Using CFD to Improve the WHL Rotor Load Aerodynamic Model

J. Beedy[1], G. Barakos, K. Badcock and B. Richards

CFD Group
Department of Aerospace Engineering
University *of* Glasgow
Glasgow G128QQ
www.aero.gla.ac.uk/Research/CFD

In current practice, the design of helicopter rotors is based on reduced aerodynamic models calibrated against experimental data while the use of computational fluid dynamics (CFD) by the industry is still restricted. In recent years, however, progress in CFD methods combined with the rapid increase of computing power has made a significant impact on the analysis and understanding of unsteady flow phenomenon. It is now possible to perform a complete analysis of a rotor using the Navier-Stokes equations and appropriate turbulence modelling. It is equally possible to couple the fluid-flow analysis with a structural model of the rotor in order to model the complete dynamics of the system. Despite this progress, two issues remain concerning the adoption of CFD by the industry as a design tool. Firstly, the CPU time required to run full unsteady calculations is still prohibitive to industry, and secondly, there is still much validation needed of the numerical and turbulence modelling schemes employed in CFD.

The aim of the present research is to bring CFD methods to industry without replacing directly the design techniques currently employed. It also offers a new approach in the validation of the CFD codes currently available at the CFD lab of the University of Glasgow. The key idea is to use a method other then direct comparison with experimental data to show that CFD is capable of predicting accurately the flow phenomena encountered in the rotor environment. This validation method aims at using CFD to improve the reduced order aerodynamic models in current industrial practice. The method also involves using flight simulation tools coupled with the improved aerodynamic models in order to make comparisons against flight test data.

The objectives of the research program are: (a) To validate the CFD code PMB [1] for turbulent, unsteady flows around the tips of helicopter blades. (b) To perform parametric studies on the Reynolds and Mach numbers, the geometry of the rotor and its motion. (c) To extract understanding out of the CFD calculations and the available experiments. (d) Exploit this information to modify the current rotor aerodynamic load code employed by Westland Helicopters Ltd (WHL). (e) To validate all modifications directly, against flight test data using the helicopter flight simulation code RASCAL[6].

Starting with the direct validation of CFD against experiments it is evident that experimental investigations for rotor flows are currently restricted to low Reynolds and Mach numbers, and simple geometries while the main quantity measured is the surface pressure distribution. Furthermore, most of the CFD investigations of flows pertinent to rotorcraft blades has been conducted in two-dimensions [2]. Recently, experimentalists have employed field methods like Laser Doppler Velocimetry (LDV) and triple hot-film probes to obtain detailed information about the velocity field within the tip vortex of unsteady laminar and turbulent flows. The work of Chang *et al.* [5] is used in the present research program to provide direct validation of the PMB code. This is a relatively easy case with a low Reynolds number so there is no need for turbulence modelling. Moving to higher Reynolds numbers, the experiments by Ramaprian *et al.* [7] were used. The problem studied in both cases is the behaviour of the tip vortex as it

[1]Ph.D Candidate, email: jbeedy@aero.gla.ac.uk

evolves from the tip of an oscillating square wing. Of significant importance is the trajectory of the vortex as well as the relationship between the motion of the vortex and the motion of the wing. The blocking topology and mesh distribution for the wing used for this study are shown in Figure 1. Indicative results for the tip vortex behind a non-oscillating rectangular wing are presented in Figure 2. About 1 million points have been used to adequately resolve the flow field and for the high Reynolds number cases a two-equation turbulence model has been employed. Further CFD results for the oscillating cases for the rectangular wing, corresponding to the further work of Ramaprian *et al.* [8], indicate that the position of the vortex follows the motion of the wing at a fixed phase angle. This finding is in good agreement with the experimental data. Several cases have been investigated and parametric studies have been performed for this complex flow. Detailed comparisons with the experimental data for both the laminar and turbulent cases are to be presented in the final paper.

Moving to the reduced aerodynamic models, two main approached have been identified. The first one is currently in practice with WHL and is based on the aerodynamic model developed by Beddoes [3, 4]. The second one has been developed by ONERA to predict the effects of dynamic stall on aerofoils [9]. A comprehensive comparison of both reduced models against data for dynamic stall cases will be presented to highlight the merits and shortcomings of each approach. Indicative results of these comparisons are presented in Figure 3. As can be seen the ONERA model is able to represent most of the characteristics of the C_L loop measured during experiments. The Beddoes' model was also found able to fit the azimuthal variation of the C_L and C_M coefficients.

References:

1. Badcock, K., Cantariti, F. and Gribben, B. Theory guide to Pmb2D Version 3.1 *Aerospace Engineering Department Report* University of Glasgow, June 2000

2. Barakos, G. and Drikakis, D., Separated Turbulent Flows Manoeuvring Lifting Surfaces. *Phil. Trans. R. Soc. Lond. A: Mathematical Physical and Engineering Sciences*, Issue 15th, 358(1777):3279-3292, December 2000.

3. Beddoes, T. S., Representation of of Airfoil Behaviour. *Vertica*, **7**(2), pp. 183–97 1983.

4. Beddoes, T. S., Practical Computation of Unsteady Lift. *Vertica*, **8**(1), pp. 55–71 1984.

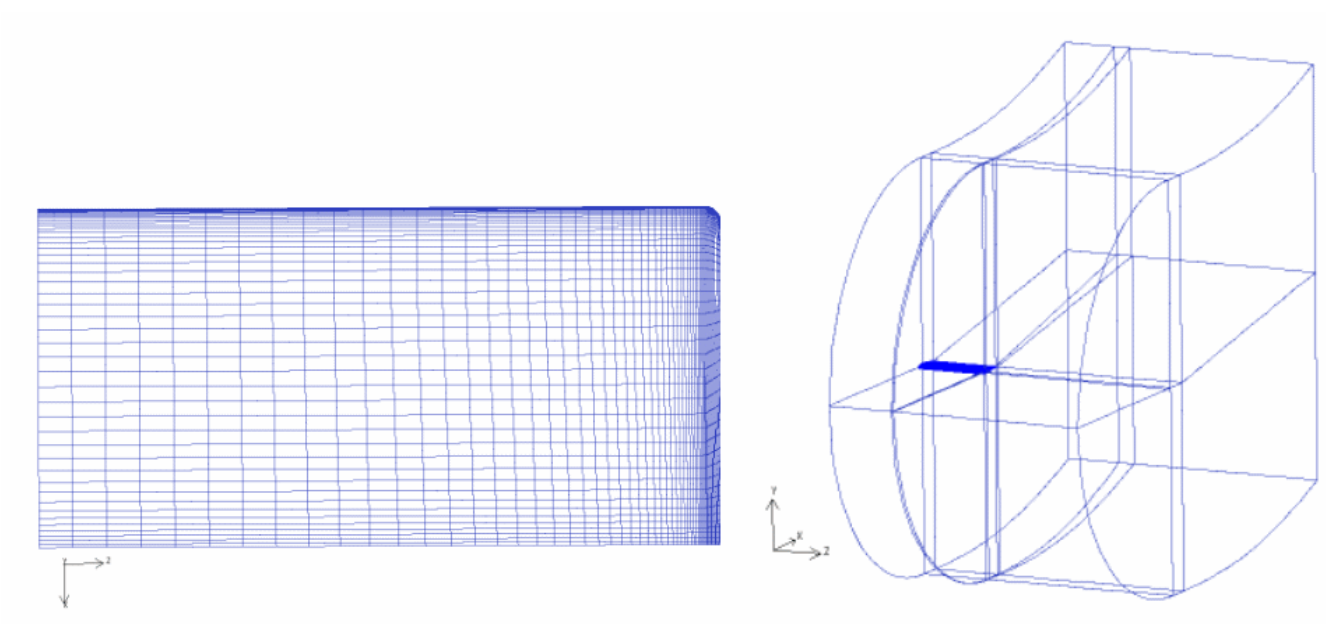
5. Chang, J. W. and Park, S. O., Measurements in the Tip Vortex Roll-Up Region of an Oscillating Wing. Technical Notes *AIAA Journal*, **38**(6), pp. 1092–95 2000.

6. Houston, S. S., Validation of a non-linear individual blade rotorcraft flight dynamics model using a perturbation method. *Aeronautical Journal*, **98**(977), pp. 260–66 1994.

7. Ramaprian, B. R. and Zheng, Y., Measurements in Rollup Region of the Tip Vortex from a Rectangular Wing. *AIAA Journal*, **35**(12), pp. 1837–43 1997.

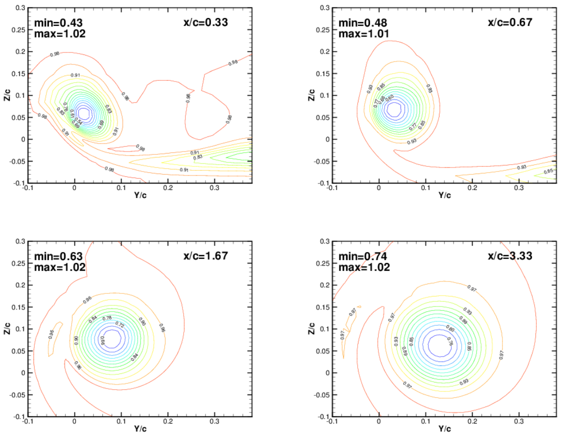
8. Ramaprian, B. R. and Zheng, Y., Near Field of the Tip Vortex Behind an Oscillating Rectangular Wing. *AIAA Journal*, **28**(1), pp. 73–94 1998.

9. Tran, C. T. and Petot, D., Semi-empirical model for the dynamic stall of aerofoils in view of application to the calculation of responses of a helicopter in forward flight. *Vertica*, **5**(1), pp. 35–53 1981.

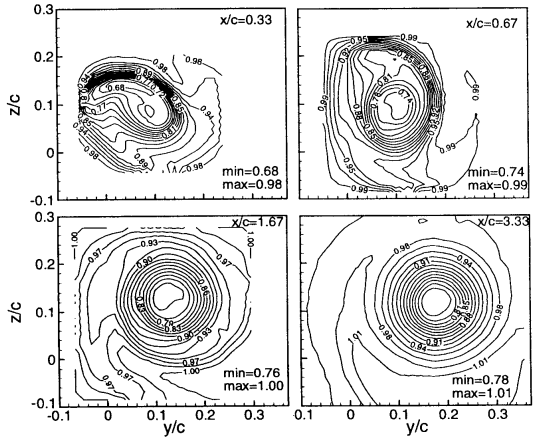


(a) Mesh Distribution (b) Blocking Topology

Figure 1: Mesh distribution and blocking topology for NACA 0015 square wing used for CFD calculations

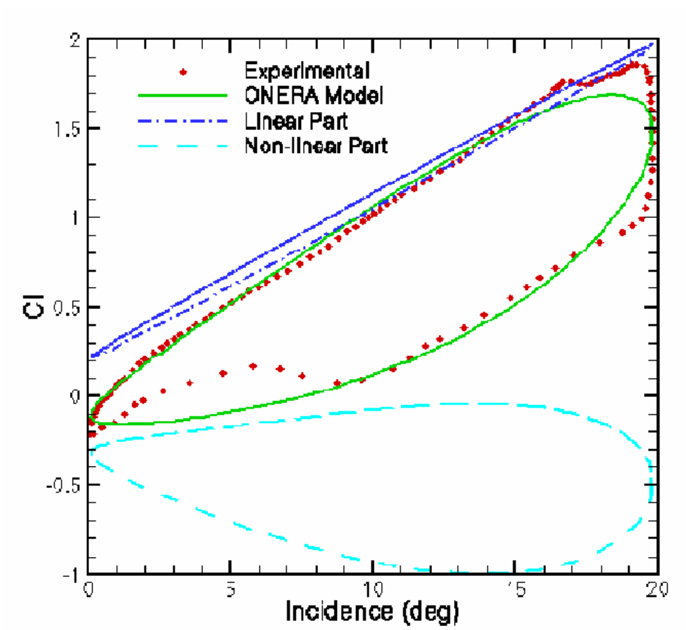


(a) Numerical

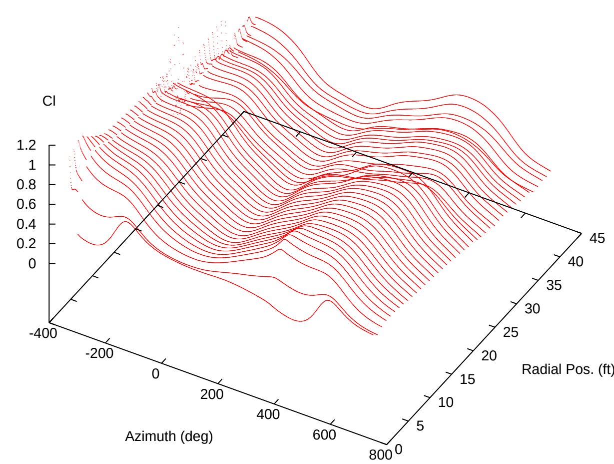


(b) Experimental

Figure 2: Contours of longitudinal velocity $\frac{U}{U_\infty}$ within tip vortex for an incidence of 10 deg. Mach number = 0.15, Reynolds number = 1.8×10^5. Positions correspond to distances behind the trailing edge in chordlengths. Origin corresponds to tip of trailing edge.



(a) ONERA Model



(b) Beddoes Model

Figure 3: Onera Model and Beddoes Model