CFD MODELLING OF WIND TURBINE AERODYNAMICS

Niels N. Sørensen

Professor MSO, Department of Civil Engineering, Alborg University &
Wind Energy Department, Risø National Laboratory, Technical University of Denmark
Frederiksborgvej 399, P.O. Box 49 DK-4000 Roskilde
Denmark

Foreword

Most of the examples shown in the following are based on work performed together with my good colleagues at the Wind Energy Department Risø National Laboratory, and the Department of Fluid Mechanics at the Technical University of Denmark. Special thoughts go to Jess A. Michelsen, who passed away much too early in august 2005. Jess was responsible for many of the essential parts of the Risø/DTU common flow solver EllipSys.
The results presented in the present notes are heavily biased towards results generated by the author and his colleagues, mainly due to easy availability but still these are relatively representative for state of art in the field. So in this respect, the text is not meant to be a review article but more an illustration of the possibility of modern Navier-Stokes solvers within the field of wind turbine aerodynamics.

Introduction to computational rotor aerodynamics

In the following, the application of CFD (Computational Fluid Dynamics) to wind turbine aerodynamics will be described, and the definition of CFD as given by Anderson [1], as "the art of replacing the integrals of the partial derivatives (as the case may be) in these equations with discretized algebraic forms, which in turn are solved to obtain numbers of the flow field values at discrete points in time and space" is used. The main focus of the text will thus be finite volume or finite difference formulations of the Navier-Stokes equations, neglecting other important approaches such as the vortex line and panel methods.

Historically, the first applications of CFD to wings and rotor configurations were studied back in the late seventies and early eighties in connection with airplane wings and helicopter rotors, [2, 3, 4, 5, 6] using potential flow solvers. The natural evolution towards solving unsteady Euler equations were seen through the eighties [7, 8, 9, 10], with the goal of removing some of the limitations of the potential flow solvers. Even though some problems connected to wind turbine rotor aerodynamics can be studied with Euler solvers, the fact that flow separation is nearly always present on some part of the rotor limits the application to very few cases. In the late eighties and early nineties, the computing power allowed the first solutions of the full Reynolds Averaged Navier-Stokes equations, and the first investigations of helicopter rotor aerodynamics appeared in literature, [11, 12, 13, 14].

In the late nineties, with the increased availability of computer resources and CFD solvers capable of handling viscous flow around rotors, application to wind turbine rotors became of practical interest. The first full Navier-Stokes computations of rotor aerodynamics was reported in the literature in the late nineties [15,16,17, 18, 19]. The European effort to apply Navier-Stokes solvers to rotor aerodynamics had been made possible through a series of national and European projects through the nineties. The European projects dealing with development and application of Navier-Stokes methods to wind turbine rotor flows was the Viscous Effects on Wind turbine Blades (VISCWIND) from 1995-1997 [20], Viscous and Aeroelastic effects on Wind Turbine Blades, (VISCEL), 1998-2000 [21, 22], and Wind Turbine Blade Aerodynamics and Aeroleasticity: Closing Knowledge Gaps, (KNOW-BLADE), 2002-2004, [23, 24, 25, 26, 27].

Approaches

As a consequence of the origin of most CFD rotor codes in the aerospace industry and related research, many existing codes are solving the compressible Navier-Stokes equations and are intended for high-speed aerodynamics in the subsonic and transonic regime, [28, 29, 30, 31, 32]. For helicopter applications, where compressibility plays an important role, this is the natural choice. For wind turbine applications the choice is not as obvious, one reason being the very low mach numbers near the root of the rotor blades. As the flow here approaches the incompressible limit, it may be very difficult to solve the compressible flow equations. One remedy to this is the so-called preconditioning, that changes the eigenvalues of the system of the compressible flow equations by premultiplying the time derivatives by a matrix. On the other hand, the compressible solvers have many attractive features, among these the ease of implementation of overlapping and sliding meshes, application of high-order upwind schemes, and very well developed solution methods.

Another very popular method, especially in the US, is the Artificial Compressibility Method, [33, 34], where an artificial speed of sound is introduced to allow standard compressible solution methods and schemes to be applied. In case of transient computations sub-iterations are taken within each time step to enforce incompressibility [34]. The method has several attractive features: Among these a similar ease of implementation of overlapping grids as the compressible codes. Overlapping grids are a necessity, to solve rotor/stator problems that are present when the rotor, tower and nacelle are included in the computations. The main shortcoming of the method may be problems to enforce incompressibility in transient computations without the need for a considerable amount of sub-iterations, and the problem of determining the optimum artificial compressibility parameter.

Due to the low Mach number encountered in wind turbine aerodynamics, an obvious choice is the incompressible Navier-Stokes equations. These methods are generally based on treating pressure as a primary variable [35, 36, 37]. Extensions to general curvilinear coordinates can be made along the lines of [38]. The method is not as easily extended to overlapping grids as the compressible and the artificial compressibility methods, due to the elliptical pressure correction equation. However, the method is well suited for solving the nearly incompressible problems often experienced in connection with wind energy. For steady state problems, the method can be accelerated using local time stepping, while the method using global time stepping still is well suited for transient computations.