Simplified Numerical Model of an Axial Impeller

Andrei-Mugur Georgescu[1], Sanda-Carmen Georgescu[2]

[1]Hydraulics and Environmental Protection Department, Technical University of Civil Engineering, Bucharest, Romania

[2]Hydraulics Department, University “Politehnica”, Bucharest, Romania

Abstract

Introduction: This paper proposes a simplified numerical method to model the flow field downstream of an axial impeller. This method can be used for any axial hydraulic machinery for which, one is less interested by the actual flow between the blades, than by the results of the flow field downstream of the hydraulic machinery. Use of COMSOL Multiphysics: "Simplified method" means that one doesn't have to model the actual blades of the axial hydraulic machinery and use a rotating mesh to numerically calculate the flow field downstream of the machinery, but rather to insert some force coefficients (variable with the flow rate through the hydraulic machinery) in a subdomain representing the impeller, and define some integration variable giving the flow rate at the inlet of the impeller subdomain for each computing iteration. The above force coefficients (i.e. body forces fields of the Navier-Stokes equations) will produce on the flow the same average effects as the blades of the impeller. The only needed data are the rotational speed of the hydraulic machinery and its head pressure-flow rate curve. There are many applications of such a "simplified model". Suppose we want to numerically model air cooling in a desktop computer, we would have to model at least the cooler fan and the source fan to get the correct flow field. This would be computational resources and time consuming if done using the actual geometry of the impeller and a rotating mesh algorithm. With the simplified proposed method we only have to define the subdomains of the impellers and add inside those domains some force coefficients and define some integration variable. This was just a trivial example, but there are other significant applications like forced coolers, cooling towers, drying kilns and all sorts of axial mixers for liquid solutions. This method could also be applied (with an inverse sign for the variable force coefficients) to model fields of axial wind turbines or fields of axial marine current turbines where the interaction and most efficient arrangement of the turbines in the field are of major importance. Results: In this paper, the "simplified numerical model" is applied to an axial fan for which, in the studied configuration, head pressure-flow rate curve is available from the literature. Numerical results are obtained using COMSOL Multiphysics' 3D turbulent incompressible flow built on a Reynolds average formulation of Navier-Stokes equations with k-ε closure. The xOz slice of the velocity field is presented in Figure 1. The xOz slice of the pressure field is presented in Figure 2. Conclusion: Our computed results are found to be in good agreement with measured or computed values of the flow downstream of such a hydraulic machinery. The method has proven to save a lot of computational time, e.g. a computation took less than 18 minutes on a workstation with 16 GB memory and 2 quad-core Intel Xeon 2.66GHz processors.

Reference

1. F. N. le Roux, The CFD simulation of an axial flow fan, MSc Thesis, University of Stellenbosch, South Africa (2010).
2. S. J. van der Spuy, T. W. von Backström, Performance of rotor-only axial fans designed for minimum exit kinetic energy, R & D Journal, 18(3), 63-69 (2002).
3. S. J. van der Spuy et al., An evaluation of simplified methods to model the performance of axial flow fan arrays, R & D Journal of the South African Institution of Mechanical Engineering, 26, 12-20 (2010).

Figures used in the abstract

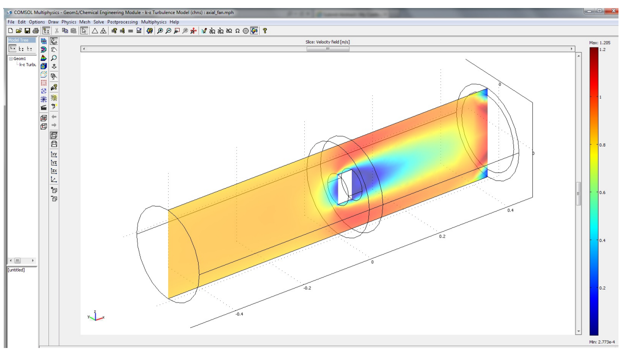

Figure 1: Velocity field (xOz slice) before and downstream the axial fan.

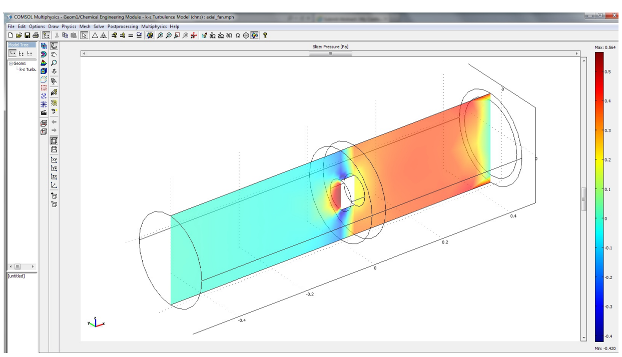

Figure 2: Pressure field (xOz slice) before and downstream the axial fan.