3-D CFD prediction playing a key role in the aerodynamic design of high-performance fans

Dr. Wilfried Rick, Dr. Eribert Benz, Alain Godichon

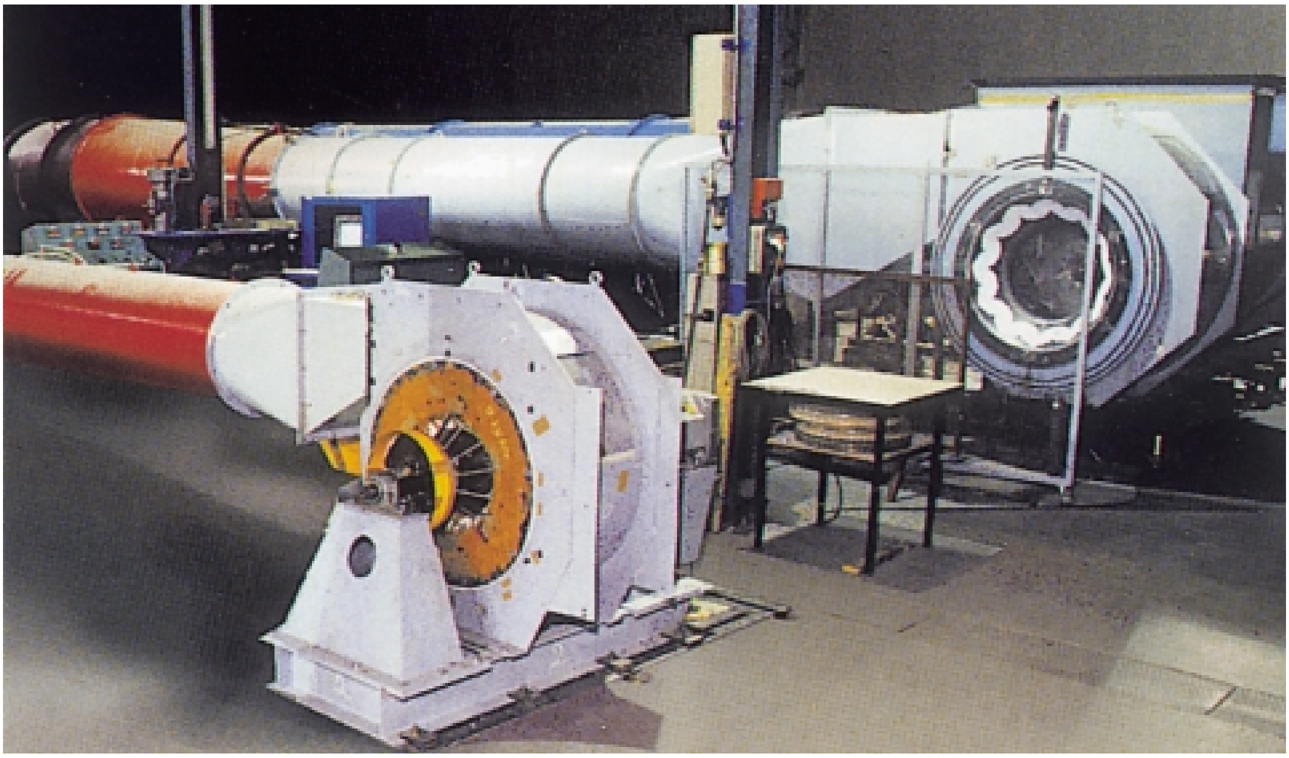
While experimental work still plays an important role in flow physics research, simulation is increasingly becoming the method of choice for the detailed study of flow phenomena. At ABB, CFD (Computational Fluid Dynamics) simulations based on 3-D Navier-Stokes equations have been incorporated in the design process for high-performance fans in order to find new, optimum component configurations and reduce experimental investigation to a minimum. Overall fan performance is also improved, while new products can be developed faster and with a smaller degree of uncertainty. The simulations have further shown that CFD can be used routinely for matching machine components in many other applications.

In addition to carrying out experimental work, most fan manufacturers today rely heavily on empirical one- (1-D) or two-dimensional (2-D) flow equations when determining the fundamental aerodynamic design of their machines [1].

Since relatively little has been published on the use of CFD for studying the internal flows of centrifugal fans, it can be concluded that it is still common practice for the manufacturers of these fans to apply the principle of similitude and scale new machines directly from preceding designs. This procedure makes use of extensive databases and is known to be reliable, but it is also time-consuming and provides no insight into the machine's internal flow pattern. To find out more about the flow phenomena within the impeller blade passages or inside the fan casing, tedious and costly experimental investigations are necessary. ABB Solyvent Ventec has consequently started to use CFD simulations based on 3-D Navier-Stokes equations in fan design. CFD brings a systematic approach to the design and development process, enabling new, optimum configurations to be found and experimental investigation to be kept to a minimum. What is more, CFD allows a more comprehensive understanding of the flow phenomena, providing a basis for improved fan performance and faster development of even completely new products while reducing uncertainty and minimizing costs.

Designing industrial fans for new demands

In the traditional market for centrifugal fans there has been a tendency in the past to emphasize the first-time cost of the machines, and improving or controlling performance usually has had to take second place to concerns about ease of



The ABB Fan Group has several laboratories for tests according to international and national standards such as ISO, AMCA, NF, BS.

manufacture and installation. Now that the fan market has become global and highly competitive, operating costs are being given a higher priority. The large forced draft fans used in thermal power plants, for example, are expected today to exhibit high aerodynamic efficiency over a much wider range of operation than they did a few years ago.

A case in point, and one which illustrates how well CFD simulations are suited for predicting performance and studying fan aerodynamics in detail, is the double-inlet, high specific speed centrifugal fan with adjustable inlet guide vane assembly (for flow regulation) and scroll casing shown in 2 .

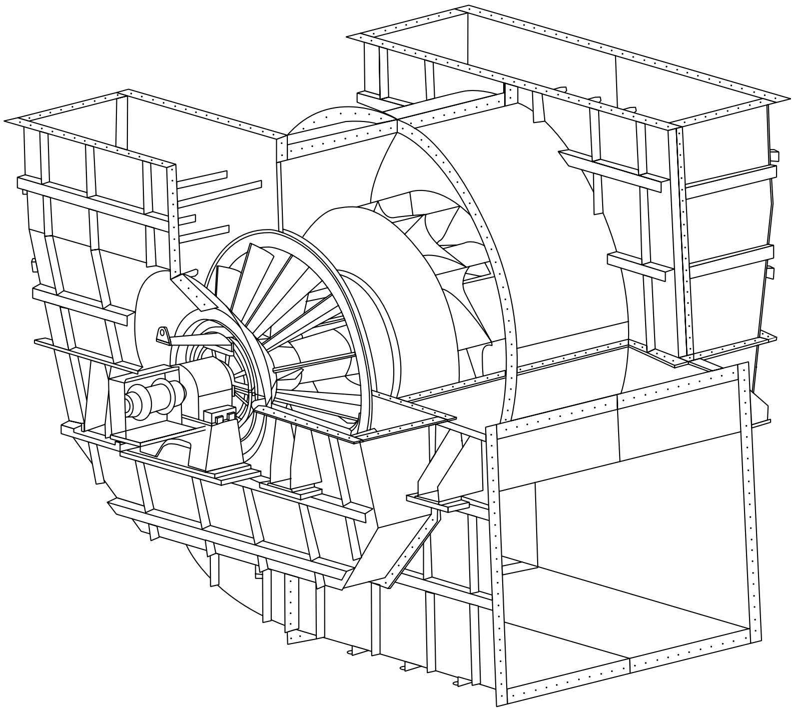
Seen in this diagram are the inlet boxes with adjustable inlet guide vanes (IGV), the inlet mouth and the double inlet impeller, which discharges into a single tongue scroll housing of constant width. The adjustable IGVs match the fan's flow characteristic to the load by throttling the flow yet still ensuring a good inlet flow to the impeller. The pre-swirl imparted to the inlet flow at constant rotational speed by varying the stagger of the vanes simultaneously reduces the fan head, so that the working point of the machine can be controlled and shifted along constant lines of resistance.

The IGV assembly, which is fitted diagonally, is mounted in the conical outlet of the inlet box. Typically, the IGVs for industrial fans consist of flat sheet-metal blades which are turned through large angles of up to 90 degrees. This simple, low-cost design ensures ease of manufacture, low losses at zero stagger position and almost zero flow. However, the meridional and simultaneous circumferential turning of the flow within the IGV section at off-design load results in a complex flow exhibiting significant three-dimensional effects that make one-dimensional tools rather unsuitable for checking for pressure losses or for analyzing the IGV and impeller interaction.

The shrouded radial double-inlet impeller shown is a high-flow design with highly backward leaning, aerofoil-type blades. Under design operating conditions the flow coefficient is 0.307 and the head coefficient is 1, corresponding to a specific speed of 1.65. The figures for the Mach number and the Reynolds number are 0.14 and 0.6×10^6, respectively, in each case referred to a stagnant inlet condition, the impeller tip speed and the tip width.

Computational methodology based on a commercially available CFD code

CFD is a very powerful tool for predicting complex flow structures and provides a



2 Cut-away drawing of a forced draft fan, showing the inlet boxes with the adjustable guide vanes, the double inlet impeller and scroll housing

better understanding of the flow characteristics in turbomachinery components. ABB Solyvent-Ventec and ABB Corporate Research[1] have made use of this fact by incorporating CFD in the design process for centrifugal fans. The results have shown that CFD is accurate enough to be used to calculate the performance characteristics of these fans.

A commercially available CFD code was used to study the aerodynamics of the described fan. Incompressible fluid properties were assumed. The code, which solves the Reynolds averaged Navier-Stokes equations on structured grids, was applied to a coupled single-blade passage of the IGVs and the impeller as well as to the coupled system of the full rotor and the scroll housing. The surface grid of the computational domain for the scroll housing is given in 3, which shows one half of the impeller and scroll housing. In order to show the blade design better, part of the impeller centerplate has been cut away.

All steady-state flow calculations carried out are based on turbulence modeling using the standard 'high-Reynolds number' K-ε model in conjunction with a wall function approach for the closure of the Reynolds-averaged Navier-Stokes equations.

In all cases the CFD models used captured the effects of the recirculating flow along the impeller shroud caused by the clearance flow through the gap between the stationary inlet mouth and the rotating impeller shroud.

Generally, a mixing plane approach was used at the interfaces between the

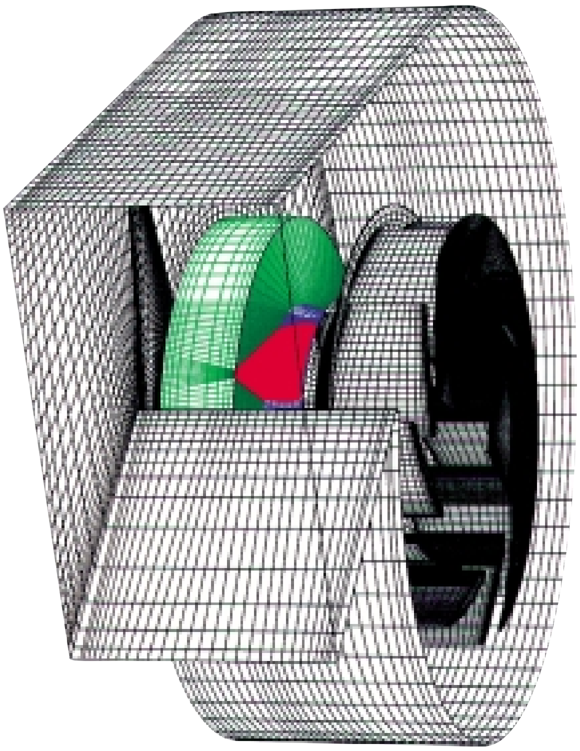
[1] ABB Corporate Research activities in this field have since been transferred to ABB ALSTOM POWER TECHNOLOGY Ltd.

Industrial fans get a boost from R&D

Fans come in an enormous variety of types and sizes, with designs for every conceivable use. In thermal processes, they may be used to transport gases, vapors and gas/solids mixtures for the purpose of ventilation, cooling, drying, air-conditioning or combustion. Industrial sectors that depend heavily on them include power plants, highway construction (in tunnels), cement-making, chemical/petrochemical plants and mining. Smaller fans are used to cool electronic components and motors.

The competitive forces driving today's global economy have resulted in many of the newest technologies being introduced progressively, starting with the aerospace industry and leading to stationary gas turbines for power plants, then to the industrial compressors and fans. As a leading fan supplier, ABB is duty-bound to keep abreast of all the latest developments.

Recent R&D work has yielded important new knowledge, which ABB uses, as this article shows, to improve the aerodynamic design of its fans. Another investment, in 'environmentally safe engineering', has produced fans that run on less power. These include axial fans from ABB Fläkt with power inputs of up to 2.5 MW and radial fans built by ABB Solyvent-Ventec, with wheels as large as 5 meters in diameter and power inputs of up to 10 MW.



3 Computational grid for the scroll housing and impeller. Such CFD models are used to capture the flow effects in the fan components.

stationary and the rotating grids. At the interface to the stationary grid of the scroll housing, however, a frozen rotor approach was used as the circumferential variation of the flow could be large compared with that across an impeller blade passage at the tip. (The mixing plane approach is based on conservative circumferential averaging at the grid interfaces; the frozen rotor approach gives frame changes across the interface without relative changes in the grid positions over time.) Mass flow and turbulence levels were imposed at the inflow boundaries of the computational domain.

Calculations were performed for four different operating points at constant rotational speed over a range of 80% to 140% design flow. The zero stagger position of the IGVs was used to investigate the design and high-flow operating conditions. For the purpose of comparison, stagger angles of 60 degrees and 45 degrees measured from the meridional plane were used with 80% and 90% design flow, respectively.

Detailed flow analysis

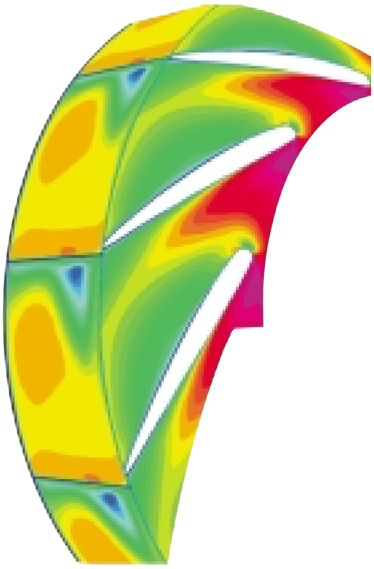
Some examples of actual CFD results obtained with a radial fan are used in the following to show how CFD can be used to investigate complex flow physics and how the accuracy of modern CFD methods gives the designer a reliable tool with which to optimize turbomachinery components.

As is well known, the incidence with respect to the impeller blading is an important parameter in fan aerodynamics. For instance, if the blade loading or the level of positive incidence (stagnation point shifted towards the blade pressure side) is high, adverse pressure gradients acting on the blade suction surface increase the boundary layer growth, and thereby the risk of boundary layer separation. This can lead to blade stall, which is associated with high losses. On the other hand, it is known that the profile loss is insensitive to the incidence at low inlet Mach numbers, this being particularly true for profiles with blunt leading edges.

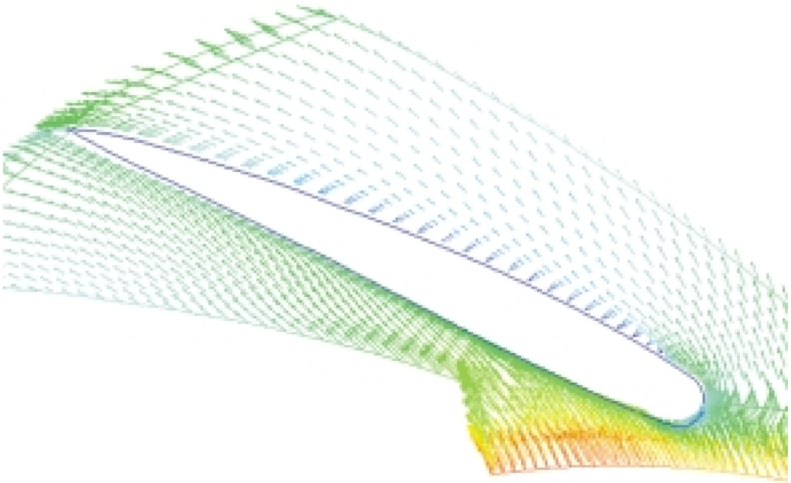
A uniform incidence distribution along the span can nevertheless contribute to a significantly increased efficiency and pressure rise [1]. To check the blade design for the radial impeller considered, with its typically strong meridional flow turning upstream of the blade's leading edge and a clearance jet impacting onto the leading edge at the impeller shroud, details of the flow on the blade's leading edge are needed.

Design flow conditions

The flow pattern close to the impeller shroud and impeller exit under design conditions is shown by the velocity distribution in 4. Clearly seen is the typical formation of the jet/wake pattern in the downstream parts of the impeller, which results in an accumulation of low-energy fluid in the near shroud/suction



4 Velocity distribution at design load. This figure shows the flow pattern close to the impeller shroud and the impeller discharge.

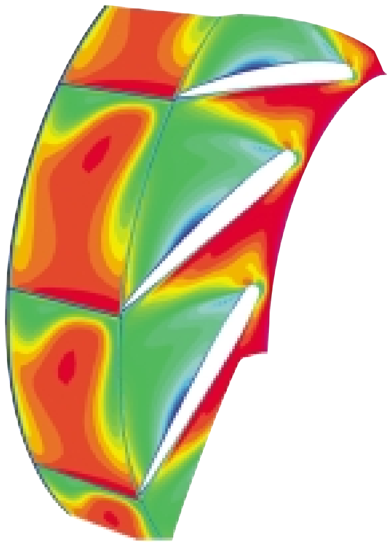


5 Vector flowfield at design load (impeller shroud). The CFD analysis shows that even under design conditions the boundary layer flow along the pressure side is close to separating - a condition known as 'corner stall'.

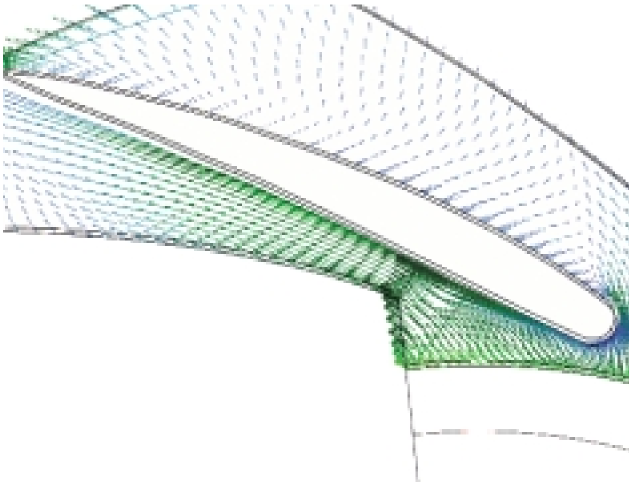
side area and high-velocity fluid in the centerplate pressure-side region. The generation of streamwise vorticity along the impeller passage can be explained by the centrifugal forces prevailing in the axial-to-radial bend region (these move low-momentum fluid towards the shroud from the blade pressure and suction surfaces) and the Coriolis forces acting in the radial part of the passage (forcing low-energy fluid to migrate from the centerplate and shroud walls onto the blade suction surface). However, from 5, which shows the corresponding vector flowfield, it is seen that there is no backward flow involved in this flow process. Nevertheless, the CFD analysis indicates that even under design conditions the boundary layer flow along the pressure side is close to corner stall (flow separation), being caused by the negative incidence onto the blade's leading edge and the imposed diffusion further downstream. Details of this mechanism are discussed in [2].

Part-load flow conditions

At 80% design flow operation with pre-swirl imparted to the inlet flow by 60-degrees staggering of the IGVs, blade loading is reduced and the wake flow pattern becomes less pronounced 6. In contrast to the situation with design flow, the low-speed region at a position on the blade pressure side at half chord length and the vector flowfield at the impeller shroud together reveal a radially inward



6 Velocity distribution at part load with pre-swirl imparted to the inlet flow by 60-degrees staggering of the inlet guide vanes. It can be seen that the wake flow pattern is less pronounced.



7 Vector flowfield at part load (impeller shroud). A radially inward flow can be seen in this part of the impeller.

the boundary layer and wraps itself around the leading edge. The course of the stagnation streamline and the saddle point, which divides the separation streamline on the pressure side and suction side leg of this leading edge vortex, is seen in 8 . The boundary layer, which re-develops beyond the separation line, experiences a strong expansion around the blade's leading edge. This results in separation further downstream in the centerplate/suction-side corner, being caused by an adverse pressure gradient.

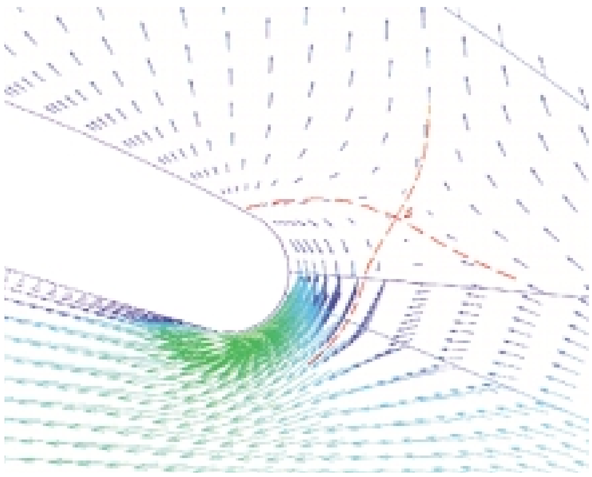
reason for this was found to be a tip/leading edge corner stall vortex developing on the blade pressure surface and originating from leading edge separation at the blade tip due to a large negative incidence. This vortex interacts with the shroud end wall flow and leads to the mentioned backward flow along the shroud at the blade pressure surface. A detailed study of the clearance flow, which is injected from the fan-side cavity through the clearance between the stationary inlet mouth and the rotating shroud into the inlet flow in this region, indicates that it contributes to the negative incidence onto the blade's leading edge at the impeller shroud. Also, the clearance flow, which leads to a circulating flow along the impeller shroud, is another cause of volumetric loss.

The flow at the leading edge close to the impeller centerplate has also been analyzed in detail at this off-design working point. In contrast to the predicted strong negative incidence at the blade tip, the vector flowfield in 8 reveals a large positive incidence onto the leading edge, resulting in the formation of a leading edge vortex. The underlying reason is that the incoming boundary layer flow on the centerplate cannot withstand the pressure gradient in front of the blunt leading edge and starts to separate upstream of it. This process is associated with the formation of a vortex that rolls up

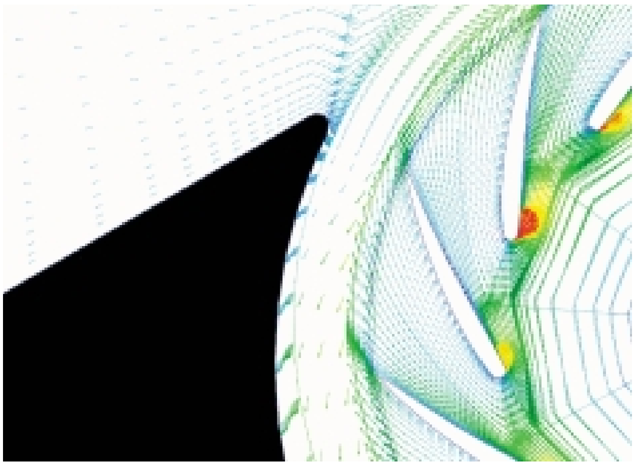
Matching the impeller to the scroll casing

To predict the overall fan efficiency and reveal where the highest losses are located in the fan, it is also necessary to investigate the flow in the scroll casing.

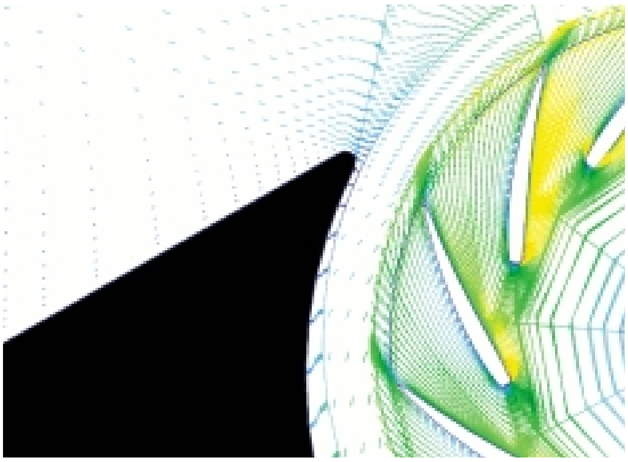
In accordance with the impeller discharge swirl that the scroll is designed for, there is just one working point per



8 Vector flowfield at part load (impeller centerplate). The large positive incidence onto the leading edge results in the formation of a leading edge vortex.



9 Vector flowfield of the impeller and scroll casing at design load. The stagnation point is located on the leading edge of the tongue of the scroll casing.



10 Vector flowfield of the impeller and scroll casing at 140% design flow. Fluid is pushed back into the diffusing exit duct of the casing, so that the stagnation point is located inside the scroll passage.

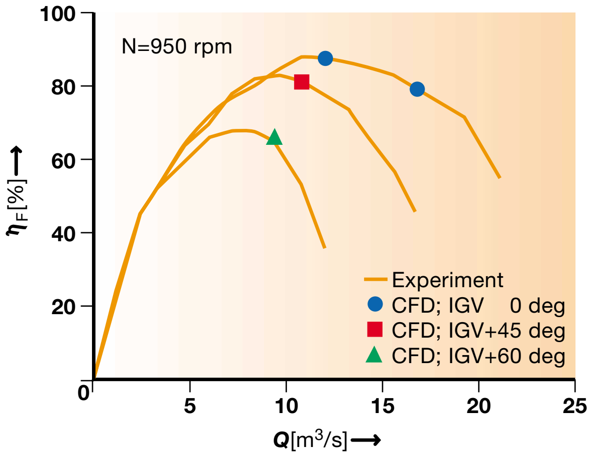
constant-speed line (normally the design point) for which the scroll imposes an almost circumferentially uniform pressure distribution at the impeller tip. During an excursion from this operating point, a peripheral asymmetric pressure field occurs in the scroll housing, causing a circumferential variation in the blade loading and mass flow distributed to the different blade passages and leading to strong non-steady interaction between the impeller and scroll flow. The quasi-steady frozen rotor approach with the rotor position fixed in relation to the scroll tongue is used for simplification and to reduce turn-around times for the computations.

At the design flow rate the stagnation point of the streamline, which divides the flow entering the scroll casing from the flow being discharged, is located on the leading edge tip of the scroll tongue. At higher flow rates the casing is too small to accommodate the flow, which then accelerates circumferentially. The static pressure at the first cross-section of the scroll passage is consequently higher than outside, ie at the inlet to the diffuser. The stagnation point is shifted inside the scroll casing and the corresponding static pressure gradient pushes fluid back into the diffuser. Conversely, at low flow rates the flow is decelerated up to the diffuser exit, so that fluid is pushed inside the scroll casing under the tongue and the stagnation point moves along the casing wall towards the diffuser exit.

Details of the flowfield at the tongue of the scroll casing at design load are given in 9 . As stated above, the stagnation point is located on the leading edge of the tongue due to the circumferentially uniform pressure at the scroll inlet. The blade passages approaching the scroll tongue are affected locally by a higher back-pressure, resulting in a reduced through-flow that is associated with higher blade loading and large positive incidence angles. As already mentioned, at a high mass flow rate (140% design flow), fluid is pushed back into the diffusing exit duct of the casing. Thus, the stagnation point is located inside the scroll passage. The associated backflow experiences strong expansion around the leading edge of the tongue, followed by a massive flow separation due to the adverse pressure gradient prevailing in this exit section of the casing 10 .

Performance characteristic – measurements versus CFD prediction

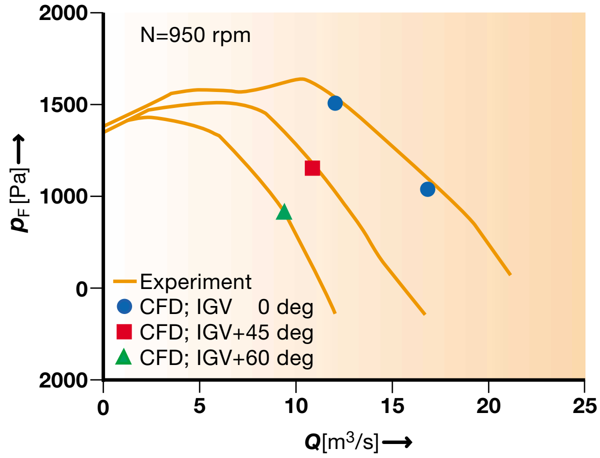
ABB Solyvent-Ventec conducted performance measurements on a scaled test rig and used the data to compare the CFD predictions with the global measurements of total fan pressure rise and total efficiency. From 11 and 12 it can be seen there is very good agreement between the measured and predicted values. The uncertainties in overall total efficiency (neglecting the bearing losses) predicted for flows ranging from 80% to 140% design flow were found to be smaller than 1.5%. The results show



11 Performance map showing the good agreement between measurements carried out on a scaled test rig and CFD predictions

η_F *Fan efficiency*

Q *Flow rate*



12 Fan pressure versus flow rate. Comparison of measurements and CFD predictions

p_F *Fan pressure*

Q *Flow rate*

that CFD is able to correctly predict the complex flow structure. In addition, the detailed analysis of the local flow structures provides a better understanding of the performance characteristic of a specific fan. At the same time, in-depth knowledge of the flow gives the designer a better overview of the performance characteristic of radial fans over their full operating range.

Benefits for evaluating design candidates

As the results of the described investigation show, CFD flowfield predictions permit a very detailed study of internal flow phenomena. This would otherwise require expensive and time-consuming experimental work.

What is certain is that such flow effects cannot be captured using empirical 1-D or 2-D flow equations, although the use of these methods can still be justified for pre-optimization in the early design phases. The CFD code should be looked at together with other preliminary design tools as part of an integrated system for evaluating design candidates. The investigation supports ABB's accumulated experience with CFD codes for fan design work by confirming that the accuracy of the numerical flow calculations – even for complete fans - is sufficient to justify using CFD routinely for matching machine components in a wide range of applications.

Authors

Dr. Wilfried Rick
Dr. Eribert Benz
ABB Corporate Research Center AG
P.O. box 5102
CH-5405 Baden-Dättwil
Switzerland
Telefax: +41 56 486 7359
wilfried.rick@ch.abb.com
eribert.benz@ch.abb.com

Alain Godichon
ABB Solyvent-Ventec SNC
P.O. box 67
F-69882 Meyzieu-Cedex
France
Telefax: +33 3 854 17322
alain.godichon@fr.abb.com

References

[1] **I. Bennett, M. Vezza**: Flows within industrial centrifugal fans. VDI reports no. 1249, 1996.
[2] **W. Rick**: CFD analysis of the radial fans DXW, DZW. Internal ABB report, ABB CHCRC, Sept 1998.