantly more cost efficient, on the order of tens of times cheaper to obtain a solution compared to a moving blade simulation.

This significant reduction in computational cost is why the first study in this work is focused on determining which actuator disk model is able to better capture the physics of the problem. This was done by running a set of fully time-accurate simulations that fully resolved the blades and their motion at key test points using Overflow. Then, solutions produced from steady state runs using the different actuator models are then compared to the fully unsteady runs. The model that most closely matches the aerodynamic performance predicted by the fully resolved propotor simulations was then used to perform subsequent parameter sweeps from the experiment. Specifically, a component build-up study, a propotor turning direction study, and a flap deflection study. Table 2 provides a breakdown of each of these studies. The goal of running these studies is to find a set of best practices for using these types of models for this

type of problem. The hope is to then use experimental data from the wind tunnel test, once it is available, to validate these methods.

III. Mesh Generation Procedure

The newly developed EPOGS toolset[6][7] in NASA's Chimera Grid Tools (CGT) software package[8] is used to generate the meshes for this work. This software has been developed at NASA Ames to automatically generate surface grids, volume grids, perform domain connectivity, and output the flow solver input files starting from a BRep solid. This method has been shown to work on complex geometries in Chuen et al.[9] and was applied to all of the individual meshes in this work. The major benefit to using EPOGS is the large reduction in the time it takes to generate these meshes, especially for the more complex geometries. The axisymmetric actuator disk model meshes had to be generated manually since there is no geometry associated with them that can be used by EPOGS. These simple cylindrical polar meshes are created using script library procedures from CGT. OVERFLOW's mixed domain connectivity mode is used for performing the domain connectivity between the rotor blades/disks and the static components. The domain connectivity of the static components alone is already automatically performed by EPOGS as part of the automatic overset mesh generation procedures.

Table 3 Mesh parameters used for manual and automatic mesh generation.

Mesh Parameter	Max Stretching Ratio	Δs_{max} (Wing)	Δs_{max} (Proprotor)	np_{min}	Δs_{wall}	Max Turning Angle
Value	1.1	0.05 ft	0.0045 ft	15	4.58×10^{-6} ft	4.0°

The general meshing guidelines follow Chaderjian's[10] best practices for accurately simulating a rotor in hover. The main recommendation is to keep the maximum mesh size to under 10% tip chord in all directions for the rotors. A similar metric was applied to the wing mesh, where its maximum spacing was also limited to 10% of its tip chord. All of the meshes were generated using wall spacings estimated from a $y^+=1$ from flat plate theory. The inputs to EPOGS are given in Table 3. Table 4 shows a comparison of the EPOGS meshing statistics for the various geometric configurations. This table shows that for the most complicated version of the geometry, the Wing + Flap 20° + Brackets + Nacelles configuration, it required roughly 2 days of manual effort to repair. This was mainly caused by the acute angle of the intersections of the flap brackets with the lower surface of the flap. To manually construct a set of meshes for this same geometry would take approximately 2 weeks. This significant reduction in manual effort allowed for the rapid generation of all the meshes used in this work. For example, the meshes needed for the four different geometries needed to complete the component build-up study were completed in about 1 week.

Table 4 EPOGS Meshing Statistics

Case Name	# of Points (Millions)	# of Grids	Auto Meshing Time (minutes)	Manual Repair Time (hours)
Clean Wing	29.1	30	5	2
Clean Wing + Nacelles	56.7	106	7	4
Wing + Flap 40°	53.1	50	10	4
Wing + Flap 40° + Brackets	93.2	50	13	12
Wing + Flap 20° + Brackets + Nacelles	120.3	250	15	16
Wing + Flap 40° + Brackets + Nacelles	106.3	232	15	12
Wing + Flap 60° + Brackets + Nacelles	125.2	236	15	12
Proprotor Blade	7.1	27	2	0

Examples of surface meshes that were generated to model the experiment are shown in Figure 3. One important note is that the tunnel floor is modeled as an inviscid wall. This was done because examining the wall effects is beyond

the scope of this work. Figure 3a and Figure 3b show the differences in mesh topologies between the clean wing and the Wing + Flap 20° + Brackets + Nacelles configurations. Figure 3c and Figure 3d show the difference between the actuator meshes and the fully modeled proprotor blades. All of the meshes for this work are generated using these same general parameters to ensure that the meshes are as consistent as possible across all geometric configurations.

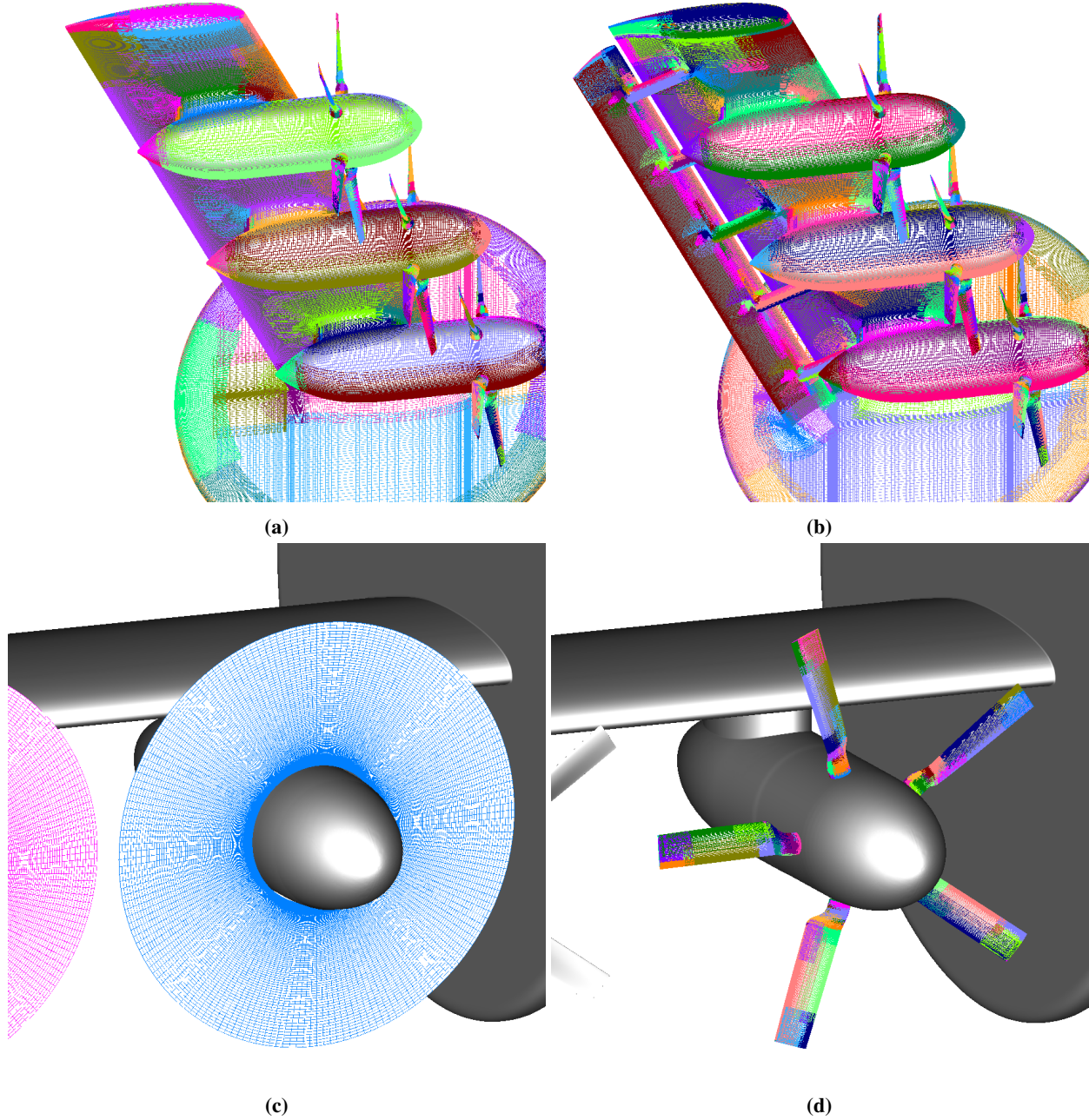


Fig. 3 Surface grids of the TWTR. (a) Clean wing configuration. (b) 20° flap configuration with the flap brackets and nacelles. (c) Inboard most actuator disk grids. (d) Inboard most rotor blade grids

IV. Computational Method

All simulations in this work were run with the 2.4c version of the OVERFLOW code which is a finite-difference, structured overset grid, Navier-Stokes flow solver. It was run in both the steady-state and time-accurate modes. The turbulence was modeled using the Spalart-Allmaras (SA) turbulence model and with the Rotation/Curvature correction (RC) and the 2013 version of the Quadratic Constitutive Relation (QCR). Overflow's Global Linear Solve (GLS) option was also activated for every simulation since it has been shown to improve convergence. For the steady state, Reynolds Averaged Navier-Stokes (RANS) runs, the Roe upwind scheme was used with the WENO option for the spatial discretization. The solution was advanced using the improved successive over-relaxation scheme (SSOR) with 10 subiterations and the flow was computed using Low-Mach Preconditioning.

The time-accurate runs utilize the Roe and HLLC++ upwind schemes with the WENO scheme for the spatial discretization and the solution was advanced using the improved SSOR. The intention was for every case to be run using the Roe upwind scheme, but some cases required the use of the more dissipative HLLC++ scheme to better converge. To improve convergence at the start of these cases, the number of SSOR subiterations was limited to 2 to get through the initial transients and then was raised to 10 after the solution had stabilized, similar to what is described in Derlaga[11]. Additionally, the time-step was ramped following the method described in Chaderjian's[10] work with a slight modification. In Chaderjian's paper, the case is started with a coarse time-step, roughly 8x the final value, for a few revolutions before running more revolutions at the final time-step value. For these cases, using the coarse time-step was leading to instabilities on the simulation start. To overcome this, the simulations were run for half of a rotor revolution with the fine timestep, restarted using the coarse timestep for 4 revolutions, and finally restarted from there using the fine timestep. The inclusion of the full wind tunnel in the flow domain was beyond the scope of this work. Therefore, the floor was modeled as an inviscid wall that extends to the far-field.

The two propotor models used in this work are the Pressure Difference Model (pdiff) and the Rotor Disk Model (rdisk)[12]. The pressure difference model applies a constant pressure difference at a specified grid plane, while the rotor disk model uses Blade element theory to model the rotors, and introduces the induced effect of the rotors on the flow-field through source terms in the momentum conservation equations. For this work, two rotor disk models were considered: a constant thrust mode and the C81 airfoil table mode. The first takes the coefficient of thrust (C_T) as the only input and will non-uniformly modify the source terms to yield an integrated value across the boundary that matches the specified C_T . The second mode uses C81 airfoil tables, twist, and thickness distributions to approximate the blades. As the run progresses the model will trim the blades using the pitch angle to reach the specified C_T . Unless specified otherwise, all cases using the Rotor Disk model in this work use the clockwise turning direction for all three propotors.

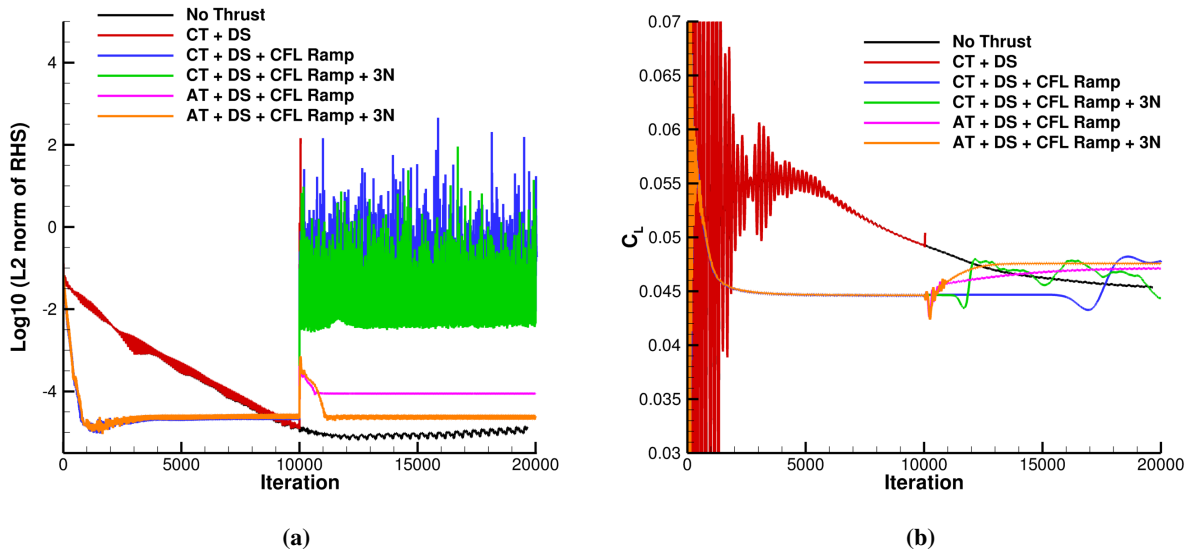


Fig. 4 Convergence comparison between the constant thrust (CT) and airfoil table (AT) versions of the Rotor Disk model. (a) Residuals. (b) C_L .

A comparison between the different disk models for a simplified version of the TWTR, a clean wing with the

actuators in front, is shown in 4. This was used to determine the best way to use this boundary condition for this problem. Initial simulations with the constant thrust (CT) version of the boundary condition (BC) activated at simulation start were unable to converge. The first attempt to fix this was to delay the start (DS) of the BC until further into the simulation to allow the flow field to better converge. However, these simulations still diverged once the BC was activated. To try and preserve stability, CFL ramping was added. Next, applying the BC over multiple indices rather than a single index was attempted. In this case, applying the BC over three planes (3N) seemed to work well. Using more was found to not have any improvements to the flow field convergence. Figure 4a shows that using both of these options allowed the constant thrust simulations to stay relatively stable, but they do not converge. This lack of convergence is especially clear in Figure 4b. This is in contrast with the observed behavior of the airfoil table (AT) version of the Rotor Disk model. Figure 4a shows that the single index and three index versions of the BC reach a reasonable level of convergence, with the three index version reaching slightly deeper convergence. Both versions are able to converge in the loads as well, with the latter reaching the converged value more quickly, as shown in Figure 4b.

V. Results

The results from the various CFD studies performed on the TWTR are split into four sections. The first is the study of different disk models, the second is investigating the effects of adding geometric complexity to the computational model, the third is varying the turning directions of the proprotors, and the fourth is a flap deflection study. The best disk model from the first study is used in the subsequent studies.

A. Proprotor Modeling

The goal of this study is to determine the optimal method for modeling the proprotor. The two candidates are the Pressure Difference model (pdiff) and the more complex Rotor Disk model (rdisk) within Overflow. The desire to model the proprotors stems from the fact that the unsteady simulations using full 3D motion are prohibitively expensive. For one of the moving blade simulations in this work, it required the use of 800 CPUs for 14 days of wall time while a single disk model case only took 520 CPUs for 16 hours. That is a savings factor of 32x when running one of the disk models. To determine which model was the best, a set of six simulations were run using the moving blades, which were then used as the baseline for the comparisons. This is because the two actuator models have simplified physics compared to the moving blade cases. The model that more closely matched the loads and flow field characteristics from these baseline simulations was then chosen to run the remainder of the cases in this work.

The conditions for the proprotors were assigned based on the maximum attainable RPM in the wind tunnel and specifying a constant disk loading of 14 psf for all of the cases. The only exception to this is one cruise configuration case where the disk loading is set to 4 psf to match with a more realistic cruise setting. The disk loading values were used with the tip radius of 1 ft to calculate the input C_T for the Rotor Disk model and the corresponding $\Delta P/P_\infty$ for the input to the Pressure Difference model. A set of C81 airfoil tables created from the blade CAD, thickness distribution, and twist distribution were also used as inputs to the Rotor Disk model. The moving blade cases used the RPM, tip radius, and blade pitch radius as the inputs. The RPM and tip radius were used to calculate the physical timestep needed to advance the solution as well as the non-dimensional rotational speed. This was verified by comparing the tip speed from the simulation results to the desired tip speed.

The blade pitch angles were taken from previous analyses of the proprotors that were done using the Comprehensive Hierarchical Aeromechanics Rotorcraft Model (CHARM)[13] tool. Due to the differences in how the blades are modeled between CHARM and Overflow, the two codes typically predict different amounts of thrust for the same proprotor conditions.

Due to the expensive nature of the time-accurate simulations needed to fully model the proprotors, the number of cases for this study was kept as low as possible. The original intent was to run every case listed in Table 5 but, due to time and resource constraints, only the Hover condition was simulated. The remainder of the cases will be run as part of future work. These six conditions were chosen to best cover the ranges of free stream velocities, tilt angles, and disk loadings that will be used in the rest of this work.

Figure 5 shows the residual convergence across all the cases in this study. One important note is that for the case with the moving blades, the plotted value is the total drop in the residual over the sub iterations for each time step. In the input files for the case, this was set to be 2.5 orders of reduction based on experiences with similar problems in past work. Figure 5a shows that initially, the residuals for the moving blade case are only dropping by a single order of magnitude. This is caused by the large initial transients in the flow field as well as the larger timestep used at the beginning of the simulations. After the simulation is restarted using the finest timestep, the sub iteration convergence

Table 5 Conditions for The Proprotor Modeling Study. Configuration A is the Clean Wing + the

Case Name	T_∞	V_∞	α	RPM	Disk Loading	Geometry
Hover	59°F	0 knots	90.0°	4000	14 psf	Clean Wing
High Thrust Cruise	59°F	110 knots	0.0°	4000	14 psf	Clean Wing
Low Thrust Cruise	59°F	110 knots	0.0°	4000	4 psf	Clean Wing
Low Speed Transition	59°F	20 knots	80.0°	4000	14 psf	Clean Wing
High Speed Transition	59°F	40 knots	55.0°	4000	14 psf	Clean Wing
Flapped Transition	59°F	20 knots	55.0°	4000	14 psf	Flap 40.0°

reaches approximately 2 orders of magnitude. While this is not the 2.5 specified in the input file, it was deemed good enough for this study. With the finest time step being set such that the blades complete a revolution in 1440 steps, the moving blade case has run approximately 15 revolutions. This includes the 4 revolutions run using a coarser timestep.

The Pressure Difference and Rotor Disk models are both showing relatively good convergence by reaching 4-5 orders of magnitude of reduction. The Pressure Difference model tends to converge slightly better than the Rotor Disk model, but that is not universally true. Despite this general level of convergence, there are still small amplitude oscillations present in the residuals, pointing toward the fact that there is unsteadiness in these flow fields.

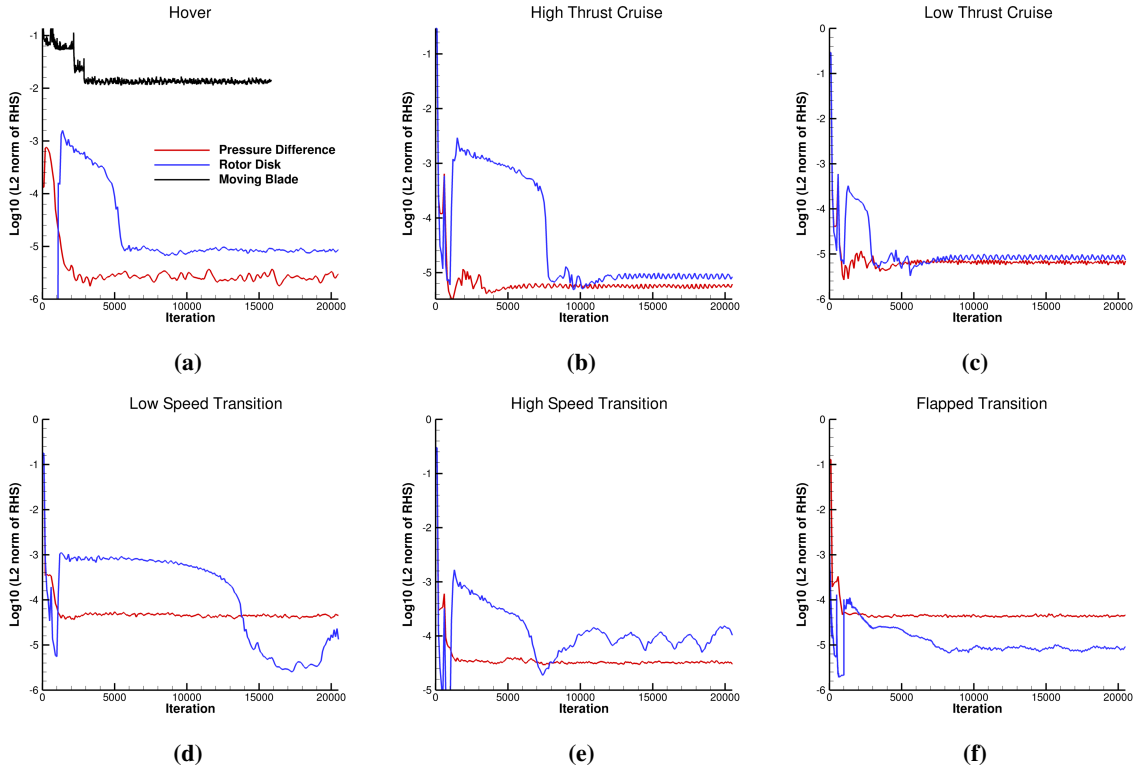


Fig. 5 Residual convergence plots the proprotor modeling simulations.

Before looking at the loads for the whole test article, the thrust produced by the three proprotor models were compared against each other to see how closely they matched each other as well as the target thrust values. Figure 6 shows that both disk models are doing a very good job of matching the target thrust of 44 lbf for the cruise conditions, while the Pressure Difference model is over predicting the thrust by about 1.5 lbf for the non-cruise cases. The Rotor Disk model is doing a better job of matching the target thrust across all six conditions.

For the Hover case, the moving blade simulation is predicting a higher value of thrust than the target. This is being

caused by the blade pitch angle being set based on the CHARM analyses, as mentioned earlier. For this particular case, the moving blade thrust is averaging a value between the Rotor Disk and Pressure Difference models. While the three methods are not predicting exactly the same thrust, they are within 3 lbf of one another. Having the models predict similar levels of thrust provides a solid foundation for making further comparisons.

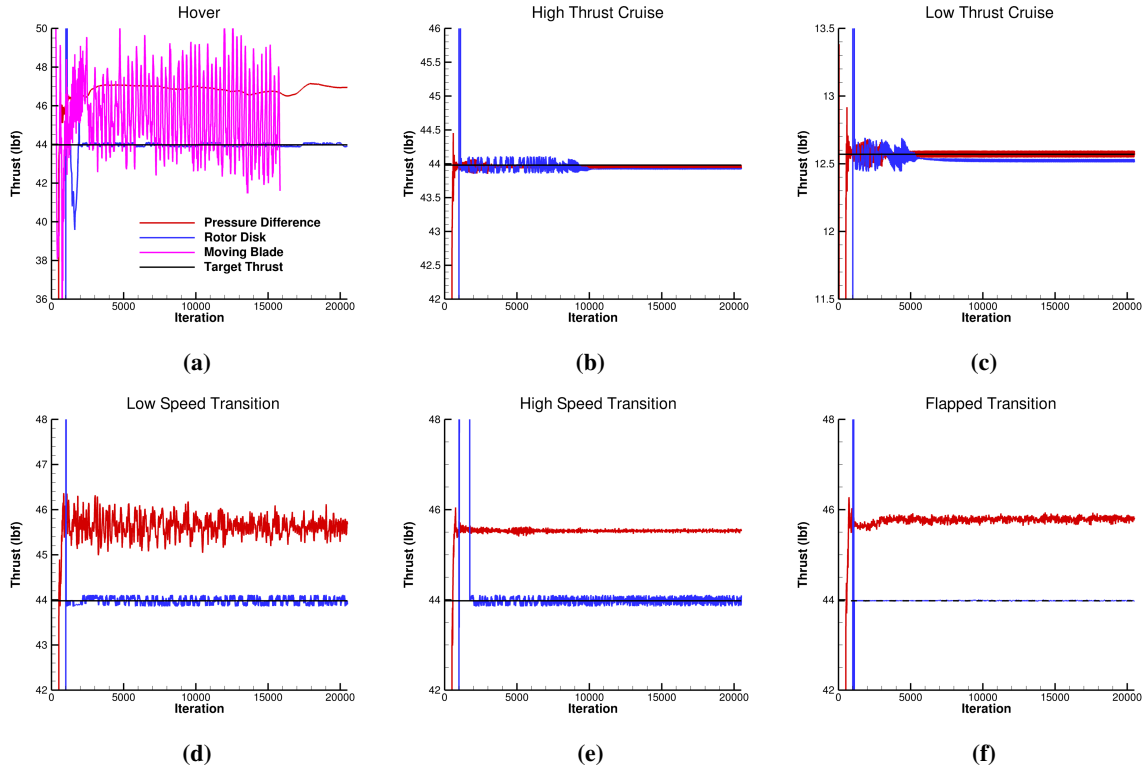


Fig. 6 Thrust convergence plots the proprotor modeling simulations.

Looking at the loads convergence plots within Figure 7, the general trend is that the Rotor Disk model and Pressure Difference model are predicting similar loads to each other. For the Hover, High Thrust Cruise, and Low Thrust Cruise, the Rotor Disk model is predicting more thrust and drag than the Pressure Difference model. While the inverse is true for the Low Speed, High Speed, and Flapped Transition cases. For the Hover case, both models are matching the Moving Blade lift well but the Rotor Disk model is predicting more drag while the Pressure Difference model is predicting less drag. This is being caused by the difference in the thrusts that are being produced by the proprotors. While the thrust values are similar, they are different enough to be causing differences in the loads over the entire model. Another cause of these difference is the large amount of unsteadiness in these simulations. Only the moving blade cases are being run with a time-accurate approach, so there is potential for significant differences in the Rotor Disk and Pressure Difference predictions due to them using RANS. While there are differences between the different models, the main takeaway is that both disk models are predicting similar performance to each other.

In addition to looking at the quantitative comparison of the loads and the thrust. The qualitative comparison was made between both disk models and the moving blades. Figure 8 shows the differences in the flow field when viewed down the span of the wing. When viewed from this angle the differences the Pressure Difference model is showing a wake with much lower velocity magnitude than the Rotor Disk model and the Moving Blade simulations. This same trend is seen when looking directly downstream onto the proprotors, as in Figure 9. There are significant differences in the magnitude of the velocity field and the distribution of the velocity due to the swirl effects. Due to the lack of torque in the Pressure Difference model, the wake behind the disk shows the acceleration of the velocity in the flow direction, but none of the mixing effects that are present in the moving blade and Rotor Disk simulations. Since the Rotor Disk model does include some of the torque effects, it produces a proprotor wake that matches more closely to the moving blade simulations. Figure 10 shows how the changes in the proprotor wakes propagate downstream past and over the wing. The differences between the two actuator disk models is even more pronounced when looking at the wake behind

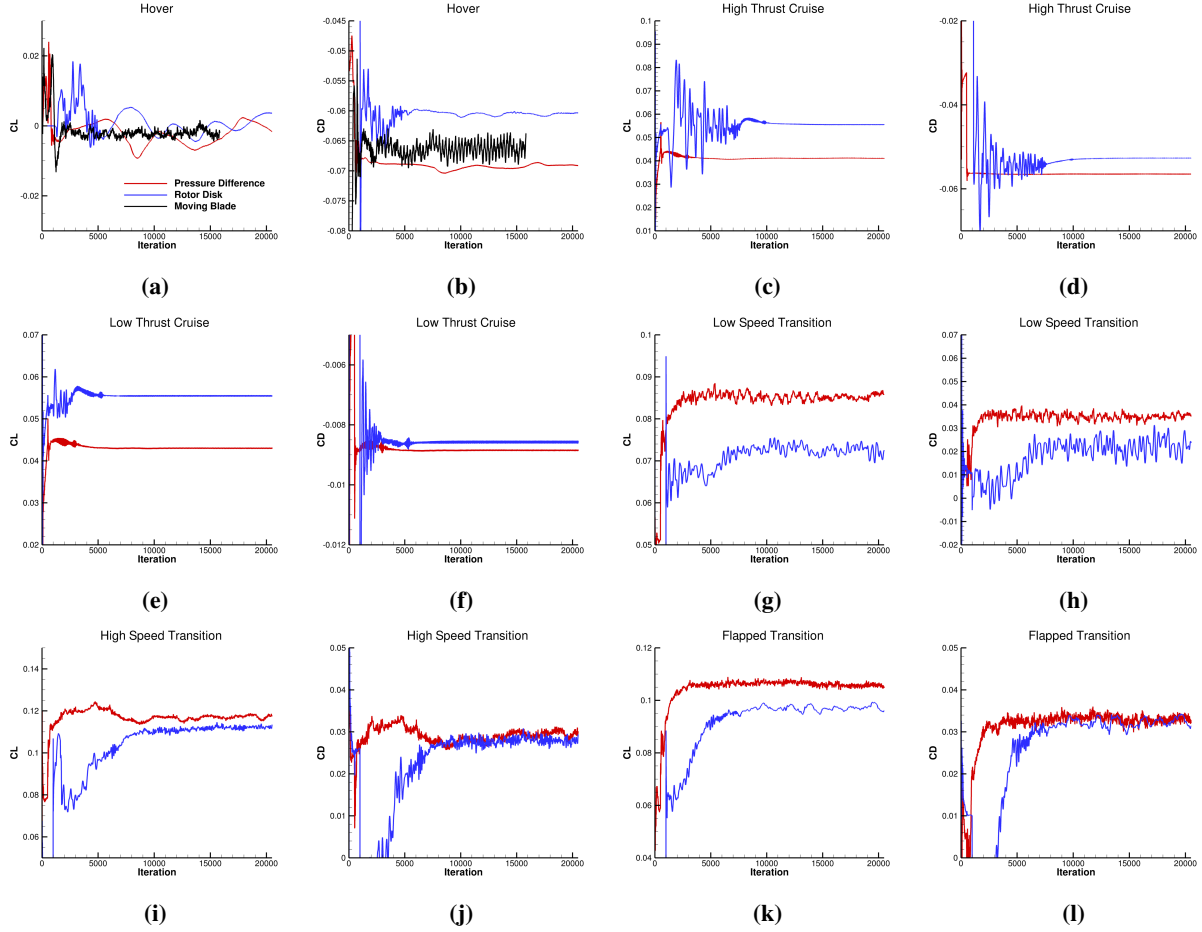


Fig. 7 Coefficient of lift and drag convergence plots for the proprotor modeling study.

the wing. The impact of the wakes on the wings and their interactions with each other are critical to predicting the loads on the wing. The different proprotor models are producing different wakes and those wake interactions with the wing are leading to the variations in the predictions shown in Figure 7. Since the Pressure Difference model neglects any torque effects, it does a worse job of matching the proprotor wakes. Therefore, all other simulations in this work will use the Rotor Disk model since it does a better job modeling the physics of the moving blades.

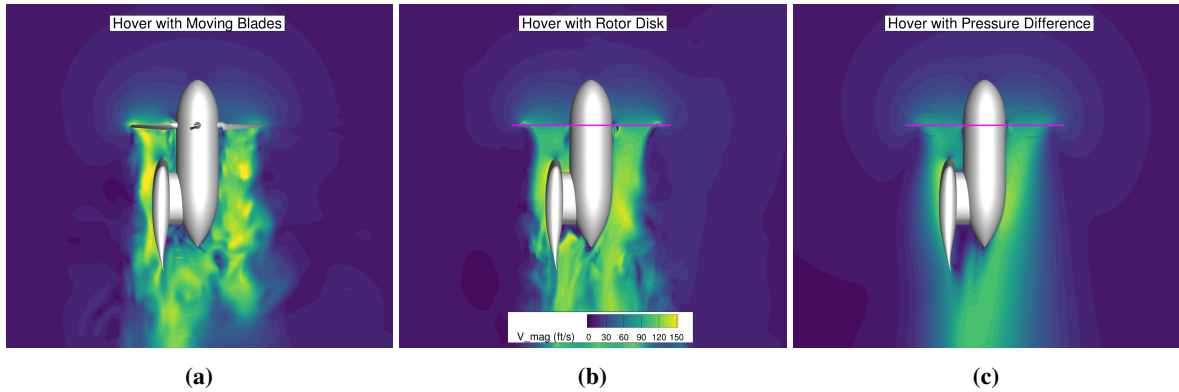


Fig. 8 Constant Y slice through the centerline of the outboard proprotor colored by the velocity magnitude for the Low Thrust Cruise case. The pink line denotes the location where the model BCs are applied.

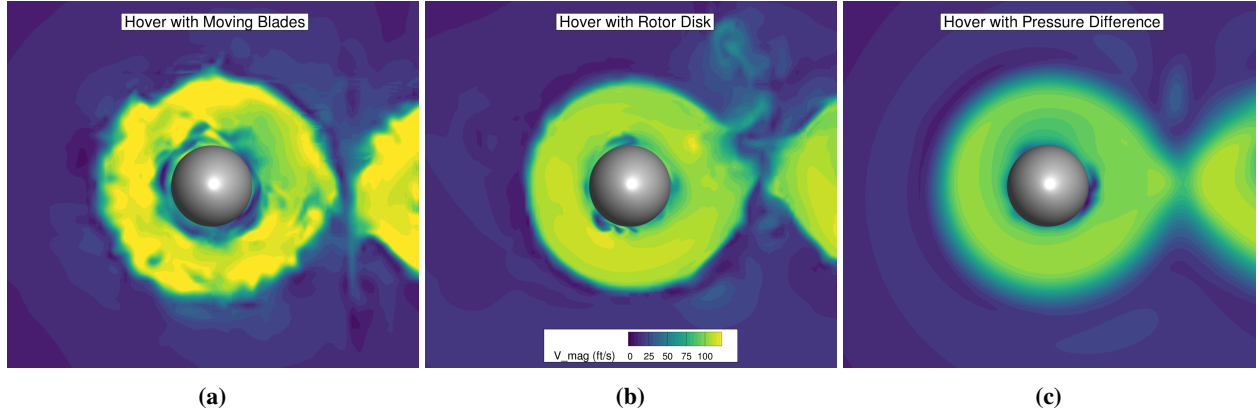


Fig. 9 Constant X slice behind the outboard proprotor colored by the velocity magnitude for the Low Thrust Cruise case.

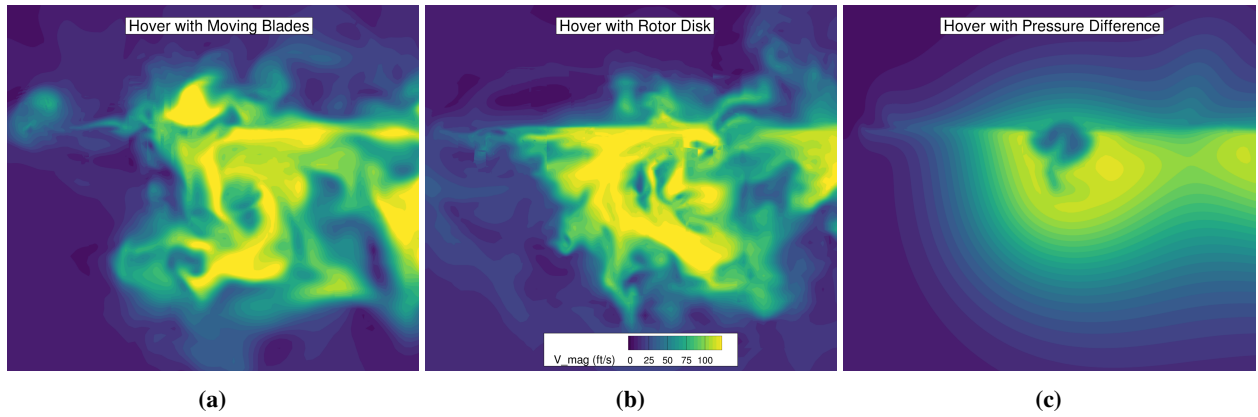


Fig. 10 Constant X slice behind wing colored by the velocity magnitude for the Low Thrust Cruise case.

B. Component Build-Up

The component build-up study was performed to better understand how each piece of the geometry contributes to the overall loads predicted on the model. This study performed following a similar process as what is being done with the experiment. The model is physically assembled in a few discrete steps: the wing, the flap with its mounting brackets, and the nacelles that are attached to the wing via their pylons. In the experiment, the proprotors are only able to be attached to the nacelles. For the CFD study in this work, it was decided to follow a similar process with two key differences. The first is that the geometry was broken into four steps by separating the flap and flap bracket increments. The second is that the proprotors were modeled for every geometric configuration. This was done to better understand the effects of the flap brackets themselves and to examine the differences in the performance of the wing with the proprotors included while adding complexity. This means the study was done using four configurations: clean wing, flapped wing with 40° flap, flapped wing with flap and flap brackets, and flapped wing with flaps, brackets, nacelles, and pylons.

These four different geometries are shown in Figure 11. The cases for this study were all run with a free stream temperature of 59° F, free stream velocity of 20 knots, and a disk loading of 14 psf. Each version of the geometry went through a simulated transition between the cruise and hover positions which was simulated by varying the α between 0° and 90° using 10° steps.

The resulting load curves are shown in Figure 12. When the geometry is limited to just the wing, the peak lift is occurring around $\alpha = 40^\circ$ while the maximum drag is all the way at the hover condition (90°). Once the Flap is included, the maximum lift shifts to $\alpha = 30^\circ$ and the maximum drag location is reached at $\alpha = 80^\circ$. Including the brackets slightly decreases the maximum predicted lift while not having a major effect on the drag. Adding the nacelles

and pylons causes a drop in the predicted lift across the α sweep. The maximum lift is still predicted to occur at $\alpha = 30^\circ$ but the lift falls off from the peak more quickly than the no-nacelle cases. The drag increases significantly when the nacelle and pylons are added due to the sheer size of the nacelles. The nacelle wakes prove to be major driver behind the changes in the lift and drag profile of the wing.

Figure 13 shows a comparison of the flowfield through the center of the middle propotor. Each row of figures corresponds to a different geometry, starting with the wing only at the top and ending with the full model that includes the flap, brackets, nacelles, and pylons. Each column of the figure is the $\alpha = 0^\circ$, maximum CL, and $\alpha = 0^\circ$ cases. This figure is showing that the flow over the wing stays attached up to the α of the highest CL. Once past that angle of attack, the flow over the upper surface of the wing starts to massively separate and causes a large reduction in the predicted lift. In this figure, the flap is shown to be increasing the pressure gradient across the wing, increasing the amount of lift. It also shows that the flap brackets aren't significantly contributing to the drag because the flow on the under side of the wing is already separated by $\alpha = 30^\circ$, leading to a less pronounced effect on drag. This figure also shows how the sheer size of the nacelles is creating a large wake that is directly impinging on the leading edge of the wing. As the angle of attack is increased, this effect becomes even more pronounced. Overall, the component build-up study showed that the lift was significantly increased with the addition of the flap, the inclusion of the brackets didn't significantly change the performance of the model, and the nacelles have a large impact on the overall performance of the wing.

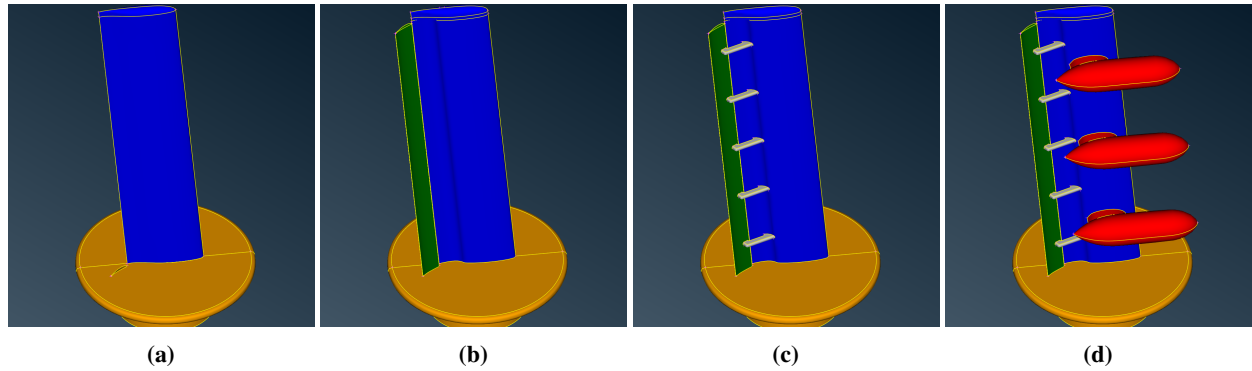


Fig. 11 Geometries for the component build-up study. (a) Clean Wing. (b) Flapped Wing. (c) Flapped Wing with Brackets. (d) Flapped Wing with Brackets and Nacelles.

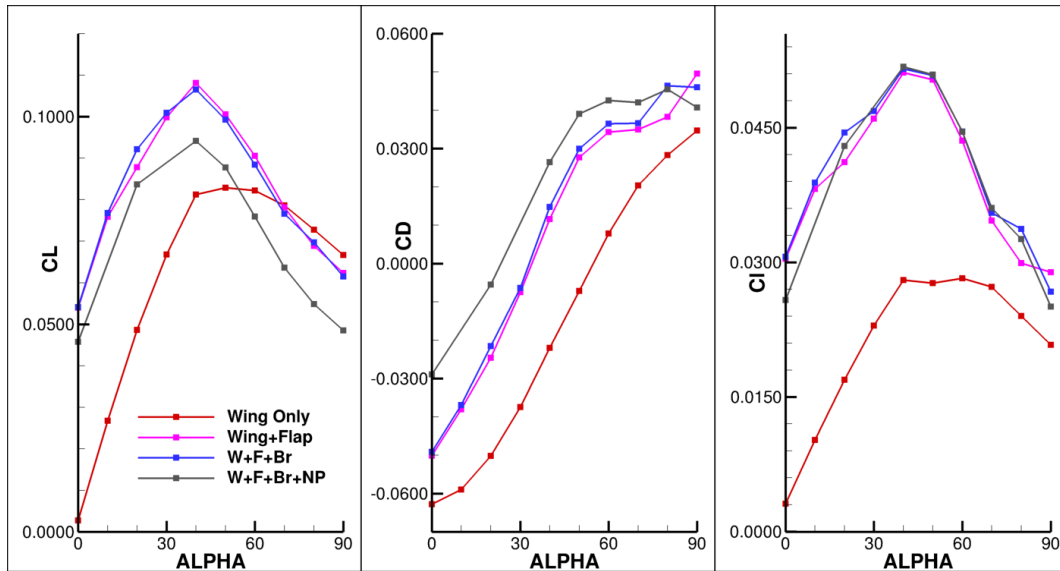


Fig. 12 Component build-up loads.

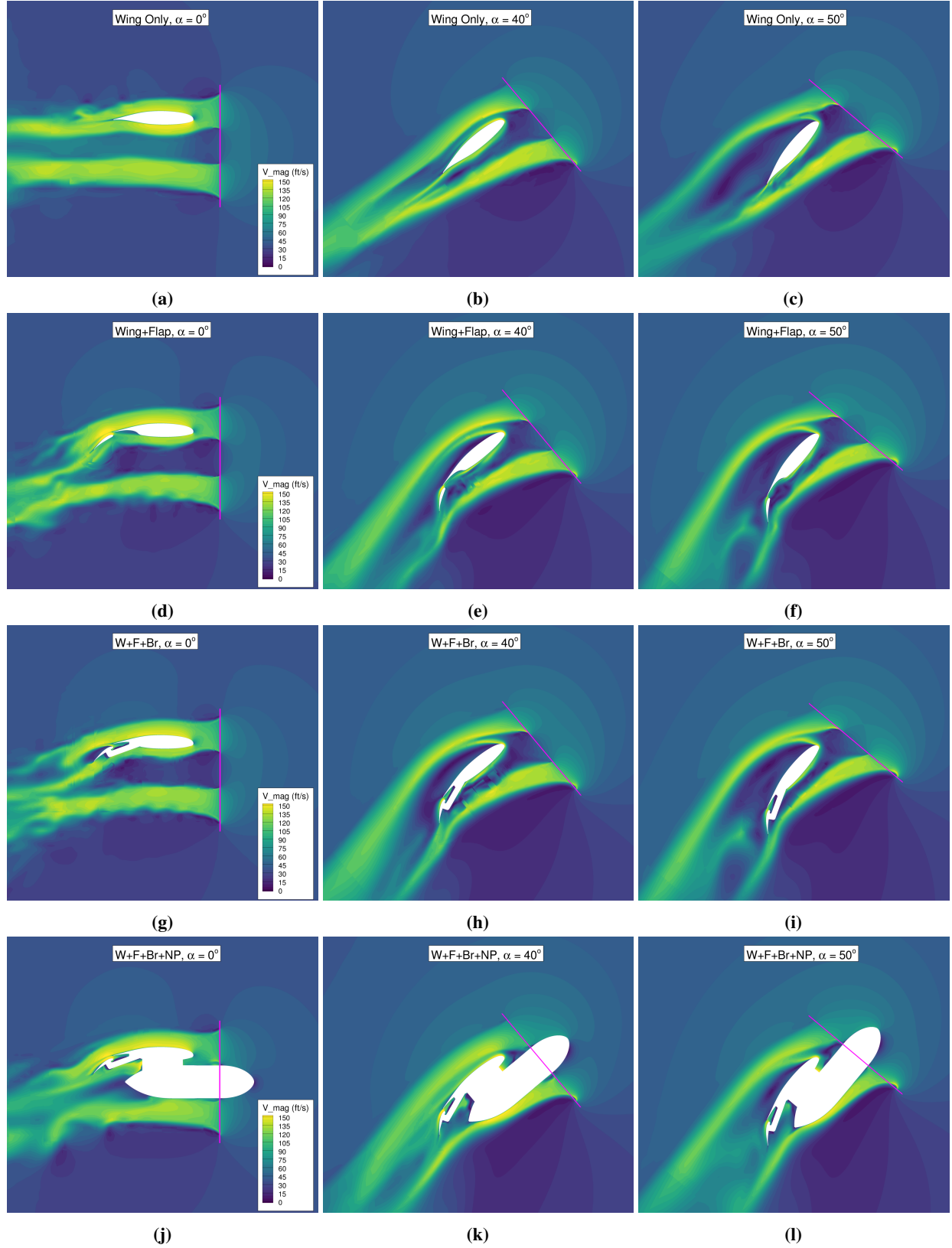


Fig. 13 Constant Y slices through the centerline of the middle nacelle colored by the velocity magnitude. (a), (b), and (c) are the Wing Only; (d), (e), and (f) are the Wing+Flap; (g), (h), and (i) are the Wing+Flap+Brackets; (j), (k), and (l) are the Wing+Flap+Brackets+Nacelles & Pylons. The pink line denotes the location where the model BCs are applied.

C. Proprotor Turning Direction

The first of the two experimental parameter sweeps that was examined was the turning direction of the proprotor. This study was enabled through the use of the Rotor Disk model's inclusion of torque effects. Four different rotor turning combinations were used, every proprotor turning clockwise, every proprotor turning counterclockwise, inboard turning clockwise with the two outboard turning counterclockwise, and inboard and outboard turning clockwise while the middle turns counterclockwise. Clockwise and counterclockwise are defined as if looking out the cockpit of the aircraft, equivalent to looking from the perspective of the -X axis. The 20° Flap version of the geometry was chosen for this study. The cases for this study were all run with a free stream temperature of 59° F, free stream velocity of 20 knots, and a disk loading of 14 psf. Each version of the geometry went through a simulated transition between the cruise and hover positions which was simulated by varying the α between 0° and 90° using 10° steps.

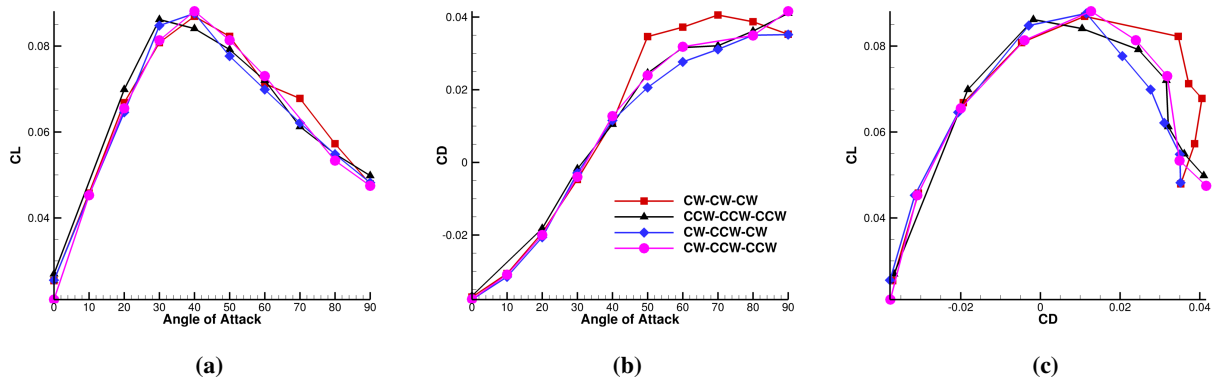
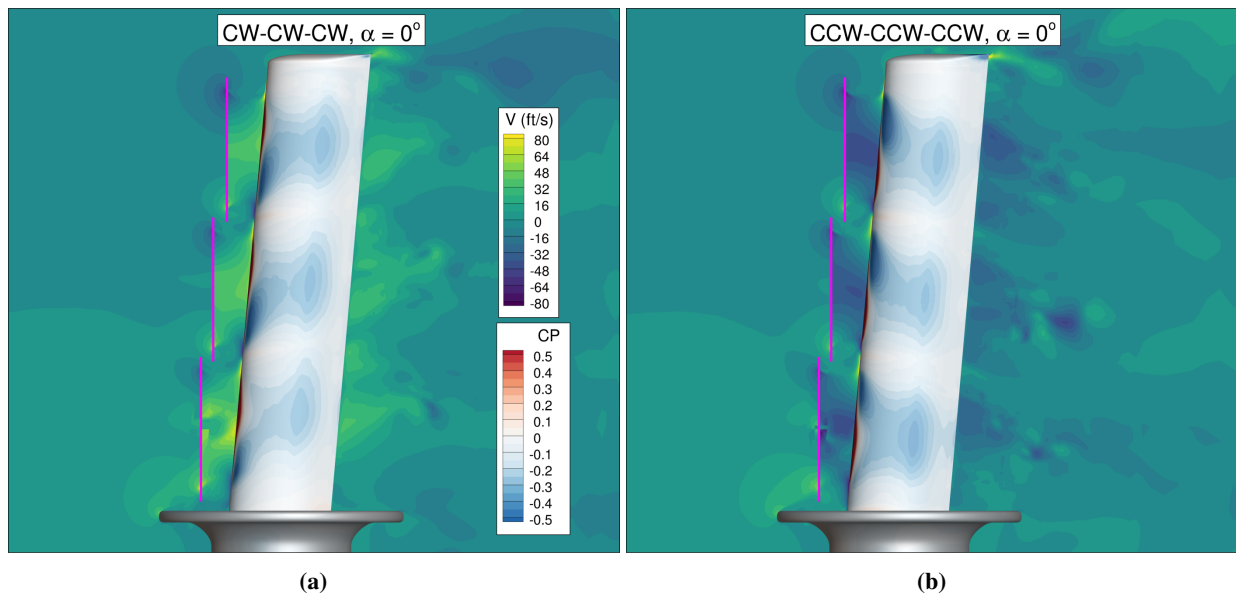


Fig. 14 Proprotor turning direction study loads.

Figure 14 shows the loads data from these simulations. With the exception of the all counterclockwise case, the angle of maximum lift occurs at $\alpha = 40^\circ$. The all counterclockwise case has the maximum lift occur at $\alpha = 30^\circ$. While the angle where it occurs is different, the magnitude of the maximum lift is similar for each turning direction combination. When looking at the drag, the all clockwise case is showing the highest drag for the higher angles of attack. This is leading it to have the most unfavorable lift to drag ratio. In terms of the lift to drag ratio, all four turning directions are showing similar performance until they reach their maximum lift. Beyond that, the clockwise-counterclockwise-clockwise configuration is showing the best performance.



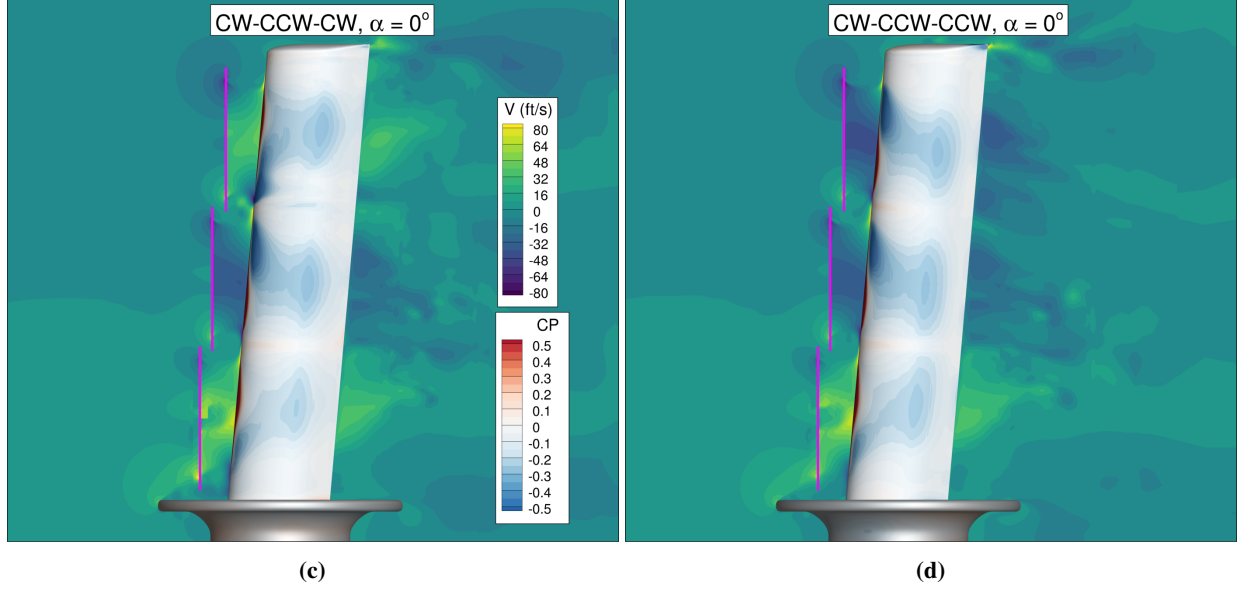


Fig. 15 Constant Z slices through the leading edge of the wing colored by V-velocity with the pressure coefficient on the surface of the geometry for the proprotor turning direction study. The pink line denotes the location where the model BCs are applied.

Figure 15 shows the differences between the different turning direction combinations at $\alpha = 0^\circ$. The velocity contours and surface pressure show how the flow field behind the rotor is interacting with the wing's upper surface. Taken in combination with the loads data, the effects of having the inboard and outboard proprotors turning clockwise is leading to more favorable flow over the wing at the high angles of attack. This study proves that the Rotor Disk model is able to show some of the sensitivity caused by changing the turning direction of the proprotors. Making any conclusion about the validity of those predictions requires experimental data that is not yet available.

D. Flap Deflection

The final test parameter sweep investigated in this work is looking at the different flap deflection angles. In the experiment, three different flap settings were tested, 20° , 40° , and 60° . To assess the performance of the flaps, all three settings were put through an α sweep from 0° to 90° with 10° increments. The general free stream and proprotor parameters are the same as the turning direction study. This means a T_∞ of $59^\circ F$, V_∞ of 20 knots, and disk loading of 14 psf.

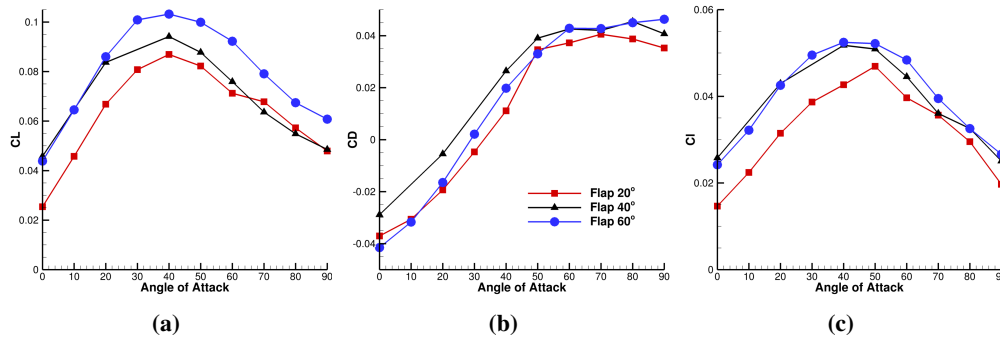


Fig. 16 Flap deflection study loads.

The predicted loads for the different flap deflections are shown in Figure 16. It can be seen that the 20° flap setting has the lowest lift, drag, and rolling moment across the whole α sweep. The peak lift is predicted to be at an α of 40°

for every flap setting. Interestingly, the flap 60° case has drag between the 20° and 40° cases. Additionally, figure 17 confirms that the wing has mostly separated by the time the angle of attack reaches 50°. While the flow along the upper surface of the wing is staying attached, the higher flap setting is providing a lift benefit, but once the wing separates, this benefit is lost. Being able to study how the flap and the blowing from the prop rotor interact to affect the predicted loads is useful for verifying the performance of this type of aircraft. Utilizing the rotor disk model allows this type of analyses to be performed using much less resources than if they were simulated using fully unsteady simulations with 3D moving blades.

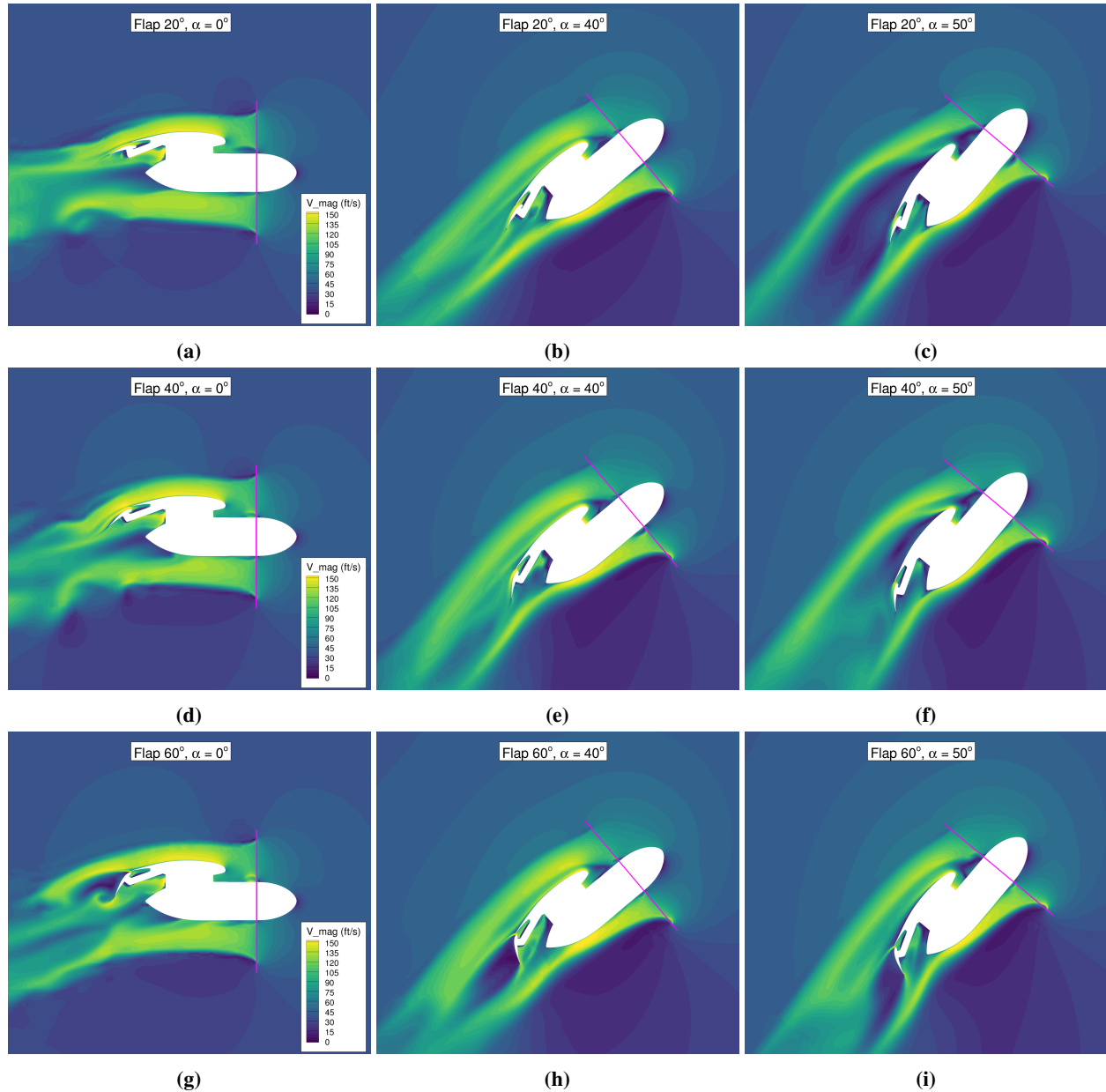


Fig. 17 Constant Y slices through the centerline of the middle nacelle colored by the velocity magnitude for the Flap 20° and Flap 40° cases.

VI. Conclusion

A pre-test CFD study on the TWTR experiment was performed using the EPOGS automatic overset meshing tool and the Overflow flow solver. A significant reduction in turn around time was demonstrated in preparing the meshes for the various configurations needed for the studies. Meshes needed for the component build-up study and the flap deflection study each took approximately 1 week to create using EPOGS, while a completely manual approach would have taken about a month for each case.

A propotor model study was done using the TWTR geometry and conditions to determine the best disk model to use in place of the 3D moving blades. After comparing the Pressure Difference and Rotor Disk models within Overflow, the Rotor Disk model was chosen for its ability to better representation of some of the critical flow physics. This lead to a better match to the target thrust value and to the loads predicted by the moving blade simulations. The Rotor Disk model was then used to perform a component build-up study, propotor turning direction study, and a flap deflection study. The component build-up study showed that adding the flap significantly increased the predicted lift and including the nacelle and pylon significantly increased the predicted drag. The propotor turning direction showed that the clockwise-counterclockwise-clockwise configuration had the best performance at the high angles of attack while the all clockwise configuration had the worst performance. The flap deflection study showed that increasing the angle of the flap lead to an increase in the predicted lift and drag, while maintaining the same angle of maximum lift. Across each of these studies, a universal set of best practices were applied to running the Rotor Disk boundary condition and were shown to be able to produce converged results. These results will eventually be compared against experimental data to assess the quality of the predictions presented in this work.

Acknowledgments

This work is funded by NASA's Revolutionary Vertical Lift Technology (RVLT) Project. Computational resources have been provided by the NASA Advanced Supercomputing (NAS) facility at NASA Ames Research Center. The authors would also like to thank Dr. Joeseeph Derlaga for help with questions relating to Overflow and the rotor disk model, his help was instrumental in finding the parameters that would keep the rotor disk model and URANS cases stable. The authors would also like to acknowledge the critical contributions from Brandon Litherland, from helping identify the most relevant test parameters to study, providing the geometry for the model, and providing the CHARM analysis of the proprotors. The authors would also like to acknowledge Dr. Jasim Ahmad for many productive conversations about the rotor disk model and its implementation. Finally the authors would like to acknowledge Eduardo Solis for generating the BRep solids used in this work, without his help, we would not have been able to use POGS to generate our many different meshes.

References

- [1] Whiteside, S. K. S., Pollard, B. P., Antcliff, K. R., Zawodny, N. S., Fei, X., Silva, C., and Medina, G. L., "Design of a Tiltwing Concept Vehicle for Urban Air Mobility," Tech. Rep. NASA/TM-2021-0017971, NASA Langley, 2021. URL <https://ntrs.nasa.gov/api/citations/20210017971>.
- [2] Gentry, J., Garl L., Quinto, G. G. M., P. Frank, and Applin, Z. T., "The Langley 14- by 22-Foot Subsonic Tunnel: Description, Flow Characteristics, and Guide for Users," Tech. Rep. NASA-TP-3008, NASA Langley, 1990. URL <https://ntrs.nasa.gov/citations/19900018333>.
- [3] Zawodny, N. S., Pascioni, K. A., and Thurman, C. S., "An Overview of the Propotor Performance Test in the 14- by 22-Foot Subsonic Tunnel," *The Vertical Flight Society's 79th Annual Forum Technology Display*, The Vertical Flight Society, West Palm Beach, Florida, 2023. URL <https://ntrs.nasa.gov/citations/20230004067>.
- [4] *Tiltwing Validation Test: Conceptual Design and Research Objectives*, NASA Langley, 2024. URL <https://ntrs.nasa.gov/api/citations/20240002905>.
- [5] Buning, P. G., and Pulliam, T. H., "Near-Body Grid Adaption for Overset Grids," AIAA 2016-3326, 2016. <https://doi.org/10.2514/6.2016-3326>.
- [6] Chan, W. M., Pandya, S. A., and Chuen, A. M., "Automation of Overset Structured Mesh Generation on Boundary Representation Geometries," AIAA 2022-3607, June 2022. <https://doi.org/10.2514/6.2022-3607>.
- [7] Chan, W. M., Chuen, A. M., and Jensen, J. C., "Advances in Automation of Overset Structured Volume Mesh Generation and Domain Connectivity," AIAA 2024-4305, June 2024. <https://doi.org/10.2514/6.2024-4305>, URL <https://arc.aiaa.org/doi/abs/10.2514/6.2024-4305>.

- [8] Chan, W. M., Gomez, R., Rogers, S., and Buning, P., “Best Practices in Overset Grid Generation,” AIAA 2002-3192, 2002. <https://doi.org/10.2514/6.2002-3191>.
- [9] Chuen, A. M., Hosseini, S. S., Jensen, J. C., and Chan, W. M., “Aerodynamic Simulations for Complex Geometries Using Automatically Generated Structured Overset Meshes,” AIAA 2023-3602, June 2023. <https://doi.org/10.2514/6.2023-3602>, URL <https://arc.aiaa.org/doi/abs/10.2514/6.2023-3602>.
- [10] Chaderjian, N. M., “A Quantitative Approach for the Accurate CFD Simulation of Hover in Turbulent Flow,” *11th International Conference on Computational Fluid Dynamics (ICCFD11)*, ICCFD11-097, Maui, HI, July 2022.
- [11] Derlaga, J. M., Jackson, C., and Buning, P. G., “A Parallel Symmetric Successive Overrelaxation Method for Overflow,” AIAA 2021-2747, 2021. <https://doi.org/10.2514/6.2021-2747>.
- [12] Ahmad, J. U., “Application of Rotor Disk Model in the Overflow CFD Code,” Tech. Rep. NASA/TM-2022-0005496, NASA Ames Research Center, 2022. URL <https://ntrs.nasa.gov/citations/20220005496>.
- [13] Quackenbush, T. R., Wachspress, D. A., Boschitsch, A. H., and Curbishley, T. B., “A Comprehensive Heirarchical Aeromechanics Rotorcraft Model (CHARM) for General Rotor/Surface Interaction,” Tech. rep., CDI Report 99-03, 1999.