Simplified Numerical Model of an Axial Impeller

Andrei-Mugur Georgescu[1], Sanda-Carmen Georgescu[2]
1. Technical University of Civil Engineering Bucharest (TUCEB), Hydraulics and Environmental Protection Department, 124 Lacul Tei, Bucharest, Romania, 020396;
2. University POLITEHNICA of Bucharest, Hydraulics and Hydraulic Machinery Department, 313 Splaiul Independentei, Bucharest, Romania, 060042.

Introduction: We propose a simplified numerical method to model the flow field downstream of an axial impeller, namely the axial fan of Armfield's FM41 experimental unit (Fig. 1), from the Hydraulics Laboratory of TUCEB. 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 flow field downstream of the hydraulic machinery.

Figure 1. FM41 Unit

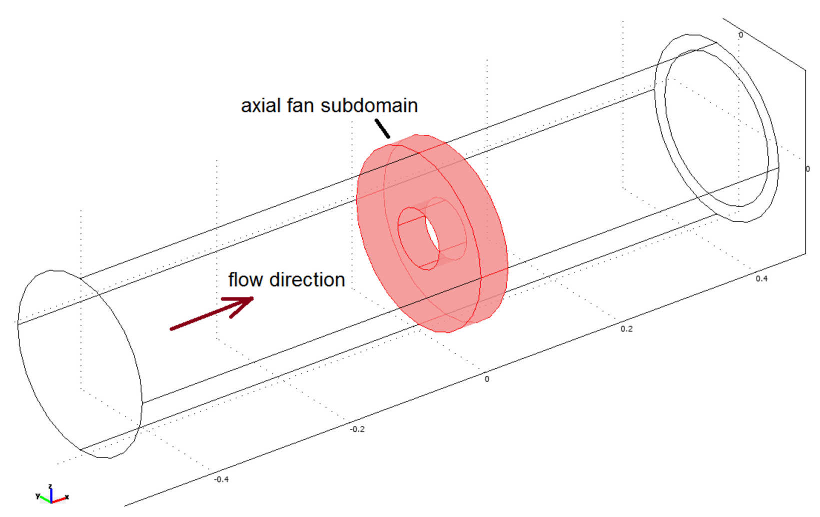

Figure 2. Fan subdomain

Computational Simplified Method: One doesn't have to model the actual blades of the axial fan and use a rotating mesh to compute the flow field downstream of the impeller, but only to insert force coefficients, namely volume forces F_x, F_y, F_z in the body forces fields of the Navier-Stokes equations (variable with the flow rate Q), as:

$$F_x = \Delta p / b = \left(60 - 42000 Q^2\right) / 0.04, \ \mathrm{N/m^3}$$

$$F_y = \frac{-\Delta p^2 z\,|z|}{7675\left(y^2+z^2\right)^2}, \quad F_z = \frac{\Delta p^2 y\,|y|}{7675\left(y^2+z^2\right)^2},$$

in a subdomain of b length representing the impeller (Figure 2), and to define integration variable giving Q at the inlet of the impeller for each iteration. The force coefficients will produce on the flow the same average effects as impeller's blades. We need only the rotational speed of the fan (2700 rpm) and its head (pressure Δp) - flow rate curve. Boundary conditions are: 0 pressure on the inlet and outlet sections (Fig. 3) and log wall function (h/2 wall offset) on model's walls.

Results: Results are obtained using COMSOL Multiphysics' 3D turbulent incompressible flow built on Reynolds average formulation of Navier-Stokes equations with k-ε closure. The velocity field as xOz slice, and the z-velocity at the exit are presented in Figures 4 and 5.

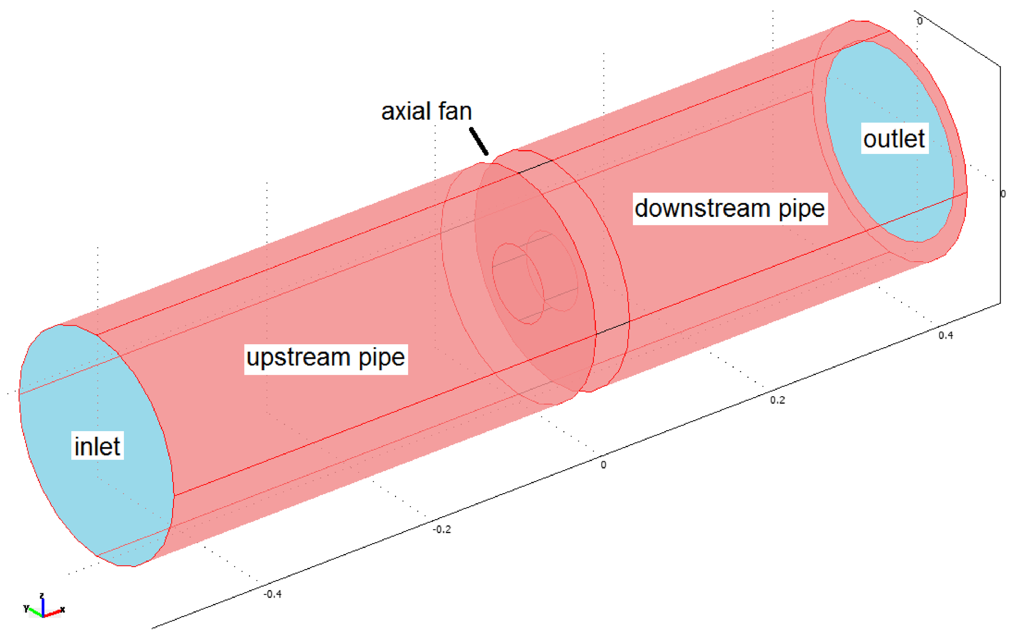

Figure 3. Flow domain

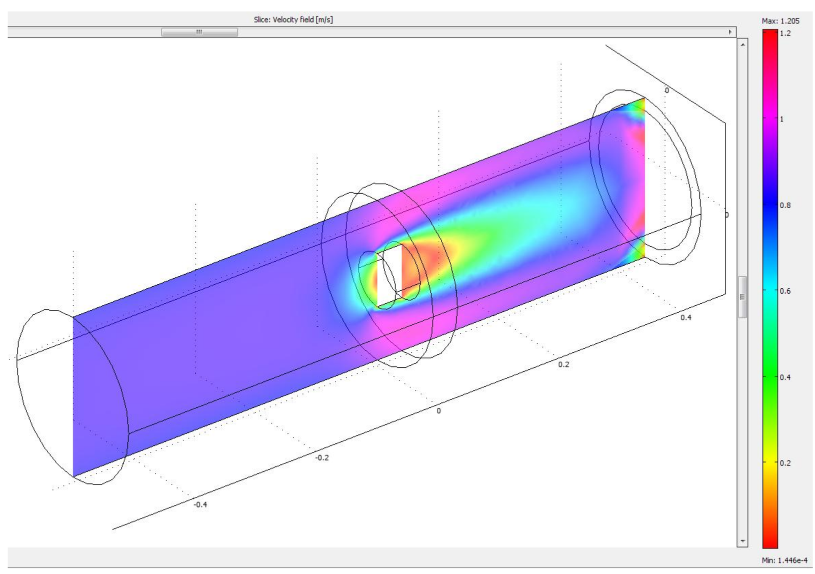

Figure 4. Velocity field

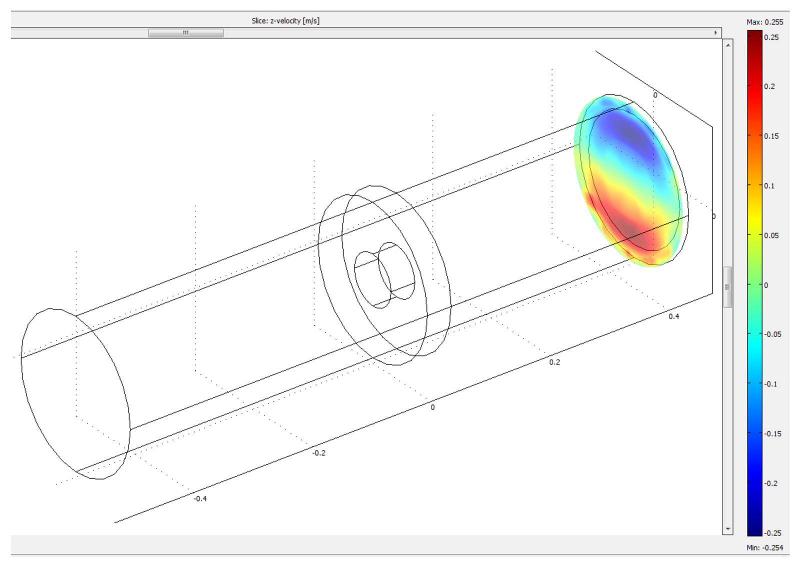

Figure 5. Exit z-velocity

Conclusions: Our computed results are 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.

References:

1. F. N. le Roux, The CFD simulation of an axial flow fan, MSc Thesis, Univ. of Stellenbosch, South Africa (2010)
2. S. J. van der Spuy, T. W. von Backström, D. G. Kröger, 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)