21st Australasian Fluid Mechanics Conference
Adelaide, Australia
10-13 December 2018

Simulations of Tandem and Coaxial Rotors using a CFD-coupled Rotor Model

D. Linton[1], **R. Widjaja**[2] **and B. Thornber**[1]

[1]School of Aeronautical, Mechanical and Mechatronic Engineering
The University of Sydney, New South Wales 2006, Australia

[2]Helicopter Systems Effectiveness, Aerospace Division
Defence Science and Technology Group, Victoria 3207, Australia

Abstract

A new rotor model has been coupled to a computational fluid dynamics (CFD) solver to reduce the difficulty and expense of simulating helicopters with complex configurations or operating in complex environments. This model uses an unsteady lifting line representation for the rotor blades and a vortex filament representation of the near wake. There is two-way coupling between the model and the CFD solver: velocities from the CFD solution are used to obtain the blade loading which is imparted back to the CFD solver by introducing momentum source terms to the Navier-Stokes equations in the region of the blade. The key advantage of this approach to rotor simulation is the removal of the requirement for a body-fitted mesh resulting in reduced preprocessing and run time. Simulations of two different complex rotor configurations using the CFD-coupled model are presented in this paper. The first is the Boeing CH-47D tandem rotor and the second the Sikorsky X2 Technology Demonstrator coaxial rotor. The CH-47D simulations accurately reproduced the results of an experimental in ground effect outwash survey while the X2 simulation results compared well with simulations from other authors.

Introduction

CFD provides a number of different methods for analysing the aerodynamics and performance of helicopter rotors. These include potential flow methods, vortex particle methods, and finite volume Navier-Stokes methods. Simpler, less computationally expensive methods have been extensively used to simulate rotors, but in the last few decades finite volume methods have become more popular as the computational expense has become acceptable and the techniques for simulating rotating bodies in relative motion (overset meshes, mesh deformation) have matured. Despite great improvements in the range of problems that are accessible to grid-resolved finite volume CFD, the preprocessing time and computational expense remains prohibitive for many applications – particularly when a parameter sweep is required. Examples of such applications are: rotorcraft operations in complex environments such as urban canyons or from the flight deck of ships and oil platforms; complex rotorcraft configurations such as tandem or coaxial rotors; and concurrent rotorcraft operations.

One simulation method, which retains the advantages of finite volume methods and reduces the expense, is to couple a rotor aerodynamics model with a CFD solver. In this approach the need for a body-fitted mesh for the rotor is removed by substituting an aerodynamic model for the rotor blades and using the CFD velocity field as the input to this model. The transfer of momentum from the rotor to the flowfield is achieved through the introduction of source terms to the Navier–Stokes equations in the vicinity of the blades or rotor disk. Different aerodynamic models may be employed (blade element momentum theory, lifting lines, or lifting surfaces) as well as different means of distributing the source terms. The family of CFD-coupled methods that arise from these choices are called (CFD-coupled) actuator disk[6], actuator line[10], and actuator surface methods[7]. In addition, these methods may employ either one- or two-way coupling between the solver and the aerodynamic model. The application of a newly developed CFD-coupled rotor model to the analysis of operations of complex rotorcraft configurations is presented in this paper. This new model belongs to the category of actuator surface methods and includes the new features of an unsteady coupling method and a CFD-convected free wake. The cases presented here provide validation of this approach to complex rotor simulations and demonstrate the utility of the method.

The first test case replicates the experimental CH-47D outwash survey performed by NAVAIR and the US Army Cargo Helicopter Division[9]. In these experiments a vertically spaced anemometer array was used to measure the velocity field around a CH-47D tandem rotor helicopter hovering in ground effect. Measurements were taken over the full 360° of azimuth angles around the vehicle and at a range of radial distances. Validation against this data set is of particular relevance for the CFD-coupled rotor model as proposed applications include the study of rotor wakes and their interaction with other airwakes.

The second test case is a demonstration of the CFD-coupled rotor model applied to the coaxial rotor of the Sikorsky X2 Technology Demonstrator. The X2 rotor is designed to enable high-speed flight by taking advantage of the aerodynamics of the coaxial configuration to reduce the tip speed at high advance ratios[1]. Other authors have performed simulations of the X2 rotor in forward flight using a rotorcraft comprehensive analysis code[5], blade-resolved CFD[5], and a vortex particle method[11]. The results of these simulations are used to validate the blade loading predictions of the present CFD-coupled rotor model.

Numerical Methods

CFD Solver

The OpenFOAM CFD code[12] was used to solve the incompressible Navier-Stokes equations (Eqs. 1 and 2) with the Improved Delayed Detached Eddy Simulation turbulence model of Shur et al.[8]. In Eqs. 1 and 2, $\boldsymbol{U}$ is the velocity, p the pressure, ν the kinematic viscosity, and $\boldsymbol{S}$ a source term. The Pressure Implicit with Separation of Operators (PISO) algorithm[2] was used to advance the solution in time and second order numerical schemes were used for the solution of all terms of the governing equations. Upwind biased schemes were used for convective terms and centred schemes for all others.

$$\frac{\mathrm{d}\boldsymbol{U}}{\mathrm{d}t} + (\boldsymbol{U} \cdot \nabla)\boldsymbol{U} = -\nabla p + \nu \nabla^2 \boldsymbol{U} + \boldsymbol{S} \tag{1}$$

$$\nabla \cdot \boldsymbol{U} = 0 \tag{2}$$

Rotor and Wake Model

The rotor model employed in these simulations consists of a lifting line representation of the blade and a vortex filament representation of the trailing near-wake circulation[4, 3]. Each blade is divided into a number of sections and the bound circulation, lift, and drag of each section is calculated via a 2D unsteady aerodynamics model. The input velocity to the 2D aerodynamics model is obtained by correcting the velocity sampled ahead of the blade from the CFD solution using the induced velocity from the bound and trailing circulation of the rotor model. The wake geometry is obtained by tracking the convection of particles shed from the edge of each blade section with the CFD flow field.

The blade loading from the rotor model is imparted to the CFD solution by adding momentum source terms to the Navier-Stokes equations ($\boldsymbol{S}$ in Eq. 1) in the location of the blade. The source terms are smoothed over the grid using a Gaussian distribution in the chord normal direction and a simple approximation of a typical airfoil loading distribution in the chordwise direction. This completes the two-way coupling between the rotor model and the CFD solver.

Test Cases

CH-47D Outwash Survey

Relevant parameters of the CH-47D rotors are given in Table 1. The target thrust coefficient is the average thrust coefficient per rotor reported by Silva and Riser[9] at the 41,000lb gross weight thrust condition. To obtain this coefficient of thrust in the simulations an existing trim solution for the CH-47D in hover out of ground effect was used initially and the collective was adjusted manually until the target thrust coefficient was obtained. A trimming routine has not been applied to the model so no other trim parameters were adjusted.

CH-47D Tandem Rotor		
Radius (m)	9.144	
Number of blades	3	
Longitudinal separation (m)	11.85	
Vertical separation (m)	1.423	
RPM	225	
Target thrust coeff. (ave. per rotor)	0.0158	
	Forward	Aft
Rotation direction (from above)	CCW	CW
Shaft incidence (°, +forward)	9.0	4.0

Table 1: CH-47D rotor parameters[9]. Separations and shaft incidence angles relative to fuselage reference line. Rotation directions: CCW – Counter-clockwise, CW – Clockwise.

The definition of the coordinate system used to describe the locations of measurements from the outwash survey is shown in Figure 1. The azimuthal and radial coordinates for points ahead of the forward rotor hub are relative to the forward hub, while points behind the aft rotor hub are referred to by coordinates relative to the aft hub. Radial coordinates for points lying between the two hubs are defined as the distance from a line joining the hubs. Azimuthal angles of 90°and 270°are defined in Figure 1. The hover position and orientation of the CH-47D is shown in Figure 2. The helicopter was hovered with rotor hubs level at 0.57D above ground level. This orientation places the fuselage reference line at 6.54° to the ground plane.

The rectangular domain for the CH-47D case extended 4.2 rotor radii from the ground plane in the vertical direction and 8.3 radii from the midpoint of the rotors to the lateral and longitudinal boundaries. The ground plane has a wall condition applied and the remaining boundary conditions are zero velocity gradient with a fixed total pressure if the flux is into the domain or a fixed pressure otherwise. The grid was refined towards the ground plane and around the rotor. Refinement was performed by splitting cell edges in half. Five such operations were performed on nested regions in the grid until the cells around the rotor were small enough to resolve the source term distributions. The resulting grid contained 3.6 million cells.

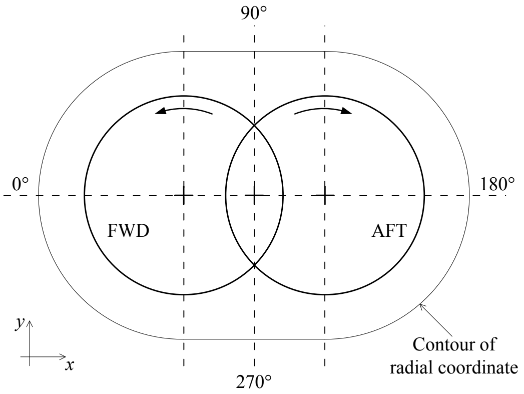

Figure 1: Definition of radial and azimuthal coordinates for the CH-47D outwash study

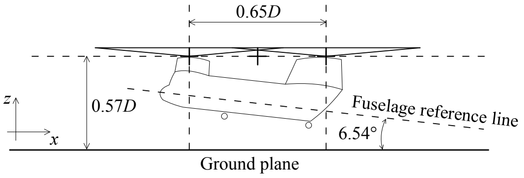

Figure 2: CH-47D hover position and orientation for outwash study

Sikorsky X2 Technology Demonstrator

Parameters for the Sikorsky X2 coaxial rotor are given in Table 2. The rotor is in low-speed forward flight at 55knots and the target thrust coefficient is from the trim solution obtained by Passe et al.[5] for this flight condition.

Sikorsky X2 Coaxial Rotor		
Radius (m)	4.023	
Number of blades	4	
Vertical separation (m)	0.443	
Shaft incidence (°, +forward)	0.0	
RPM	446	
Advance ratio	0.15	
Target thrust coeff. (ave. per rotor)	0.0143	
	Upper	Lower
Rotation direction (from above)	CCW	CW

Table 2: Sikorsky X2 Technology Demonstrator rotor parameters[5].

The rectangular domain for the X2 case extended 12 rotor radii from the hub to the aft boundary and 6 radii to all other boundaries. Freestream conditions were applied on all boundaries. The grid was refined around the rotor in the same manner as for the CH-47D case. Five levels of refinement were used for a total of 5.1 million cells.

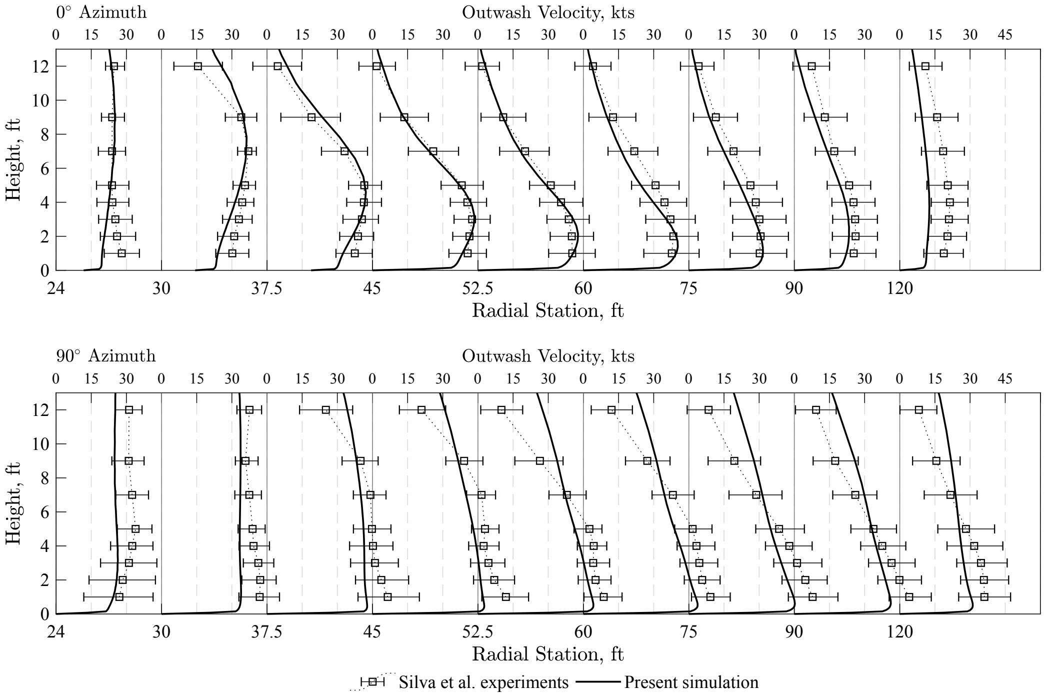

Figure 3: Outwash velocity along the 0°and 90°azimuth lines. Error bars indicate ±1 standard deviation.

Results

CH-47D Outwash Survey

The outwash velocity profiles at a range of radial stations are shown for the 0°and 90°azimuth lines in Fig. 3. Results from the present simulation are compared with the experimental data from the experiments of Silva and Riser[9]. The simulation results are similar in both trend and magnitude to the experimental data.

Sikorsky X2 Technology Demonstrator

A visualisation of the vortex wake structure behind the X2 rotors is shown in Fig. 4. This figure illustrates the distinctive structures formed by the trailing vorticity in the wake of the rotors. At the edges of the wake the vortices roll up into two counter-rotating streamwise vortices and there is noticeable asymmetry between the two sides. Note that the results shown in Fig. 4 were obtained on a mesh with additional refinement in the wake region.

Figure 5 shows the variation of normal force coefficient as a function of azimuth angle at two different radial stations (0.3R and 0.85R) for each rotor. Azimuth angle is expressed increasing counter-clockwise for both rotors. The results from the present method are compared to those from a body-fitted finite volume CFD simulation by Passe et al.[5] and a simulation using the vortex particle method by Tan et al.[11]. At the inboard station the present method shows good agreement with the blade-resolved CFD simulation for both the upper (CCW) and lower (CW) rotors. At the outboard station the present method is also in good agreement with the blade-resolved CFD and better agreement in terms of the location and relative magnitude of the loading peaks than the vortex particle method.

The computational expense of the present method was significantly less than that of the blade-resolved simulations. Passe et al. reported 9600 CPU hours on 240 processors for 8 revolutions while the present method required 227 CPU hours on 24 processors for the same period.

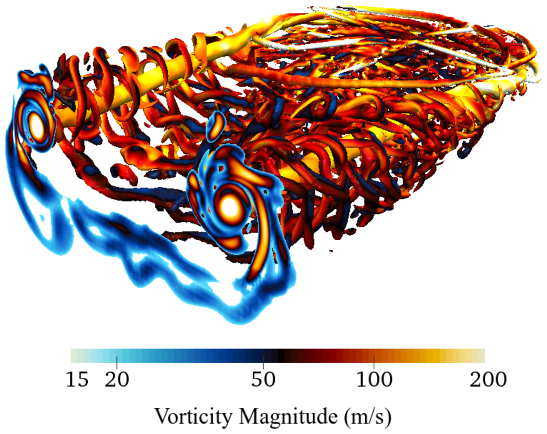

Figure 4: Contour of Q-criterion coloured by vorticity magnitude showing wake structure of the X2 coaxial rotor. Visualisation produced on finer, 12 million cell mesh.

Conclusions

A new CFD-coupled rotor model has been applied to two complex rotor simulations. The simulation of the CH-47D tandem rotor helicopter in hover while in ground effect produced outwash velocities which were in good agreement with those from the experiments of Silva et al.[9]. These results demonstrate the ability to predict velocities in the mid to far field from complex interactions of multiple rotor wakes in the presence of a

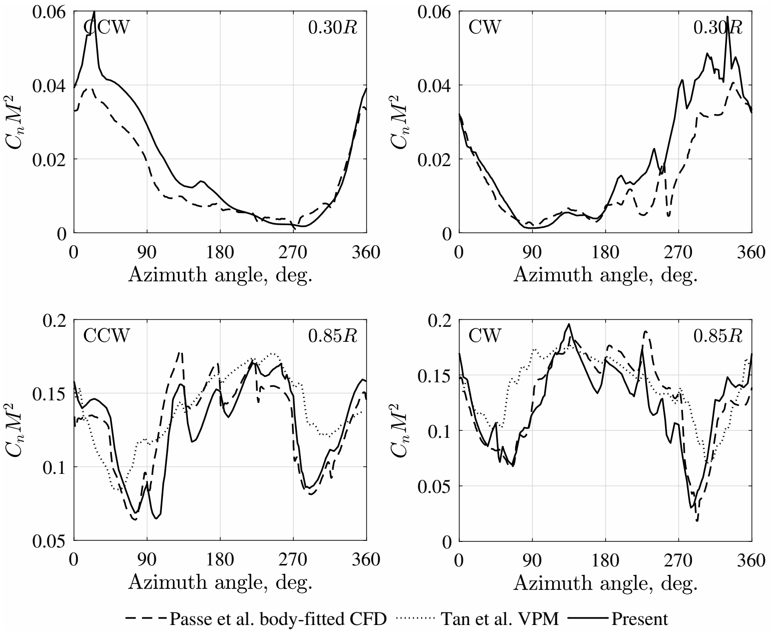

Figure 5: Normal force coefficient at two radial stations as a function of azimuthal angle for the X2 rotors

ground plane. The simulation of the Sikorsky X2 coaxial rotor in forward flight produced time-varying blade loads in good agreement with the blade-resolved CFD of Passe et al.[5]. This case demonstrated the application of the model to rotor systems with complex wake interactions as well as the ability to predict blade loads at a fraction of the computational cost of blade-resolved CFD.

The pre-processing time, including mesh generation, was significantly less than would be required for a simulation using a body-fitted grid. The simulation of the X2 coaxial rotor using the present method required only 2.4% of the CPU hours that the blade-resolved simulation of Passe et al. required. The CH-47D case illustrated how these computational savings make Navier-Stokes CFD simulations of operational scenarios, requiring tens or hundreds of rotor revolutions, more accessible. These savings are the principal advantage of the current method over simulations using body-fitted grids.

Acknowledgements

This work was supported and funded by the Defence Science and Technology Group. The authors acknowledge Artemis, the high performance computing facility at the University of Sydney.

References

[1] Bagai, A., Aerodynamic Design of the X2 Technology Demonstrator Main Rotor Blade, in *Proceedings of the 64th AHS International Forum*, AHS International, 2008.

[2] Issa, R., Solution of the implicitly discretised fluid flow equations by operator-splitting, *Journal of Computational Physics*, **62**, 1986, 40–65.

[3] Linton, D., Barakos, G., Widjaja, R. and Thornber, B., A New Actuator Surface Model with Improved Wake Model for CFD Simulations of Rotorcraft, in *Proceedings of the 73rd AHS International Forum*, AHS International, 2017.

[4] Linton, D., Barakos, G., Widjaja, R. and Thornber, B., Coupling of an Unsteady Aerodynamics Model with a Computational Fluid Dynamics Solver, *AIAA Journal*, **56**, 2018, 1–14.

[5] Passe, B., Sridharan, A. and Baeder, J., Computational Investigation of Coaxial Rotor Interactional Aerodynamics in Steady Forward Flight, in *Proceedings of the 33rd AIAA Applied Aerodynamics Conference*, 2016.

[6] Rajagopalan, R. and Sanjay, R., Three dimensional analysis of a rotor in forward flight, in *Proceedings of the 20th Fluid Dynamics, Plasma Dynamics and Lasers Conference*, American Institute of Aeronautics and Astronautics, 1989.

[7] Shen, W. Z., Zhang, J. H. and Sorensen, J. N., The Actuator Surface Model: A New Navier-Stokes Based Model for Rotor Computations, *Journal of Solar Energy Engineering*, **131**, 2009, 011002–011002.

[8] Shur, M., Spalart, P., Strelets, M. and Travin, A., A hybrid RANS-LES approach with delayed-DES and wall-modelled LES capabilities, *International Journal of Heat and Fluid Flow*, **29**.

[9] Silva, M. and Riser, R., CH-47D tandem rotor outwash survey, in *Proceedings of the 67th AHS International Forum*, AHS International, 2011.

[10] Sorensen, J. N. and Shen, W. Z., Numerical Modeling of Wind Turbine Wakes, *Journal of Fluids Engineering*, **124**, 2002, 393–399.

[11] Tan, J., Sun, Y. and Barakos, G., Simulation of Unsteady Aerodynamic Load for Rigid Coaxial Rotor in Forward Flight with Vortex Particle Method, in *Proceedings of the 43rd European Rotorcraft Forum*, 2017.

[12] Weller, H., Tabor, G., Jasak, H. and Fureby, C., A tensorial approach to computational continuum mechanics using object-oriented techniques, *Computers in Physics*, **12**.