Calculations of Flow around an Airfoil with a Trailing Edge Flap by Use of an Immersed Boundary Method

Behrens, Tim; Zhu, Wei Jun; Shen, Wen Zhong; Sørensen, Jens Nørkær; Sørensen, Niels N.; Wedel-Heinen, Jens Jakob

Published in:
EWEC 2009 Proceedings online

Publication date:
2009

Document Version
Publisher's PDF, also known as Version of record

Calculations of Flow around an Airfoil with a Trailing Edge Flap by Use of an Immersed Boundary Method

Tim Behrens[1,4], Wei Jun Zhu[2], Wen Zhong Shen[2], Jens N. Sørensen[2], Niels N. Sørensen[3], Jens Jakob Wedel-Heinen[1]

[1]Vestas Wind Systems A/S, Global Research
[2]Technical University of Denmark, Department of Mechanical Engineering, Center of Fluid Dynamics
[3]Risø DTU National Laboratory for Sustainable Energy, Wind Energy Department

[4]Corresponding author: Frederiksborgvej 399, Building 115, 4000 Roskilde, Denmark, tibeh@vestas.com

Abstract

Designing smart rotor systems for wind turbines is a promising way to reduce loads and influence blade-stability. Current research focuses particularly on trailing edge flaps, in particular on their system integration, aero-elastic and aerodynamic behaviors. Complex flap configurations with moving geometries require special care when it comes to grid generation for aerodynamic calculations. Within the field of computational fluid dynamics the immersed boundary method is a numerical procedure in which forcing terms in the governing equations are used to impose boundary conditions. This gives more flexibility to treat moving geometries by using non body-fitted and stationary meshes.
An immersed boundary method with direct one-point forcing for the momentum equations and forced values for the turbulent scalars is implemented in the incompressible flow solver EllipSys2D. Calculations of flow around a NACA0012 airfoil with a 10% trailing edge flap are performed by solving the Reynolds averaged Navier-Stokes equations with a k-ω SST turbulence model at a Reynolds number of 1.000.000. The main part of the geometry without the flap is modeled with a body-fitted mesh, whereas the trailing edge flap is modeled with an immersed boundary inside a non-fitted mesh. The flap geometry was chosen to be smoothly curved. Unsteady calculations were carried out at different angles of attack and flap angles. It was shown that this method delivers good results when comparing the pressure distributions with those calculated on full body-fitted meshes.

Keywords: *ATEF, trailing edge flap, immersed boundary method, one-point direct forcing*

1. Introduction

For the future generation of wind turbines, the use of smart rotor systems with distributed aerodynamic actuators is a promising way to reduce loads and influence blade aero-elastic stability. This leads to benefits of bigger rotor diameters, closer turbine spacing or placement in complex terrains, etc.
Smart rotor systems are undergoing heavy researches at the moment [1, 2]. In Denmark the Adaptive Trailing Edge Flaps (ATEF) project sponsored by Danish High Technology Foundation is a research project in collaboration between *Vestas Wind Systems A/S* and *Technical University of Denmark* (Risø DTU & DTU Mechanics) covering research in system integration, aero-elasticity and aerodynamics of trailing edge flap systems.
Computations of the unsteady aerodynamics that occur at a wind turbine blade become especially challenging when moving flap geometries are included. The unsteady effects are mainly covered by empirical or semi-analytical models [3, 4] inside of aero-elastic codes. For a better understanding of the prevailing aerodynamic phenomena especially the three-dimensional flow field of blades equipped with trailing edge flaps in turbulent inflow is of interest. Usually such configurations are treated with moving or

overlapping/overset grids [5]. For complex and moving geometries, mesh generation might be hard to handle and unstructured meshes are often used. To keep the inherent benefits of structured meshes and a straight forward mesh generation, one may use fixed grids together with an immersed boundary method

2. Immersed Boundary Method

The immersed boundary method [6] was introduced as a way to simulate the flow inside the beating human heart in 1972 [5]. The body boundary conditions do not rely on the shape of the mesh but are introduced as forcing terms in the governing equations. This also makes the method very flexible for moving geometries.

A general procedure (Fig.1) for an immersed boundary method is to first identify and tag all cells with their cell-centers located inside the body domain. Afterwards all cells neighboring these inside-cells are identified and tagged accordingly so that the inner cells are completely surrounded. To represent the body, forcing terms have to be introduced in the governing equations. The incompressible Navier-Stokes equations read:

$$\rho \cdot \left(\frac{\partial \vec{u}}{\partial t} + \vec{u} \cdot \nabla \vec{u} \right) = -\nabla p + \mu \Delta \vec{u} + \vec{f} \qquad (1)$$

$$\nabla \cdot \vec{u} = 0 \qquad (2)$$

with *ρ, u, p and μ* being the density, the velocity vector, the pressure and the dynamic viscosity, respectively. The value of the forcing term *f* is calculated so that the boundary condition (e.g. no-slip condition) is fulfilled on the body. The general forcing term f in Eqn.1 is evaluated such that:

$$\vec{f}_P = -\vec{P}_P - \vec{S}_P + \sum_{i=w,e,s,n} a_i \vec{u}_i + \vec{u}_{IB} a_P \qquad (3)$$

where P_p and S_p denotes the pressure and body forces, the subscript $_P$ indicates the evaluation at the present cell and *i* is the index of the neighboring cells.

The forcing velocity u_{IB} at the immersed boundary is set to be zero in the case of a stationary immersed boundary passing through the cell center. In general the immersed boundary does not coincidence with the cell centers and interpolation procedures to the present cell center become necessary. In the following a linear/bilinear interpolation is used depending on the neighboring cells of a forcing cell.

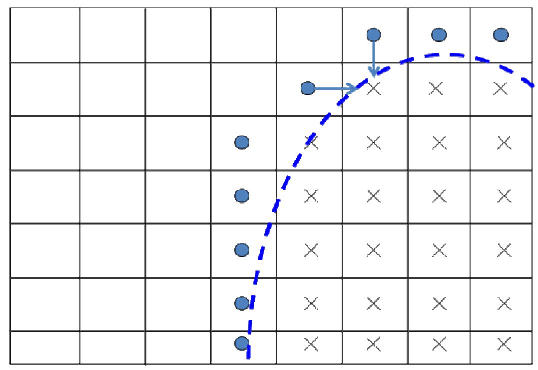

Figure 1: Immersed boundary example;
***x** inside cells, • forcing cells, -- body*

3. Implementation

The incompressible flow solver EllipSys2D was developed at Technical University of Denmark (DTU) and Risø-DTU National Laboratory for Sustainable Energy [8, 9, 10, 11, 12]. It uses domain decomposition and multi-grid methods to solve the incompressible governing equations.

Most applications of the immersed boundary method at high Reynolds numbers were using direct numerical simulation (DNS) or large-eddy simulation (LES). This work focuses on the Reynolds Averaged Navier-Stokes (RANS) equations for industrial relevant flows at high Reynolds numbers. The chosen turbulence closure is a k-ω SST model [13]. A first version of the EllipSys code for handling immersed boundaries can be found in [14]. A reviewed scheme using direct one-point forcing approach was implemented in the present work. The procedure at each time step is that first the grid cells that lie inside/outside of the fictive body are identified and tagged; afterwards forcing terms are applied to the momentum equations in such way as to fulfill the no-slip and impermeable wall condition at the stationary or moving fictive body surface.

The main challenge of applying the immersed boundary method to wind turbine blades is to treat turbulent flows at high Reynolds numbers. Here the treatment of turbulence models requires special attentions because of non-linear behaviors of the turbulent scalars close to the wall. While LES calculations can redirect this problem to the sub grid model, it is necessary for RANS models to apply additional boundary conditions,

$$\omega_{wall} = 10 \frac{6}{\rho \cdot \beta_1 \cdot n} \quad (4)$$

$$k_{wall} = 0 \quad (5)$$

These boundary conditions for the k-ω SST model are the wall values of the specific dissipation ω_{wall} and the turbulent kinetic energy k_{wall} (4, 5), where $ß_1$ is a model constant [13], ρ is the density and n is the calculated normal distance from the forcing cell center to the immersed boundary (Fig.2).

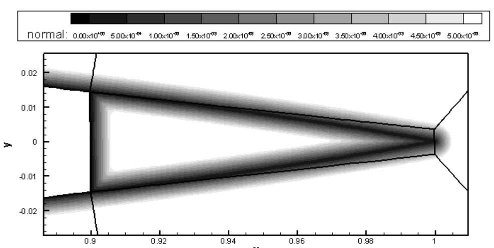

Figure 2: Computed normal distances

The flow field is then solved in the whole computational domain including all cells inside the fictive body boundaries, which of course is a drawback of this method but might be considered a plus for moving boundaries where fresh forcing cells can have appropriate initial values assigned.
The flow inside the immersed boundary may grow as time increases. In our current approach, the neighboring coefficients of each inside cell are set to zero, see Eqn. (3) such that flow is blocked from the outside of the immersed boundary and the velocity components are kept as zero.

4. Geometries

The airfoil used in the following has a symmetric NACA 0012 shape. The open trailing edge of the analytical airfoil shape is closed by a circular arc. The flap angle *ß* is defined as the angle between the line *s* – connecting the hinge point *H* with the trailing edge point - and the *x* axis.
It was shown that a smoothly curved trailing edge flap of 10% chord length is beneficial for control applications [15].

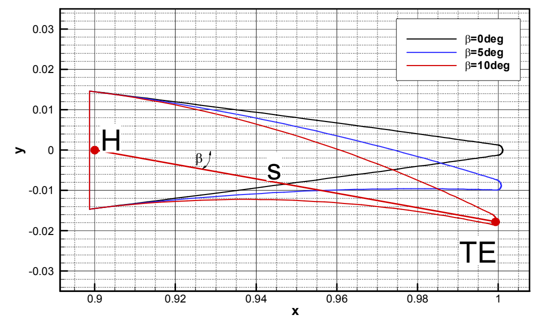

Figure 3: Flap geometries

Deflections in both coordinate directions at 90% chord are set to zero, while at the trailing edge the deflection is performed so that the length of *s* does not change. The distribution in between is then done by a square function. Resulting shapes for three different flap angles can be seen in Fig.3.

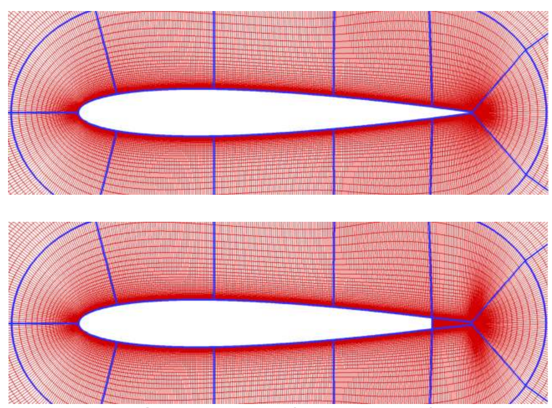

Figure 4: Grid setup: a) flap as body fitted grid (top); b) flap as immersed boundary (bottom)

Two different types of mesh are used. While the main part of the airfoil geometry is meshed by applying a body-fitted grid, the trailing edge flap was represented by an immersed boundary in a non body-fitted grid

block (Fig. 4b). The reference meshes for each flap angle are standard body-fitted meshes for the whole airfoil chord (Fig. 4a) and will be used for comparisons with the immersed boundary method.
The mesh consist of 22/23 blocks of 64*64 cells resulting in 90.112/94.208 cells inside the domain. The chord wise number of points is 320 per side for the body-fitted mesh Cells in normal direction are 128 with a far-field distance of greater than ten times the chord.

5. Results

In the present work calculations for unsteady flow at a Reynolds number of 1.000.000 were conducted. The QUICK and SIMPLE schemes were for differencing and pressure correction. Start time steps were set to a low value of Δt=0.1*10^{-3} low time step then higher one. The same time step for immersed boundary and reference calculations were used for comparison reasons. The number of sub iterations was varied between two and three.
Several fixed angles of attack were used and different fixed flap angles were investigated. The presented distributions of the pressure coefficient c_p are time averaged values of the flow after a settling period. At the body fitted part of the airfoil the wall pressure coefficient c_p is obtained from the pressure values in the cells next to the boundary while at the flap it is calculated from the pressure values in the forcing cells. The results of the immersed boundary calculations are presented in red, while body-fitted results for reference are presented in blue.

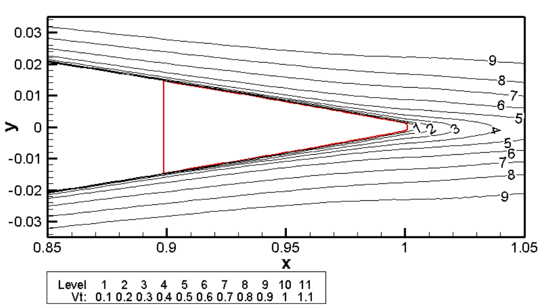

Figure 5: Total velocity for flow at α=0° and ß=0° where the immersed boundary is marked in red

At a flap angle of $\beta=0^o$ the body was well represented (Fig. 5) and good agreements with the pressure distribution on the body-fitted mesh were obtained for various angles of attack (Fig.6&7).
When comparing the pressure distributions at an angle of attack $\alpha=0^o$ and flap angles of $\beta=2.5^o$ and $\beta=5^o$ one can see that with bigger flapping angle the results show larger deviation from the body fitted ones (Fig. 8&9). The same holds for the configuration at $\alpha=5^o$ *and* $ß=-5^o$ (Fig. 10).
This deviation is caused by the flap now being in an area where the grid is too coarse and less aligned with the grid cells. Thereby the flap geometry is no longer captured as accurate as for zero flap angles (Fig. 11) and the off wall spacing is increasing. One can see streamlines passing in flow direction through the body. The hard bends in one of the streamlines indicates the actual body boundary the flow experiences.
Looking at Fig. 12 one can see that at a flap angle of $ß=5^o$ and an angle of attack $\alpha=5^o$ the pressure distributions fits well again. This indicates that the calculation is sensitive to the form of the pressure gradient along the flap when a coarser mesh is used.

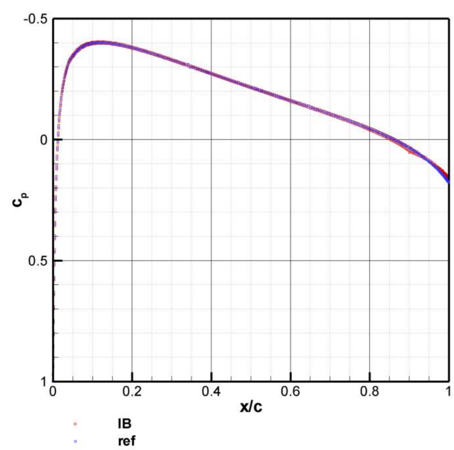

Figure 6: Pressure distribution c_p for flow at α=0°; ß=0°

6. Conclusion

The comparison of the airfoil pressure distributions calculated from the two methods showed good agreements and showed the need for a decent grid resolution. Full immersed boundary representations of an airfoil is still challenging at high Reynolds numbers. Based on the results we can conclude that the immersed boundary method is a promising tool for calculation of airfoils with flaps. This especially holds when thinking of applications to three dimensional geometries with moving flaps in complex configurations.

Acknowledgements

This work was supported by Danish High Technology Foundation (Højteknologifonden).

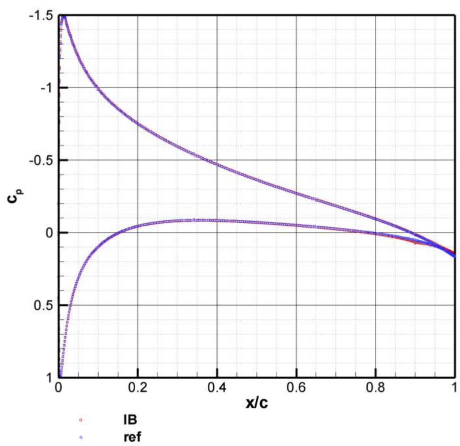

Figure 7: Pressure distribution c_p for flow at α=5°; ß=0°

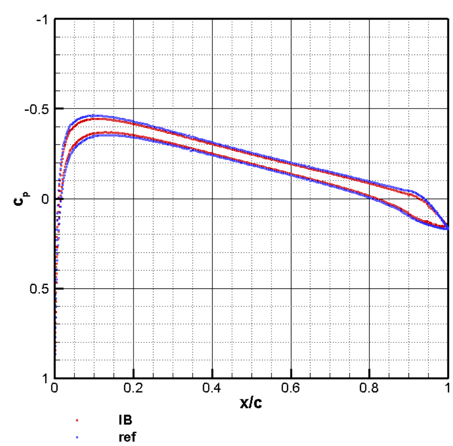

Figure 8: Pressure distribution c_p for flow at α=0°; ß=2.5°

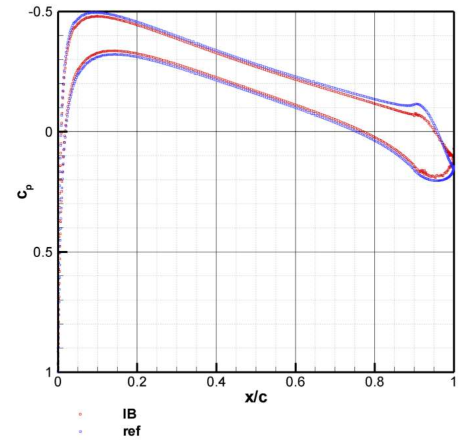

Figure 9: Pressure distribution c_p for flow at α=0°; ß=5°

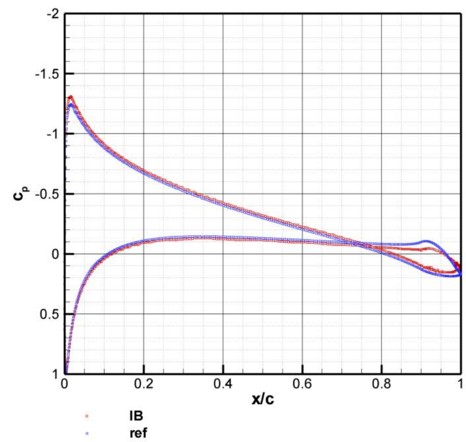

Figure 10: Pressure distribution c_p for flow at α=5°; ß=-5°

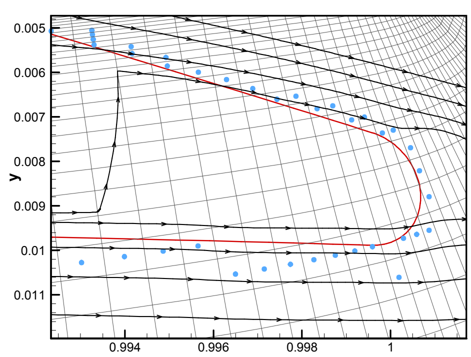

Figure 11: Body representation for flow at α=5°, ß=5°;
- immersed boundary; • forcing points

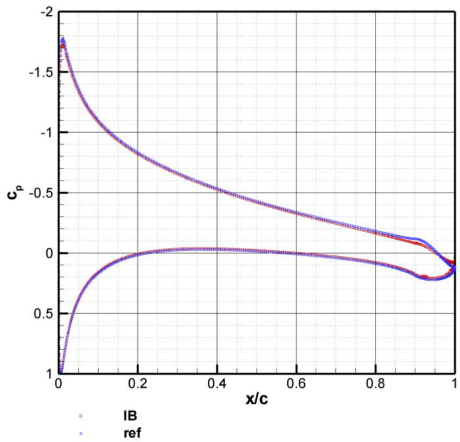

Figure 12: Pressure distribution c_p for flow at α=5°; ß=5°

Bibliography

1. Buhl T, Gaunaa M, Bak C. *Load reduction potential using airfoils with variable trailing edge geometry*, AIAA-2005-1183, 43rd AIAA Aerospace Sciences Meeting and Exhibit, Reno, Nevada, 2005
2. Barlas T, van Kuik GAM. *State of the art and prospectives of smart rotor control for wind turbines* J. of Physics, Conference series: The Science of Making Torque from Wind 75(012080), 2007'
3. Hansen MH, Gaunaa M, Madsen, HA. *A Beddoes-Leishman type dynamic stall model in state-space and indicial formulations*, Risø Report R-1354, Risø National Laboratory, 2004
4. Gaunaa M. *Unsteady 2D Potential-flow Forces on a Thin Variable Geometry Airfoil Undergoing Arbitrary Motion*, Risø Report R-1478, Risø National Laboratory, 2006
5. Heinz J. *Investigation of Piezoelectric Flaps for Load Alleviation Using CFD*, M.Sc. Thesis at Risø-DTU National Laboratory for Sustainable Energy, 2008
6. Mittal R, Iaccarino G. *Immersed Boundary Methods*, Annual Review of Fluid Mechanics 2005, pp239-261, 2005
7. Peskin CS. *Flow patterns around heart valves: A digital computer method for solving the equations of motion*, PhD Thesis, Physiology, Albert Einstein College of Medicine. Univ. Microfilms 72: 30-378, 1972
8. Michelsen JA. *Basis3D - a Platform for Development of Multiblock PDE Solvers*, Tech. Report AFM 92-05, Technical University of Denmark, 1992.
9. Sørensen NN. *General Purpose Flow Solver Applied to Flow over Hills*, Risø National Laboratory, 1995.
10. Shen WZ, Michelsen JA, Sørensen JN. *Improved Rhie-Chow interpolation for unsteady flow computations*, AIAA Journal 39(12)(2001) 2406
11. Shen WZ, Michelsen JA, Sørensen NN, Sørensen JN. *An improved SIMPLEC method for steady and unsteady flow computations*, Numerical Heat Transfer 43(3) (2003) 221
12. Michelsen JA. *Block Structured Multigrid Solution of 2D and 3D Elliptic PDEs*, Tech. Report AFM 94-06, Technical University of Denmark, 1994.
13. Sørensen NN, *k-omega Turbulence models implementation and testing*. Risø Report R-864, Risø National Laboratory, 1995

14. Reck M. *Computational fluid dynamics, with detached eddy simulation and the immersed boundary technique, applied to oscillating airfoils and vortex generators*, PhD Thesis at Technical University of Denmark, 2004
15. Troldborg N. *Computational Study of the Risø-B1-18 Airfoil Equipped with Actively Controlled Trailing Edge Flaps*, M.Sc. Thesis at Technical University of Denmark, 2004