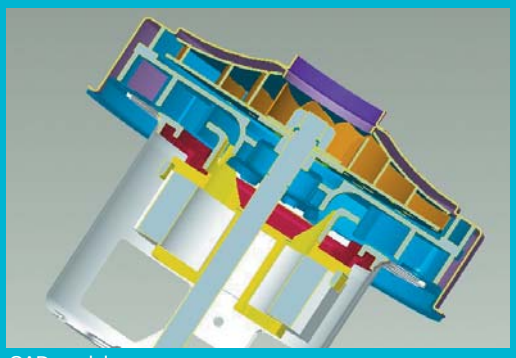
Improving Efficiency of Vacuum Cleaner Fans

ANSYS CFX is an integral part of turbomachinery performance studies focused on radial fan design.

By Philipp Epple and Caslav Ilic
Institute of Fluid Mechanics
Friedrich-Alexander University of
Erlangen-Nürnberg, Germany

At the Institute of Fluid Mechanics (Lehrstuhl für Strömungsmechanik, LSTM), a research group has been established to conduct research and development work in the field of turbomachinery. This work includes turbomachinery design based on combined analytical design considerations, numerical performance studies, rapid prototyping, and experimental investigations of prototype impellers and diffusers as well as complete fans. Extensive research and development has been done in the field of radial compressors with an emphasis on radial fans for vacuum cleaner applications. ANSYS CFX 10.0 computational fluid dynamics software is an integral part of the design process.

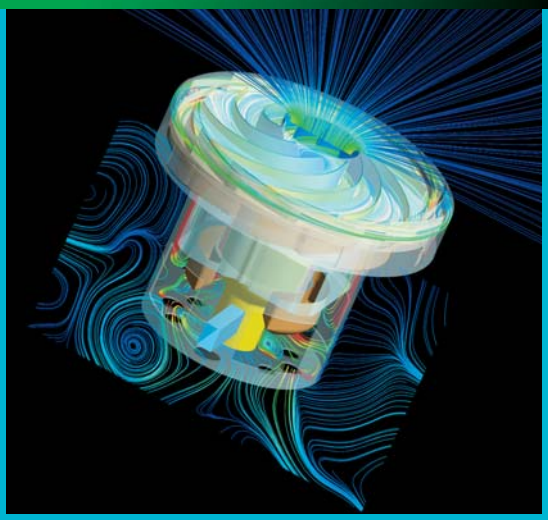


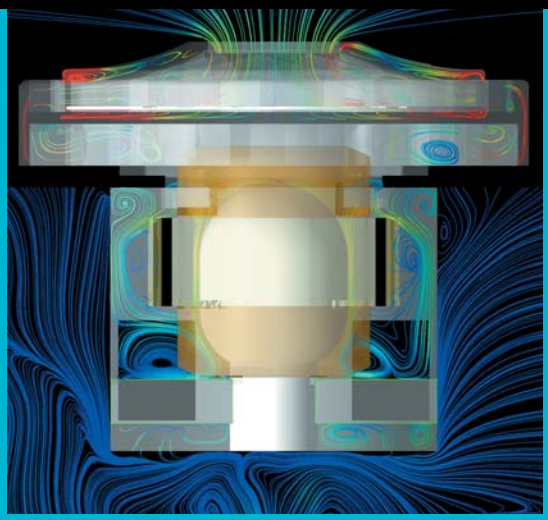
CAD model

The flow through centrifugal turbomachines is principally radial in the region in which the energy from the rotating impeller is transferred to the air flow. For axial flow fans, the rotating impeller is passed in an axial manner in the region in which the energy transfer takes place. Work on axial blowers also is being conducted at LSTM-Erlangen.

Centrifugal fans are similar in many respects to both centrifugal pumps and centrifugal compressors. The principal distinction between these machines is that pumps propel liquids that are practically incompressible, but compressors handle gases under such conditions that a clear change in the density of the flowing fluid results. Usually, fans are devices that handle compression ratios below 1.1, whereas blowers operate between 1.1 and 4, and compressors operate above the compression ratio of 4.

In the first step of the development process, the group at LSTM-Erlangen carries out performance studies of existing fans that later will be redesigned to reach higher efficiencies. These performance studies are carried out numerically utilizing CAD data of the real geometry as a basis for the grid generation. ANSYS ICEM CFD software provides excellent tools for fast grid generation for complex geometries. ANSYS CFX provides all the multiple frame-of-reference tools needed to perform computations for





Detailed simulation of a vacuum cleaner fan, including impeller, diffuser, deswirl vanes and motor

rotating machinery. With ANSYS CFX, it is quite easy to conduct performance studies of the radial fans used in vacuum cleaners. A CFD simulation with ANSYS CFX can encompass the flow from the test rig through the impeller and the diffuser into the guide vanes, through the motor and out into the space behind it. Very detailed information can be obtained from the computational results, and these can be employed to compare to corresponding results obtained during experimental studies. A complete test rig is available at LSTM-Erlangen so that combined numerical and experimental investigations of radial blowers can be carried out.

LSTM-Erlangen has specialized in compressors used in vacuum cleaners that have very high rotational speeds, ranging typically from 30,000 rpm up to 50,000 rpm.

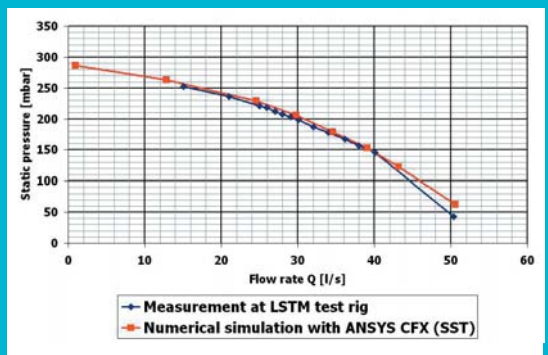
After measuring and simulating the actual fan, the improved design goals are set. These goals usually include better efficiency at the same pressure and flow rate. When a complete new design is targeted, a full system inverse mean line design is performed, considering impeller, diffuser and deswirl vanes as a unit and not separately. In addition, the system in which the fan will operate is considered, so that the fan will be perfectly matched to the operating conditions of the vacuum cleaner. Verifying the geometries generated with the inverse full system mean line design on the CFD simulation with ANSYS CFX, it is possible to iteratively improve the design of the fan to specifications. In this way, it was possible for LSTM-Erlangen to achieve improvements of more than 10 percent in efficiency of vacuum cleaner fans. One valuable feature of ANSYS CFX is the power syntax, which permits the writing of post-processing scripts using inline Perl commands. We used this method to get precise information on the flow inside the machine needed to validate and to improve the inverse full system mean line design.

Finally, with the design validated and improved with a careful and detailed ANSYS CFX simulation, there is almost no risk in building expensive prototypes.

Experimental investigations are then carried out at LSTM-Erlangen's test rig, and the results are compared with specifications of the improved fan and with the CFD computations. The detailed full system simulation agrees very well with measurements at the test rig.

If further development is needed, modifications are carried out. After careful design and CFD validation and improvement with ANSYS CFX, it is seldom necessary to make additional modifications. ■

Grid generated with ANSYS ICEM CFD



Comparison between measurement on the LSTM test rig and simulation with ANSYS CFX