Simulating a Fan for Industrial Ventilation

G. Argentini
Research & Development Dept.
Riello Burners, via Ettore Riello 5, San Pietro di Legnago (Verona) – Italy
gianluca.argentini@rielloburners.com

Abstract: This work talks on simulation of a ventilating structure for an industrial burner. The mathematical model is based on the frozen-rotor technique, and the numerical simulation is used for resolution of fluid dynamics equations for the air flow into the rotating wheel and into the static volute of the fan. The numerical results obtained with Comsol's CFD module are compared with experimental data on the same structure, showing the good agreements between tests and simulation.

Keywords: fan, frozen-rotor, Navier-Stokes equations.

1. Introduction

The ventilating structure is a fundamental component of a burner. It ensures a continuous and appropriate air-flow rate towards the combustion head, where flame occurs. The ventilation components are principally two: the rotating wheel, also called *impeller*, and the static external chassis, called *volute* ([1]).

The basic problem for a fan is a correct design of impeller and volute for obtaining good ventilating performances: an high value of *static pressure* at the outlet of volute and high *mechanical efficiency* about energy absorption of the electric motor.

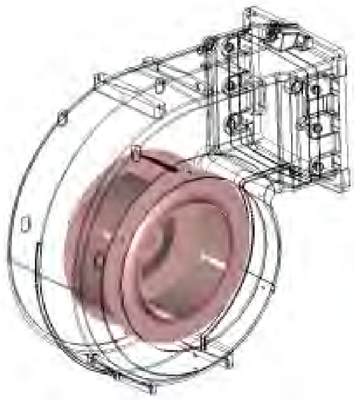

Figure 1. The impeller (colored) and the volute of a fan.

Static pressure is related to flow velocity field by Navier-Stokes equations of fluid dynamics, which are in general difficult to solve by analytical expressions. So a numerical approximation of solution of a boundary values problem is an important tool for helping the design for engineering purposes.

The fan that we have studied is a relatively complex system which is composed by an impeller with 13 backward-curved blades and a volute with a linearly growing external radius.

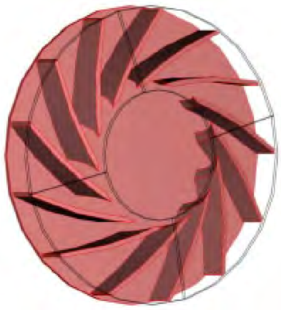

Figure 2. The impeller (colored) with 13 curved blades.

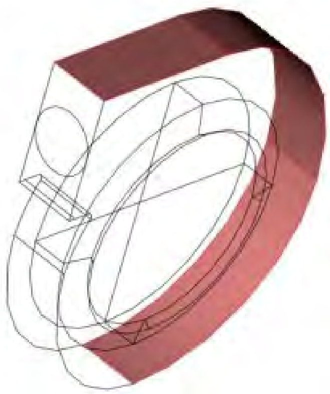

Figure 3. The volute.

2. Governing Equations

The equations that govern the physics of a fan are Navier-Stokes with no external forces. In fact, gravity is very small compared to strong centrifugal forces acting along the volute. If p is the pressure, **v** the velocity field, ρ and η the air density and dynamic viscosity, these equations are ([3])

$$\rho(\nabla\mathbf{v})\mathbf{v} = -\nabla p + \eta\,\Delta\mathbf{v}$$

which are valid on the air domain contained among the blades and between impeller and volute. The relative differential problem is completely defined imposing the boundary conditions on all the faces of impeller and volute. It is usual, in technical literature, to consider a reference value p_0 of static pressure at the inlet of impeller and a value Q_0 of flow rate at outlet of volute. Also, in all the other boundaries the condition is the vanishing of normal component v_n of velocity field: $v_n = 0$.

3. The Frozen-Rotor method

The Frozen-Rotor method is a numerical technique for the approximation of the flow velocity field in the geometrical interface between the rotating air domain D_1 in impeller and the air domain D_2 in volute. Let $\mathbf{v}_1 = (u_1, v_1, w_1)$ the field in the impeller computed in a cartesian frame $\{x_1,y_1,z_1\}$ rotating with impeller itself, $\mathbf{v}_2 = (u_2, v_2, w_2)$ the field in volute computed in a static reference frame $\{x_2,y_2,z_2\}$ and ω the angular speed of rotation. If the impeller is clockwise rotating on the plane x_1y_1, the frozen-rotor equations on the interface, where $x_2 = x$ and $y_2 = y$, are ([4])

$$\begin{cases} u_2 = u_1 + \omega\, y \\ v_2 = v_1 - \omega\, x \\ w_2 = w_1 \end{cases}$$

Note that these equations have the following interpretation: the absolute velocity $\mathbf{v}_2$ is the sum of the velocity $\mathbf{v}_1$ relative to the rotating frame (where rotor is "frozen") and the velocity of rotation $\mathbf{v}_r = \omega\,(y, -x)$, which is tangential to the interface.

Also, frozen-rotor approach is compatible with the geometry of the impeller blades: at the interface between D_1 and D_2, the volute field $\mathbf{v}_2$ depends on the two components of the impeller field $\mathbf{v}_1$, which are connected by the relation

$$v_1 = u_1 \tan\alpha$$

where α is the angle between the end of the blade and the cartesian axes x_1.

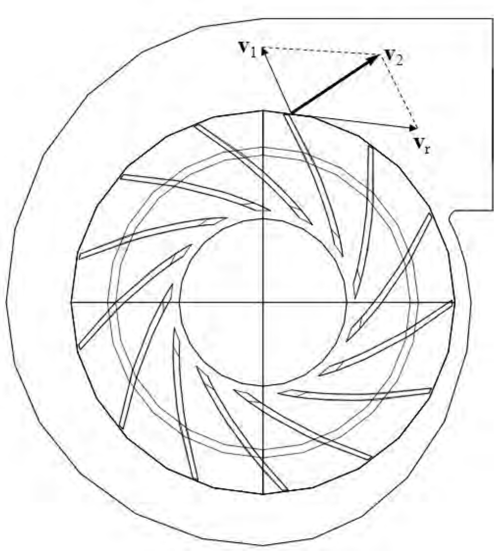

Figure 4. The composition of velocity fields in frozen-rotor approach. This rule is also known as "velocities triangle".

4. Use of COMSOL Multiphysics

We have used Comsol for numerical resolution of the described differential problem. The geometry is composed by two components: the air domain D_1 into impeller and the air domain D_2 into volute. In each domain we have imposed the physics modeled by the Comsol application mode *Incompressible Navier-Stokes* (CFD Module in ver.4 of the software). For the computation of the velocity field $\mathbf{v}_1$ in impeller, the application mode has been defined in all D_1+ D_2 , while for computation of $\mathbf{v}_2$ the application mode has been defined only in D_2.

The values of the physical properties of air were $\rho = 0.001$ g/cm^3, $\eta = 0.0001$ dyne s/cm^2. The rotation of impeller was 9000 rpm ($\omega = 942$ rad/s), which is an unusually high rotation speed for fan.

The physical boundary conditions were, as usually in fan simulations, a vanishing pressure at the inlet of the impeller and a predefined value of flow rate at the outlet of volute.

The frozen-rotor equations have been imposed as boundary conditions for the domain D_2 on the interface with domain D_1.

We have used a mesh of about 70.000 tetrahedral elements, for a total of 90.000 degrees of freedom for each application mode.

The numerical solver algorithm was the Direct UMFPACK built-in into Comsol.

At first, it has been necessary the computation of $\mathbf{v}_1$ in all the geometry. The result was so the "frozen" (i.e. referred to the frame $\{x_1,y_1,z_1\}$) velocity field into impeller, and, more important, the values of $\mathbf{v}_1$ to be used in the frozen-rotor equations as boundary data for $\mathbf{v}_2$. The second step was the computation of this field only in the domain D_2.

Using uniformly separated values of flow-rate Q at outlet, it has been possible to compute the values of static pressure p at outlet, and so to construct the *characteristic curve* of the fan, that is the profile of the function $p = p(Q)$ in a cartesian plane $\{Q, p\}$.

5. Discussion

In this section we discuss the comparison between numerical simulation and experimental data about the same fan.

As shown by graph, for flow-rate between 0 and 180 m^3/h, the fan pressure computed by simulation has a profile (dependence on flow rate) which is in very good agreement with profile obtained by experiments.

Also, the computed static pressure distribution along the external profile of volute is confirmed by experimental data.

These good results show that the frozen-rotor method can be a valid algorithm for numerical simulations of rotating fluid-domains into static casings, and that the CFD module of Comsol is a useful tool for implementing it with no necessity of writing lines of code with a programming language.

The computational model can also give a comparison with a mathematical model of air speed v distribution along the radial directions. In this model, due to Pfleiderer ([2]), the conservation of angular momentum for the flow traiectories is assumed. The mathematical expression for this property is the following formula for speed distribution along a radial direction into volute

$$v = \omega R^2 / r^a$$

where R is the external radius of impeller, r is the variable from R to the external radius of volute, and a is an experimental parameter ($a \approx 1$). The comparison of numerical distribution with the mathematical one gives a way to compute the value of a for each possible flow rate of the fan.

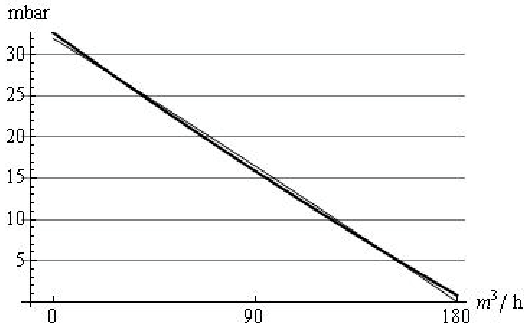

Figure 5. The fan characteristic curve. Comparison between simulation (thick line) and experimental data (thin line).

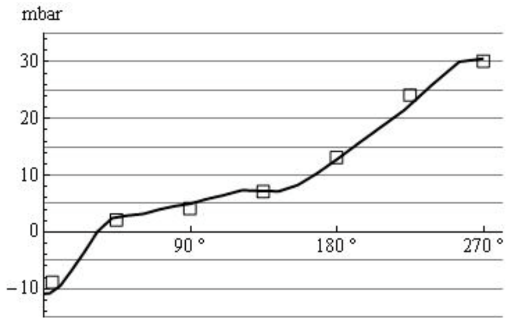

Figure 6. The static pressure distribution along volute. Comparison between simulation (thick line) and experimental data (squared).

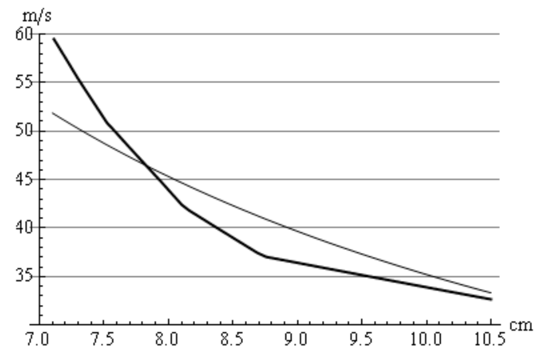

Figure 7. The radial distribution of speed flow at volute outlet in the case of flow-rate equal to 180 m^3/h. Comparison between simulation (thick line) and Pfleiderer mathematical model (thin line) for $a = 1.13$

Acknowledgments

Many thanks to Ing. Valerio Marra and Dr. Cesare Tozzo of Comsol Italia for their support and precious hints during the development of this work.

References

1. Frank Bleier, *Fan Handbook: Selection, Application and Design*, McGraw-Hill, (1998)

2. Christophen Brennen, Hydrodinamics of pumps, Oxford University Press, (1994)

3. Reza Malek-Madani, *Advanced Engineering Mathematics*, vol.2, Addison-Wesley, (1998)

4. T. Meakhail and S. Park, A study of impeller-diffuser –volute interaction in a centrifugal fan, *Journal of Turbomachinery*, **127**, pp. 84-90, (2005)