The Burning Need for Modeling

BY GIANLUCA ARGENTINI, RIELLO BURNERS, LEGNAGO, ITALY

When we think about heating and cooling, we often neglect to consider the energy consumed by components we take for granted such as fans, whether for distributing air into a room or for feeding it into a burner for combustion. The energy consumed by fans depends on many aspects, but in a high-power burner (> 1 MW) the percentage of energy consumed by fans is about 30 – 40% of the total needed to run the system, which can range from 150W for a small ventilator to 25 kW for a large burner. In fact, in residential and domestic burners (up to 100 kW thermal power), due to small geometries, the percentage can be much higher, with up to 80% of the electrical energy for the system consumed by fan.

In addition, new ecology regulations in EU Directive 125/2009 addressing fans for industrial purposes fixes limits of about 40 – 50% for fan power consumption, putting additional pressure on manufacturers to improve their products. There is, in fact, great potential for saving energy as well as manufacturing costs in a well-designed fan, and here modeling plays a very important role because it not only saves considerable engineering time but also results in optimal designs. By reducing power consumption, it is possible to use a smaller motor to obtain the same performance. Other benefits include reducing the overall weight of burners and improved design of the components in the combustion head.

This type of continual development is part of our job at Riello Burners. We are an internationally active company in the fields of oil and gas combustion, ventilation systems and hydraulic pumping. Our headquarters and main production facility is in Legnago near Verona, Italy, and we employ roughly 600 people. In terms of burners, we manufacture high-efficiency low maintenance burners with capacities from 10 kW to 32 MW, and they are used in the full range of residential and commercial heating applications as well as in industrial processes.

Two Key Components

The ventilating structure is fundamental to a burner because it ensures a continuous and appropriate flow of air into the combustion head where the flame is located. There are two principle ventilation components: the rotating wheel or impeller, and the external chassis in which it operates, and this is also called a volute. The basic challenge when designing a fan is to determine the configuration of the impeller and volute so as to obtain high values of static pressure and flow rate at the volute's outlet while also maintaining high mechanical efficiency, which means less electricity required to run the fan.

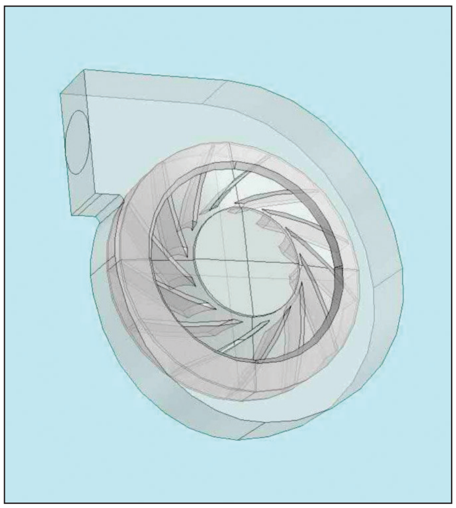

Fig 1. Geometry of a burner ventilation fan consisting of the rotating impeller in the center and an expanding volute.

The particular fan we recently studied is a relatively complex system and consists of an impeller with 13 backward-curved blades and a volute with a linearly expanding external radius (Fig 1).

Prior to using COMSOL Multiphysics, we developed our fans and created their characteristic curves using only experimental tests with Pitot tubes; these are tubes with only one open end which is pointed directly into the fluid flow, and the static pressure measured inside the tube corresponds to the flow using Bernoulli's equation. The problem with this type of testing is that if you change the geometry or some physical boundary condition you must run new tests, each requiring almost an entire day.

In an effort to make a better choice between the geometries and to reduce design time, we turned to mathematical modeling. Using the "frozen rotor" method, the construction of a model and running of the computations usually takes only two hours. I now have a rapid way to test new concepts or designs that I first work out using theoretical methods based on fluid dynamics and mathematical resolution of some partial differential equations. The numerical simulations then point me in the direction of the configuration that's probably the best one from the outset.

Before simulating, we would create and test tens of physical prototypes during the course of a year. Most of the time, these models would exhibit some problem. With COMSOL, I've used two possible geometric configurations of the volute which we created and tested. The final configuration has exhibited no problems and has very good performance. The time between starting the models and the ending of experimental tests was about one month compared to a year.

A Fixed and Moving Geometry in One Model

Simulating the flow in the total air domain in the overall structure is typically a difficult task using computational fluid dynamics due to the interaction of a moving geometry (the impeller) and a fixed geometry (the volute). To attack this problem, we use the frozen rotor method, which is a numerical technique for approximating the flow velocity field at the geometric interface between a rotating air domain in the impeller and the static air domain in the volute. Using COMSOL's CFD Module, we computed a velocity v_1 in the entire geometry assuming a flow entering axially from the center disk of the fan and leaving the housing from

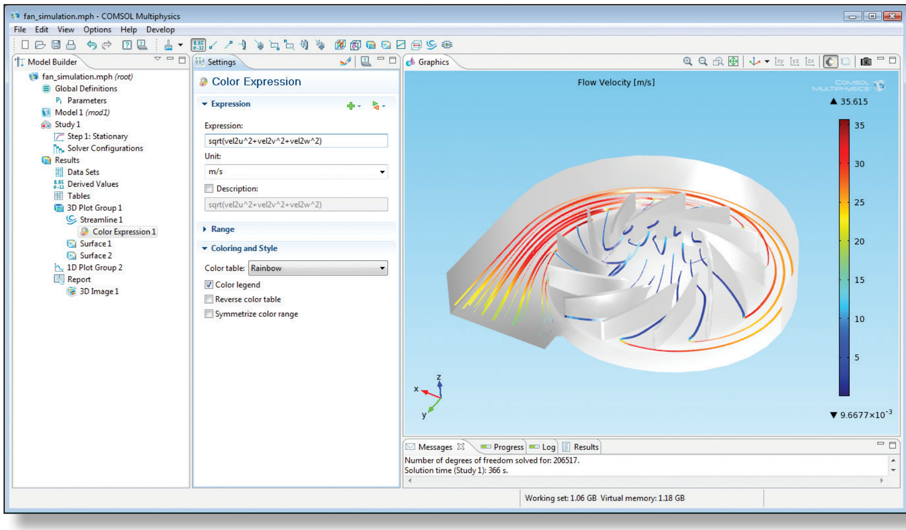

Fig. 2. Streamlines showing the velocity into the impeller and the velocity into the housing

the outer section without considering any rotation. The result is the "frozen" velocity field into the impeller, and more importantly, the value of v_1 is used in the frozen rotor equations for the boundary data for v_2. Then a second computation determines this field only in the domain of the volute (Fig. 2).

With experiments we determined that the fan pressure computed by the simulation has a profile that is in very good agreement with experiments. These good results proved to us that the frozen rotor method can be a valid algorithm for simulating the rotating fluid domains in the static casings.

I found the software particularly efficient in creating the geometry and the mathematical treatment of the boundary conditions where I can express them using analytical formulas. For example, in the frozen rotor method, the correct formulation of the velocity field on the interface between the rotating and static domain is very difficult to obtain without the ability to declare the algebraic conditions of compatibility.

I have noticed that a significant advantage of using COMSOL for our ventilation structures is a more optimized shape of the volute, which results in a more laminar and therefore more efficient air flow towards the burner's combustion head. This improved flow means that we can use less electric power and yet achieve the same flow rate. For the end user (e.g. a family or a company), the direct advantage can be a less expensive burner; the more-important indirect advantage is the total energy saving for the environment. For example, if for a ventilator we could reduce the power consumption from 250W to 200W with the same flow rate, the energy savings in the case of 10,000 parts/year – typical annual sales volume for a fan of the type being modeled – is 0.5 MW that need not burden the environment.

Future plans for COMSOL are to develop a model that starts with the frozen rotor method and couples flow and materials structures because one of the more important challenges in the field of fans is to minimize vibrations, energy losses and noise. ■

ACKNOWLEDGEMENTS

The author would like to thank his colleagues at the Angiari Combustion Research Center for their help and suggestions during executions of ventilation experiments.

LEARN MORE

www.comsol.com/papers/7988/

About the Author

Gianluca Argentini is a manager in the R&D department at Riello Burners working on ventilating structures and the fluid dynamics of rotating components. He has a Laurea in Mathematics from the University of Padua and a Masters in Scientific Computing and Fluid Dynamics from the University of Erlangen (Germany). He first worked as a mathematics instructor at the university level and then joined Riello Burners to perform scientific computations and develop mathematical models.