LES MODELING OF WIND TURBINES: INTERACTION WITH TURBULENT FLOW

Paolo Schito[*a)] & Alberto Zasso[*]
[*]*Department of Mechanical Engineering, Politecnico di Milano,*
via La Masa 1, 20156 Milano, Italy

Summary A new approach to model the interaction of a wind turbine with the turbulent structures of the wind using the actuator line method in a Large Eddy Simulation (LES). In the paper will be presented the wind turbine model, some results for laminar flow compared with experimental data in order to validate the model and the interaction with a turbulent boundary layer in order to capture the effects of turbulence on the wind turbine performances.

INTRODUCTION

Energy is a key issue in modern society: the population is growing, asks for more energy and actual energy production processes will soon be unsustainable. Nuclear power was ascribed to be the best energy supply but recent events changed the perspectives: the solution is to exploit the so called renewable energy resources. Wind Power is one of the green resources which is growing faster in the last years, the investments are increasing and Wind Turbines are designed to generate up to 5MW of power. This is a challenge to engineers who must guarantee the safety of the structure maintaining a high efficiency as the size of the wind turbine becomes larger. Moreover the concept of Wind Farms, where whole cities can be provided with energy, gives new themes to research, especially in the interaction of the large rotors with the unsteady turbulent structures of the wind and the wakes of the upwind wind turbines.

NUMERICAL MODEL

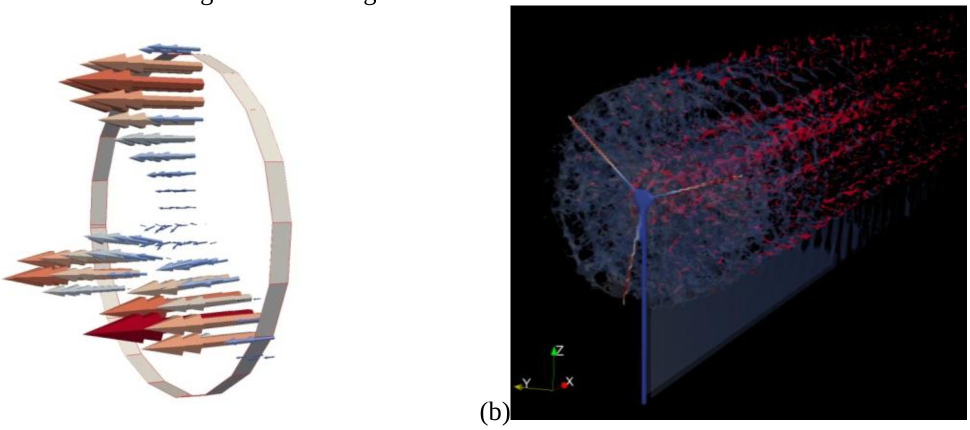
This research aims to provide to engineers new tools to locate and design new single turbines and wind farms, putting a sharp eye on the incoming flow and the wake modeling. The approach is a Large Eddy Simulation (LES) model, that is the more advanced method in simulating transient turbulent flows with complex geometries.
The model is developed in the OpenFOAM framework [3], that is the state-of-the-art in open-source Computational Fluid Mechanics solvers. The Wind Turbine model does not geometrically reproduce the machine, but introduces on the flow the aerodynamic forces acting on the turbine. This simplification is known as the “Actuator Line Method” (ALM) [2], where the forces are calculated using the aerodynamic characteristics of each section of the blade. This method is widely used in most CFD and multi-body codes to calculate the performances and the loads of a wind turbine.
An important part of this model is the definition of the effective velocity, that is responsible for the aerodynamic forces on the blades: this seems to be very simple but when a phenomenon is transient and the presence of the rotor himself affects the incoming velocity on the rotor plane this is anything than simple. Most aero-elastic codes and most CFD codes use Blade Element Momentum (BEM) to calculate the wind loads of the turbines: this is the only approach that multi-body analysis can support, while CFD can give sensible improvements to interaction of the wind turbine rotor with the turbulent structures that are present in the natural wind [1]. Most CFD codes use steady forces or estimate the aerodynamic loads using as reference velocity the wind speed at different locations than the rotor plane: this is a poor use of numerical fluid mechanic simulations since they provide full description of the flow field and this information must be used to attain a more refined model. A CFD local analysis has been carried out to determine the relation between the mean flow values near the wing profile and the reference nominal values that are used to report the aerodynamic characteristics of the airfoil. The relation between the local velocity and the nominal velocity has been defined using the mean value on a line across the rotor plane for a distance of the half of the rotor radius: this method gives mean results comparable with the more popular BEM method, but allows a more effective interaction with the turbulent structures of the flow.
Wind tunnel testing of wind turbines scaled models is a research field carried out at Politecnicio di Milano Wind Tunnel and the measurement of flow data as well as structural loads has been done in order to obtain experimental data in controlled environment to validate the numerical model. In this case the boundary conditions to be applied to the numerical model are very difficult to be achieved: the flow reaching the wind turbine is transient, with large flow turbulent structures interacting each other and finally hitting the rotor. The calculations for the reproduction of the turbulent flow have been conducted using many approaches, often very convenient from the computational point of view, but the only method giving satisfying results on the specific turbulent flow of the wind tunnel experiment is the full geometrical reproduction of the Wind Tunnel channel: this project [4] has obtained the funding for a two year project from Regione Lombardia and CILEA in order to set up a database of turbulent flows, providing space and time distributions of velocity and pressure.
The numerical solver developed is entirely written in C++, taking all advantages that the code offers, defining new classes and methods to describe the wind turbine aerodynamics. This code gives the opportunity to be refined in all parts, with the possibility to introduce the aero-elastic behaviour of the structure and the control of the wind turbine in

[a)] Corresponding author. Email: paolo.schito@mail.polimi.it.

order to simulate the complete full scale turbine features and behaviour. The validation of this code with a scaled model and a controlled environment opens the possibility to use the code to predict full scale performances of real machines.

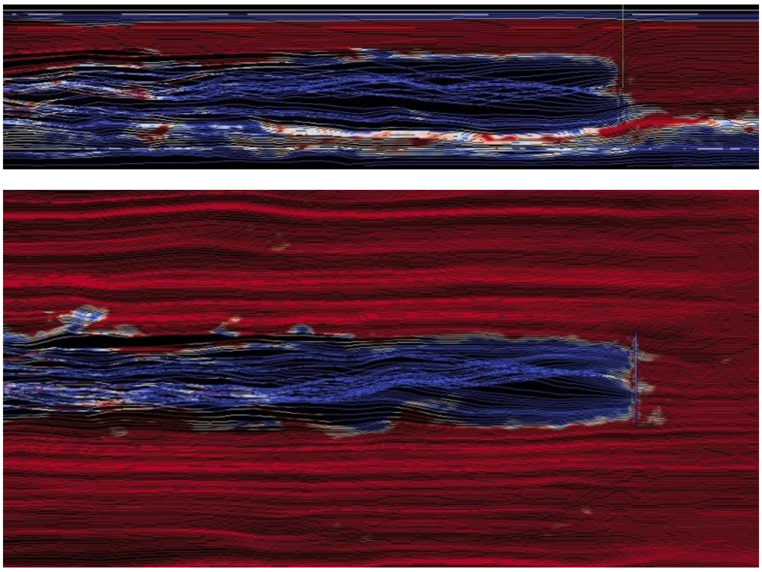
RESULTS

The forces are introduced in the flow field as can be seen in Figure 1-a, where the forces are introduced in the rotor area in the position of the blades, combining wind speed with rotational speed. In figure is possible to see also a typical force distribution on the blades for a given instant. The influence on the flow are very interesting, with the capability to capture main wind turbine flow characteristics, as can be the flow wake, the characteristic rotation of the flow, the tip and hub vortices that can be seen in Figure 1-b and Figure 2.



(a) (b)

Figure 1 - (a) Force Vectors colored by force magnitude that are acting on the flow in the rotor plane. As can be seen the forces are present only in the location of the blades - (b) Vorticity iso-surfaces in order to highlight the tip and hub vortices: it is possible to identify the position ofthe blades, the hub and the tower.



(a)

(b)

Figure 2 - Streamlines of velocity for a wind turbine in a turbulent flow colored by velocity magnitude. In (a) is represented a vertical plane, in (b) is reported an horizontal plane. It is possible to note the wind profile, the wake expansion and the rotation of the wake induced by the wind turbine power production.

CONCLUSIONS

This work gives to wind energy researches a state-of-the-art CFD tool able to reproduce the aerodynamic behavior of a wind turbine. This tool is open to further developments as can be the introduction of aeroelastic effects on the model as well as the test of control laws to ensure the model safety. This experimentally validated model against model software can be directly used to predict the performances of full scale wind turbines.

References

[1] Wu Y.-T., Porté-Agel. F.: Large-Eddy Simulation of Wind-Turbine Wakes: Evaluation of Turbine Parametrisations. *Boundary-Layer Meteorology*, **138**(3):345-366, 2011

[2] Sørensen JN, Shen WZ: Numerical modeling of wind turbine wakes: *J Fluids Eng* **124**:393–399, 2002

[3] OpenFOAM: web site http://www.openfoam.com

[4] Rocchi D., Schito P., Zasso A.: Investigation on the relation between incoming wind charactersistics and surface pressure distribution: *submitted for ICWE13 JWEIA Special Issue Selected Paper*, 2011