POLITECNICO DI MILANO

CFD for supporting renewable technology study: FLOW FIELD ANALYSIS AROUND FLANGED DIFFUSERS FOR SMALL-TYPE WIND TURBINE

Riccardo Mereu, Emanuela Colombo, Fabio Inzoli, Luca Carpinelli, Sara Orsini, Sara Reoletti, Maurizio Rota

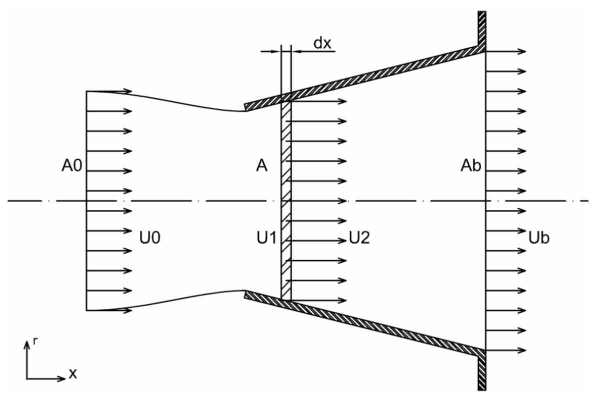
Politecnico di Milano, Dipartimento di Energia, via Lambruschini, 4 - 20156 Milano

Abstract

The European policy, expressed by the Strategic Energy Technology Plan, to reduce greenhouse gas emissions by 20% by 2020 and by 60-80% by 2050 requires a major diversification in the global energy mix. Among renewable energy technologies, wind turbines represent one of the most effective way to produce electrical power.
Wind power generation is proportional to the cube of the wind speed: a large increase in capacity may be obtained by creating even a slight increase in the velocity of the approaching wind. A possible approach to this issue is based on the local concentration of the wind energy, which can be achieved by surrounding the turbine with a flanged diffuser.
The present research is devoted to the contribution that a CFD analysis may give to the design of a wind turbine in order to increase the fluid dynamic performance by proposing a parametric study over the geometrical configuration of the diffuser.

Case Study

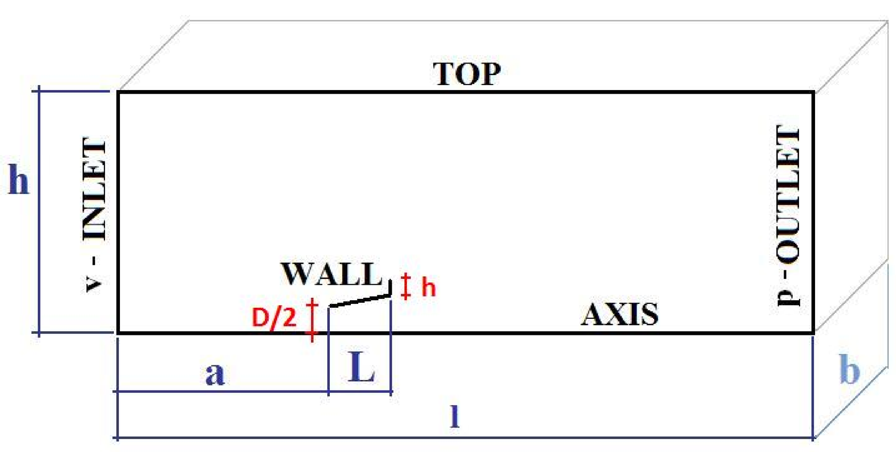
This research is focused on the analysis of flanged diffusers applied to small-type wind turbines under 1.5 kW, deriving from the experimental and numerical study of Sakuray et al. [1] and Ohya et al. [2].



Schematic representation of the studied wind turbine system

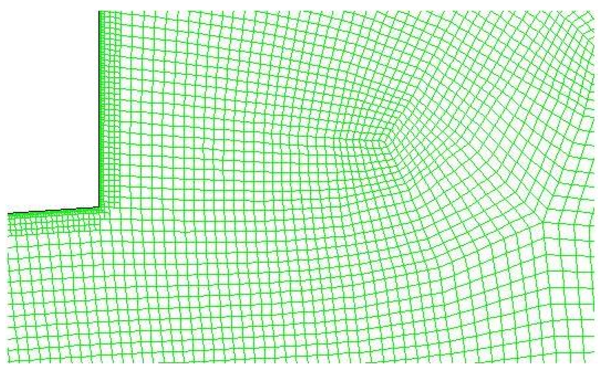
Numerical modelling

Two different computational domains (A and B) are considered for modeling the system. Domain A, based on Abe et al. [1] experimental test, has the dimensions of the wind tunnel where experiments were carried out. Domain B, based on Ohya et al. [2] numerical study, reproduces the computational alternative. No meaningful differences are obtained, and domain A was selected based on lower computational costs.



	Wind tunnel	Computational domain	
		A	B
h [m]	1	1	1.5
a [m]	n.a.	1	1
l [m]	15	3	3.3
L [m]	n.a.	0.3	0.3
b [m]	1.8	Axial-symmetric solver	

Different quadrilateral cell meshes are tested (coarse= 45,000 cells, medium_1= 90,000 cells, medium_2= 120,000 cells and fine= 300,000 cells).
From the coarse mesh the successive refinements are applied to the most significant regions of the domain close to the diffuser. A grid independency analysis is carried out up to a fine model (300,000 cells) demonstrating the grid-independency of the results for the adopted discretization (medium_2).
The near wall zone of the diffuser is adapted (y^+ close to 1) in order to use a two-layer approach.



Detail of the mesh near to the flange

The numerical simulation is performed with a segregated solver. A steady formulation of governing equations is applied on an axisymmetric domain. For the spatial discretization a Second Order Upwind method is used. The SIMPLE algorithm is used for pressure velocity coupling.
Different turbulence models (k-ε family) are compared in order to identify the most accurate setting for the numerical model. The results obtained with k-ε RNG turbulence model are here presented.
The Reynolds number referred to the inlet diameter of the diffuser is 67,000.
A flat velocity profile is imposed as inlet boundary condition, setting the turbulent intensity (3%) and the hydraulic diameter as input for turbulent conditions.

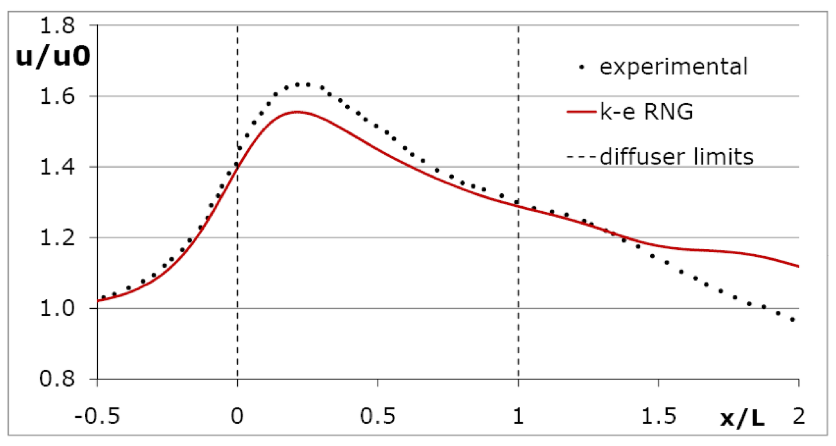
Inlet BC	Velocity 5 m/s Hydraulic diameter 2m Turbulent intensity 3%
Outlet BC	Gauge static pressure (0 Pa)
Top BC	No-slip wall / Symmetry
Bottom BC	Axis

Boundary conditions

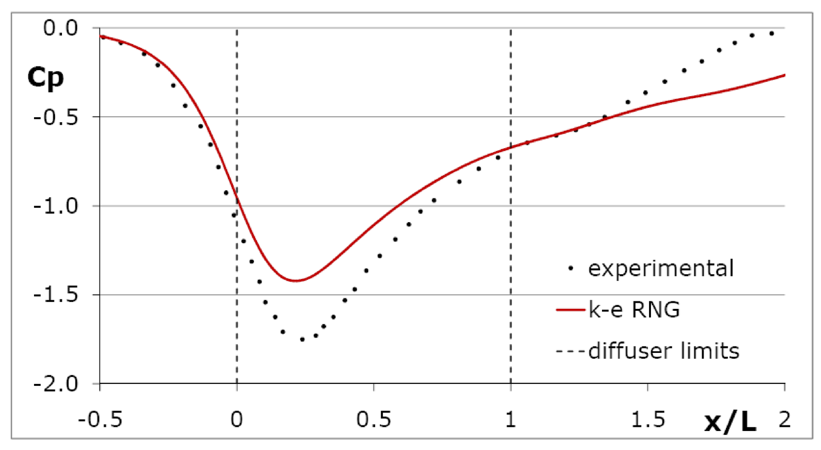
Results

In this work the influence of the turbine on the diffuser is studied. The turbine is modeled in the numerical domain with a lumped parameter model located at x/L=0.167. The load coefficient (C_t) is defined as: $C_t = \frac{2(p_1 - p_2)}{\rho U_2^2}$, where 1 and 2 refer to the pre- and post- load section respectively.
Different flange height are investigated, too.
The results obtained are based on the comparison of *dimensionless velocity* (U/U_0) and *pressure coefficient* $\left(C_p = 2(p-p_0)/(\rho U_0^2)\right)$ evaluated along the axis of the diffuser.

No Load case

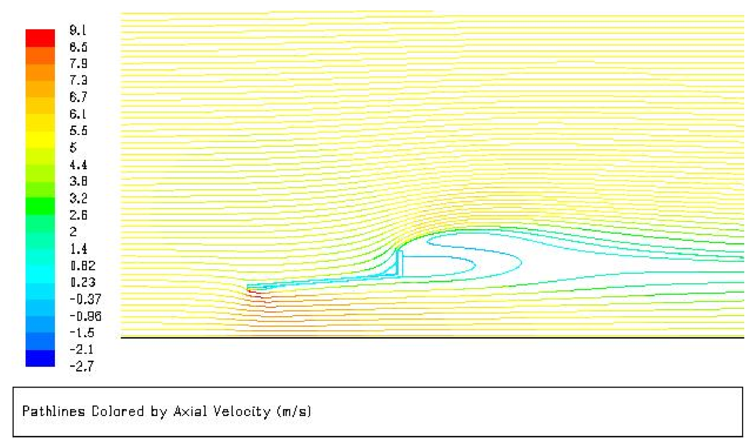


Velocity distributions on the central axis
(Re=67,000, h/D=0.25, C_t=0)

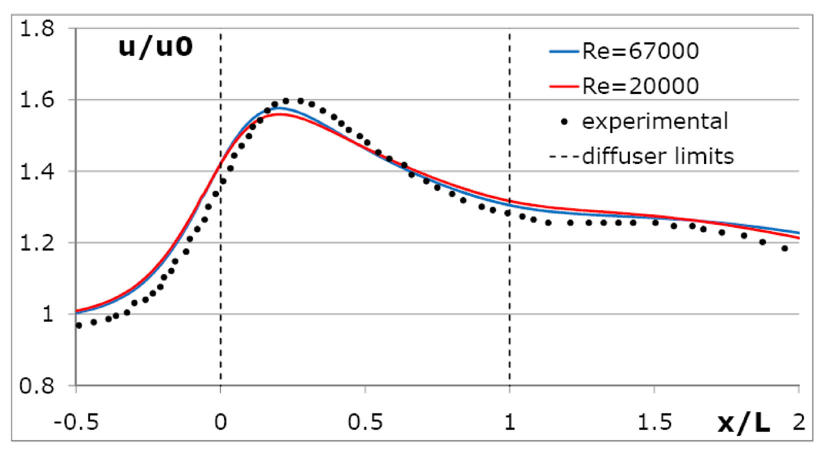


Static pressure distribution on the central axis
(Re=67,000, h/D=0.25, C_t=0)

The presence of separation and recirculation of the fluid downstream of the flange is shown in the figure.



streamlines velocity
k-ε RNG with EWT;
(Re=67,000; h/D=0.25; C_t=0)

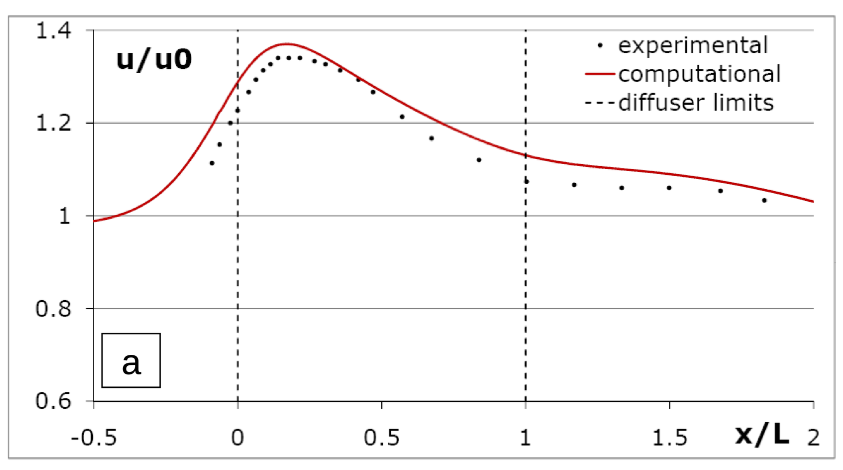


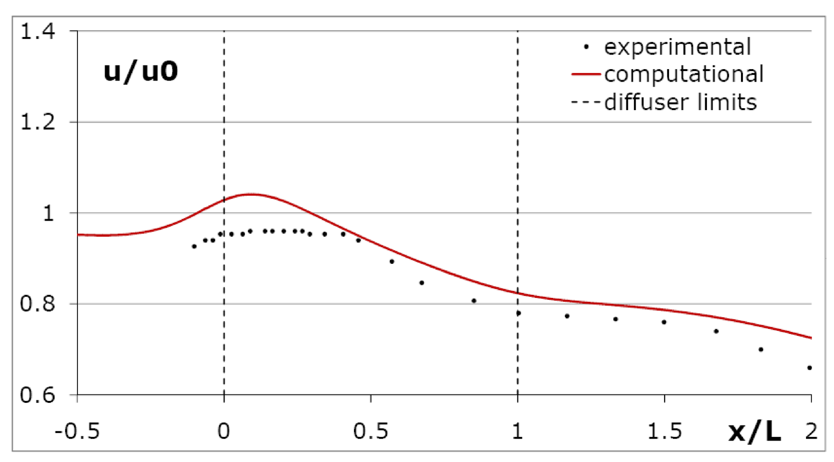
Reynolds number influence

velocity distribution
k – ε RNG with EWT;
(h/D = 0.25; C_t = 0)

Load case

As shown by the present study the diffuser drives the air flow with similar efficiency at different Reynolds number regimes.





Wind velocity distributions on the central axis
(Re=67,000; h/D=0.5) in case of C_t=0.2 (a) and C_t=0.9(b))

Conclusion

The agreement of the numerical results with experimental data provides the possibility of using the model for investigating new configurations, such as different flange lengths, diffuser opening angles and thickness. As a conclusion the study suggests that using the numerical approach to investigate different configurations obtained by varying geometry, load and work conditions may be an appropriate solution for evaluate the global performances, analyzing improvement while reducing cost and time of development compared to experimental campaign.

[1] A. Sakurai, K. Abe, M. Inoue, Y. Ohya, T. Karasudani. *Journal of Wind Engineering and Industrial* Aerodynamics 96, 2008, pp. 534-539.
[2] Y. Ohya, K. Abe. *Journal of Wind Engineering* and Industrial Aerodynamics, 2003.

riccardo.mereu@polimi.it
sara.orsini@mail.polimi.it
sara.reoletti@mail.polimi.it