DESIGN OF AIRFOILS FOR WIND TURBINE BLADES

V. PAREZANOVIC, B. RASUO, M. ADZIC***

University of Belgrade, Belgrade, Serbia, vparez@yahoo.com
**University of Belgrade, Belgrade, Serbia,* brasuo@mas.bg.ac.yu
***University of Belgrade, Belgrade, Serbia,* madzic@mas.bg.ac.yu

ABSTRACT

The main point of this paper is the design of the airfoils that could increase the overall efficiency of wind turbines. Every aerodynamic surface must undergo an expensive process of testing of its performance in a wind tunnel. The objective of this paper will be to show that we can minimize the need for costly experiments by introducing modern information technologies into the design process.

Using a commercial fluid dynamics solver such as Fluent, we will calculate the performance of several existing airfoils, which are frequently used in wind turbines. The simulated conditions will correspond to those found in a typical environment of a working wind turbine. Fluent and Xfoil simulation results will be compared with experimental wind tunnel data. Our goal is to achieve a high level of agreement between our results and experimental data, which will enable us to modify current, or design new airfoils with greater efficiency for use in wind turbines.

Keywords: Airfoil design, Wind turbine blades, Fluent, CFD

Introduction

The most important aspect of wind turbines is their aerodynamic effectiveness, the base of which is the design of the airfoils forming the blades. This paper is an introduction to a much more ambitious project of new airfoil design for use in wind turbines. Nevertheless, as presented here, it can show that it is possible to predict airfoil performance by using commercial CFD programs, and furthermore, to design new airfoils with better performance, based on those predictions.

This paper focuses on the use of non-linear solver Fluent, which employs the Finite-Volume Method (FEM). Results obtained through the use of Fluent are compared to experimental data from various wind tunnel measurements [1, 2, 3], and to results obtained from XFoil. XFoil is a panel method, linear equation solver, developed by Professor M. Drela at Massachusetts Institute of Technology (MIT) [4]. Results from XFoil, presented here, were obtained from Riso National

Laboratory, Denmark [5]. XFoil was used with 120 panels distributed on the airfoil surface; with activated viscous boundary layer and wake options, and with Orr-Sommerfeld transition criterion to simulate free boundary layer transition.

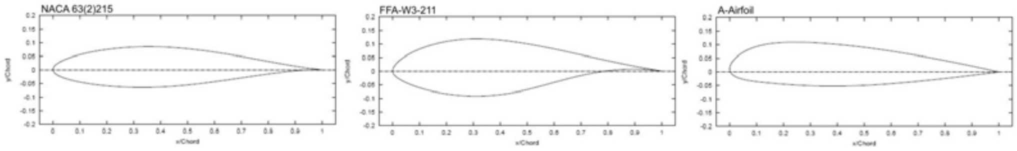

Fig. 1. *Airfoil geometries*

NACA 63(2)215

This is an airfoil from the 6th series of NACA laminar wing section family (fig.1.). Maximum relative thickness is 15%, located at 35% of the chord length. The experimental data was measured in low-turbulence pressure tunnel at NASA[1].

Computational mesh was done in Fluent's mesh tool Gambit. The resolution of the mesh is greater in regions where greater computational accuracy was needed, such as the region of the leading edge and the trailing edge wake. The mesh consists of 11970 quadrilateral cells, of which 146 is on the airfoil.

Reynolds number for the experiments and simulations is $Re=3x10^6$, and turbulence intensity is 0.07%.

A fully turbulent flow solution was used in Fluent, where k-w SST model was used for turbulent viscosity. Calculations were done for the "linear" region, i.e. for angles of attack ranging from -2 to 6 degrees, due to greater reliability of both experimental and computed values in this region.

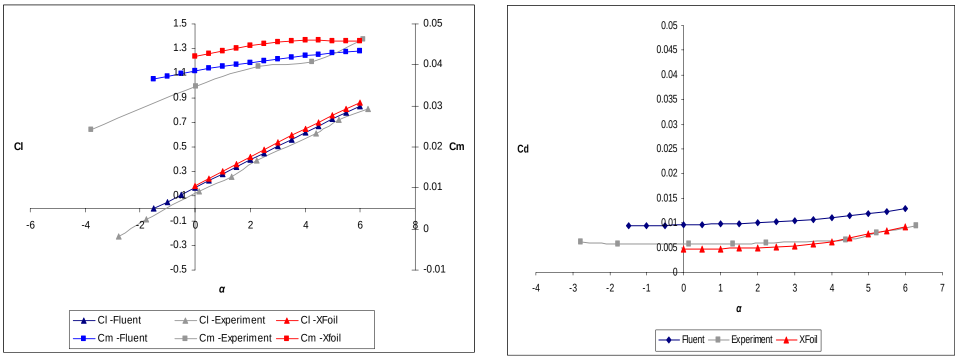

Fig. 2. *Lift and pitching moment coeff. curves (left), Drag coefficient curve (right) NACA63(2)215*

The results obtained from Fluent calculations agree very well with experimental data in regard of lift and pitching moment coefficient values (fig.2.). However, there are some discrepancies in the drag coefficient values. These discrepancies arise from modelling of boundary layer in Fluent. Since no data was available to indicate where laminar boundary layer transforms into a turbulent layer, computations were performed under the assumption that the boundary layer is fully turbulent. This problem is identified, and can be resolved in the future by compiling user-defined functions in Fluent, which would be used to predict transition effects.

FFA-W3-211

The FFA-W3-211 airfoil has been designed at The Aeronautical Research Institute of Sweden. It is a 21% thickness airfoil (fig.1.).

The computational mesh consists of 12240 quadrilateral cells, of which 156 are on the airfoil. Cell face areas range from 0.002m^2 to 1.6m^2. Gambit was used as the meshing tool.

The Reynolds number of the computation and wind tunnel measurements[2] was Re=1.8x10^6, with a turbulence intensity of 0.15%.

Again, k-w SST model was used in Fluent for turbulent viscosity, and the whole boundary layer is regarded as turbulent.

In the case of this airfoil, all results are in good agreement with experimental data (fig.3.). Although no laminar boundary region is simulated in Fluent, both lift and drag coefficient values are very close to experimental values. Drag coefficient is usually much harder to measure and much more sensitive to skin friction and other factors, but in this case it is indicated that the airfoil's performance is similar in both laminar and turbulent conditions, at least for low Reynolds numbers and small angles of attack.

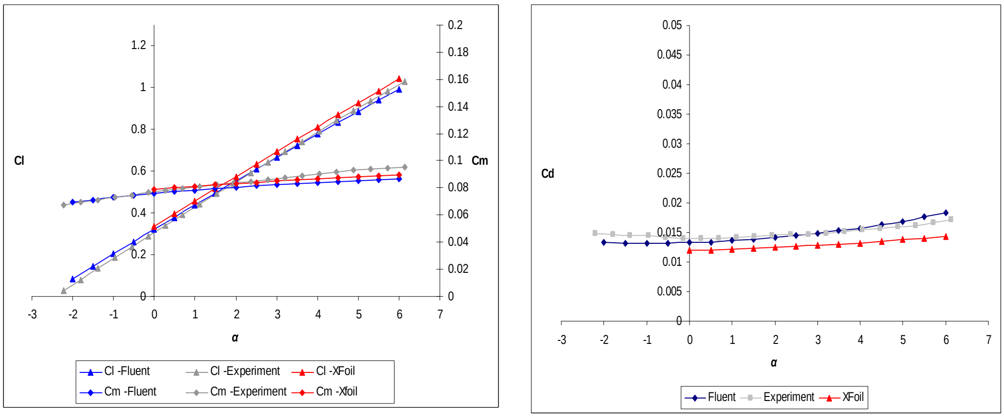

Fig. 3. *Lift and pitching moment coeff. curves (left), Drag coefficient curve (right) (FFA-W3-211)*

A-Airfoil

The Aerospatiale A-Airfoil is a 15% thickness airfoil (fig.1.). Wind tunnel measurements were carried out at ONERA/FAUGA[3]. The Reynolds number of the experiments and simulations is Re=2.1×10^6, with turbulence intensity of 0.07%.

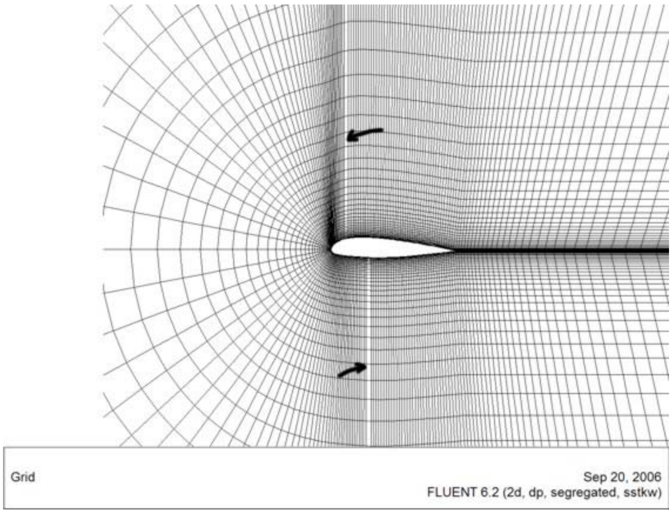

Fig. 4. *Mesh around the A-Airfoil (note the boundary between laminar and turbulent fluid zones)*

The wind tunnel experiments showed that the transition occurred at 12% of the chord length, on the upper side of the airfoil. Also, the transition point on the lower side was fixed to 30% of the chord length.

Since the locations of transition from laminar to turbulent boundary layers were available, this airfoil was chosen to simulate transition effects. This was done by meshing laminar and turbulent fluid zones separately (fig.4.).

Meshing was done in Gambit, with 3600 quadrilateral cells in the laminar zone and 9000 cells in the turbulent zone, of which 160 cells were on the airfoil.

The model chosen in Fluent was still the k-w SST turbulence model by Menter, but forward fluid zone was manually set to simulate laminar flow conditions.

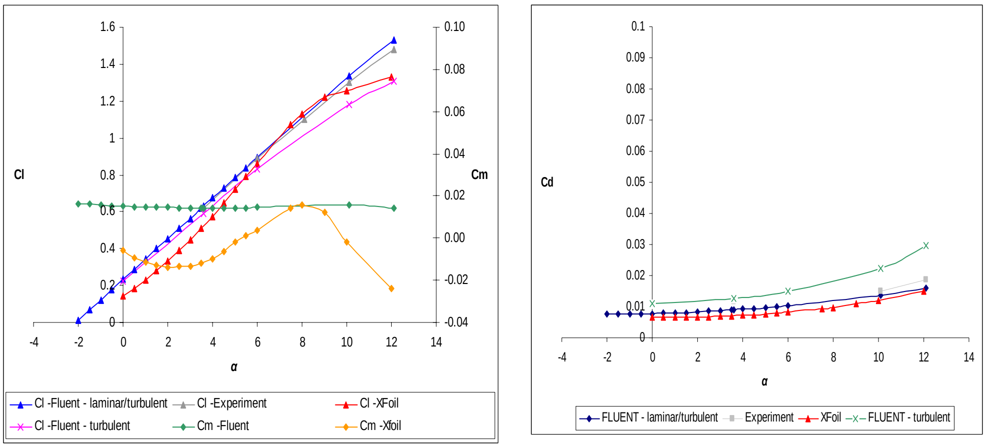

Fig. 5. *Lift and pitching moment coeff. curves (left), Drag coefficient curve (right) (A-Airfoil)*

Modelling of the laminar-turbulent transition allowed very good agreement between Fluent results and experimental data for both lift and drag coefficients (fig.5.). Unfortunately pitching moment data was not available from wind tunnel measurements so there was no base of comparison for this criteria. Also, this case was used to show the difference between using a fully turbulent model and simulating laminar-turbulent transition. Fully turbulent model simulations resulted in approximately up to 10% less lift and 30% more drag in the same conditions.

Conclusion

The simulations of these few airfoils yielded results which are, for the most part, in good agreement with available wind tunnel measurements. In those cases where they are not, the reason for this has been identified. In order to accurately simulate flow around airfoils at low Reynolds numbers as in cases investigated here, it is necessary to accurately simulate the effects of boundary level transition from laminar to turbulent conditions. This is not a major setback in cases where the location of transition is known, however, if we are going to modify or design new airfoils, which would be better suited for use on wind turbine blades, we must be able to accurately predict such occurrences. There are several models available, such as the transition model by R. Michel; this, coupled with Fluent's capabilities of using user-defined functions, could be a solution to the problems that transition effects impose.

References

[1] I.H. Abbott, A.E. von Doenhoff, L. Stivers, NACA Report No. 824 – Summary of Airfoil Data, National Advisory Committee for Aeronautics.

[2] A. Björck, A Guide to Data Files from Wind Tunnel Test of a FFA-W3-211 Airfoil at FFA, FFA-V-019, Flygtekniska Försöksanstalten, Sweden, 1996.

[3] ECARP -- European Computational Aerodynamics Research Project: Validation of CFD codes and Assessment of Turbulence Models, Notes on Numerical Fluid Mechanics, edited by W. Haase, E. Chaput, E. Elsholz, M.A. Leschziner and U.R. Müller, Vol.58, Vieweg Verlag, 1997.

[4] M. Drela, XFOIL: An Analysis and Design System for Low Reynolds Number Airfoils, Conference on Low Reynolds Number Airfoil Aerodynamics, University of Notre Dame, June 1989.

[5] F. Bertagnolio, N. Sorensen, J. Johansen, P. Fuglsang, Wind Turbine Airfoil Catalogue, Riso National Laboratory, Roskilde, Denmark, August 2001.