Journal of Computational and Applied Mechanics, Vol. 5., No. 2., (2004), pp. 251–261

NUMERICAL AND EXPERIMENTAL INVESTIGATIONS OF CROSS-FLOW FANS

Martin Gabi and Toni Klemm
Universität Karlsruhe (TH)
D-761289 Karlsruhe, Germany
martin.gabi@mach.uka.de

[Received: November 12, 2003]

Abstract. The prediction of performance characteristics and flow field data of cross-flow fans (CFF) is difficult due to the complex flow structure and working principle. Numerical methods will become more important in order to get a better knowledge of the complex flow-phenomena in CFF and will lead to a reduction of experimental work in the process of development and construction. CFD calculations are carried out to compute the flow field and the performance of CFF. For the validation of the numerical computations, experimental investigations are made to measure flow field and performance data. In the flow field measurements mainly Particle Image Velocimetry (PIV) is used.

Keywords: cross-flow fan, CFD, particle image velocimetry

1. Nomenclature

D_2	$[m]$	impeller outer diameter
D_1	$[m]$	impeller inner diameter
L	$[m]$	blade length
M	$[Nm]$	rotor torque
n	$[s^{-1}]$	rotation speed
Δp	$[Pa]$	pressure difference
Q	$[m^3/s]$	volume flow rate
R_e	[-]	Reynolds number
S	$[m]$	blade chord length
U_2	$[m/s]$	peripheral speed
V	$[m/s]$	velocity (local or global)
φ	[-]	volume flow coefficient
ψ	[-]	pressure coefficient
ν	$[m^2/s]$	kinematic viscosity
ρ	$[kg/m^3]$	fluid density
ω	$[s^{-1}]$	angular velocity

2. Introduction

Cross-flow fans (CFF), as per Figure 1, are used in cooling, heating, air conditioning systems, in automotive applications as well as in industrial applications e.g. for heating and drying purposes.

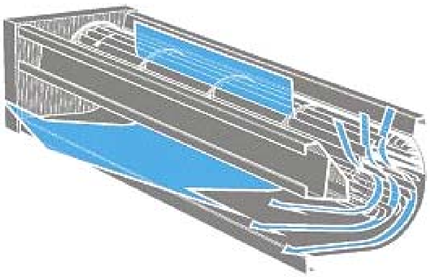

Figure 1. Typical construction of a CFF

They consist of a broad cylindrical rotor with many forward curved blades of a high ratio of diameter $D1/D2$. The air passes the blade grid twice, driven by a large vortex, the steering vortex, situated inside and outside of the impeller, near the so called stabilizer, see Figure 2 and 3.

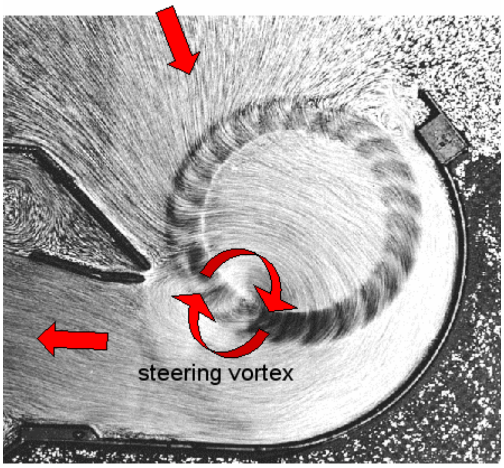

Figure 2. Flow field

Position, size and intensity of this vortex is strongly influenced by geometrical parameters of the casing and the point of operation (volume flow rate and pressure difference) of the CFF [1] .

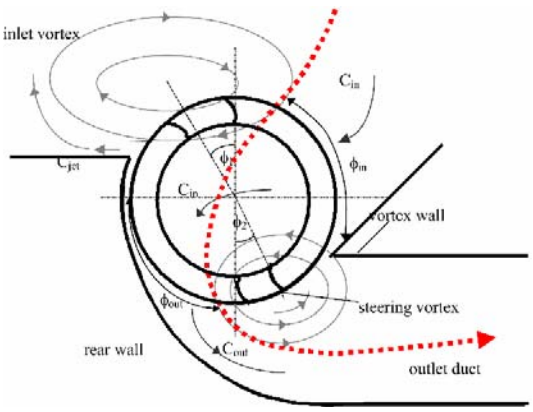

Figure 3. Global flow pattern

Depending on geometry and operating conditions also secondary vortices in the inlet and outlet zones may appear. The flow field and the performance characteristics of a CFF depend much more significantly on the shape of the surrounding casing than on the design of the rotor and the blade grid. Because of the principle of operation, the performance characteristics of a CFF is characterized by high volume flow and pressure coefficients, in comparison to other fan types, see Figure 4.

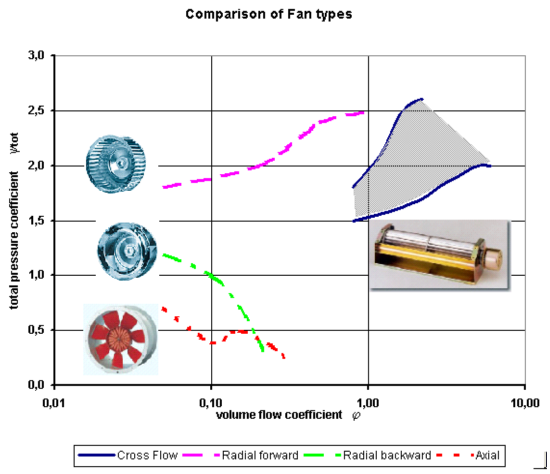

Figure 4. Comparison of fan types

Due to the complex flow structure, the design process is still mainly based on numerous experiments, because classical design methods are not applicable to a CFF in a satisfying way. It is helpful to use numerical methods for getting a better impression of the complex flow phenomena of a CFF and to reduce the number of experiments in the process of design, development and construction. Therefore CFD calculations are used to compute the flow field and the performance characteristics of CFF [4].

In order to validate and prove the numerical computations, experimental investigations are made to measure the flow field and the performance characteristics. For the flow field measurements mainly the Particle Imaging Velocimetry (PIV) method is used [3, 5].

3. Numerical simulation

The object of the numerical simulations is to get a better impression of the flow field in a CFF, particularly inside the impeller. The results of these numerical investigations should be the basis to develop guidelines for the design and construction of efficient CFF. The main objective is to develop numerical procedures to obtain the flow variables determining the characteristics of CFF, such as pressure coefficient and efficiency vs. volume flow coefficient as well as velocity and pressure fields.

In the literature only few papers dealing with CFD calculation of the flow in CFF are published. Mainly the flow is considered to be two-dimensional, inviscid and steady. The impeller usually is modelled by an infinite number of thin blades. More recent studies try to take into account transient and viscous effects, but need the calibration of constants used in the mathematical modelling using experimental investigations. Today the development of hardware and calculation methods allows calculation of the complete flow inside the CFF. Most of the commercial CFD software packages are based on finite volume methods (FVM). The FVM is also implemented in the software package STAR-CD, which is used for the present investigations [6].

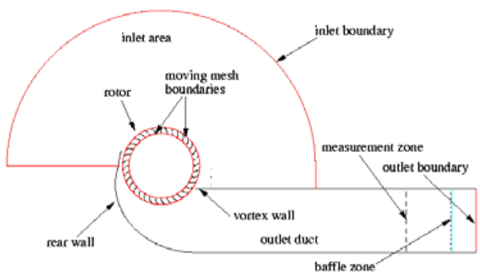

Figure 5. Computational domains and boundaries

Figures 5 and 6 show computational domains and boundaries. The model of Figure 5 shows a very simple configuration, which is very well documented in literature [2]. Figure 6 shows the typical configuration of a frequently used geometry with a 90 degree deflection. Because of the distinctive transient character of the flow through the impeller, it is necessary to use a transient method in the impeller domain. The link between the rotating impeller domain and the casing domain is realised by a sliding interface.

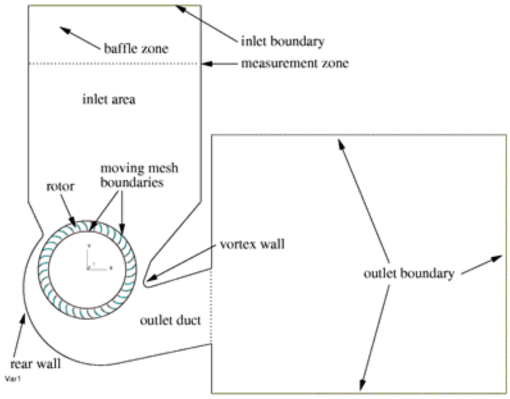

Figure 6. Computational domains and boundaries

An 'arbitrary sliding mesh method' (ASI), which is implemented in STAR-CD, is used to simulate the movement of the different mesh regions. The 'sliding' involves a continuous change in connectivity for cells on each side of the interface. Due to an indirect addressing method, the two sides of the interface remain coupled implicitly during this process. Appropriate interpolations in time and space preserve flux continuity across the interface and avoid the production of perturbations to the flow field. It is assumed, that in a broad CFF-impeller the flow can be taken as two-dimensional with negligible changes perpendicular to the main flow direction. The influences of the side walls of the casing and the end and intermediate discs of the impeller are neglected. The $k-\varepsilon$ turbulence model is used for turbulence modelling.

Flow characteristics and performance of CFF are usually described by the following dimensionless parameters:

$Re_S = \frac{U_2 S}{\nu}$	Reynolds-number (chord length)
$Re_D = \frac{U_2 D_2}{\nu}$	Reynolds-number (global)
$\varphi_c = \frac{Q}{L D_2 U_2}$	volume flow coefficient (CFF)
$\varphi = \frac{4Q}{\pi D_2^2 U_2}$	volume flow coefficient (global)
$\psi_s = \frac{\Delta p_{stat}}{\frac{1}{2}\rho U_2^2}$	stat. pressure coefficient

4. Experimental investigations

4.1. **Test installation.** Planning the experimental investigations, measurements of the global performance data pressure rise, volume rate and input torque, as well as velocity distributions have to be provided. A test rig was set up as a closed circuit channel, which contains all installations to control and measure the volume rate, the pressure rise and the input power of the test CFF. The principle of the channel is similar to a standard airway for fans [7]. Figure 7 shows the view of the complete test channel from the top.

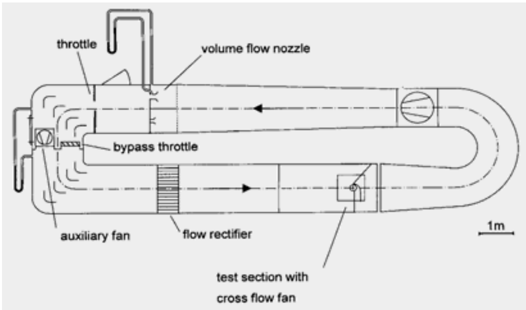

Figure 7. Test installation

4.2. **PIV measurements.** Figure 8 shows the details of the test installation for the CFF with the PIV system. Due to the complex geometry, several transparent wall sections are realized for the laser light sheets. The PIV measurements were carried out in the inlet and outlet zones, in a representative plane perpendicular to the axis of rotation. Inside the impeller, PIV measurements cannot be performed actually.

The global performance data are measured at positions which correspond to the analysis of the numerical data.

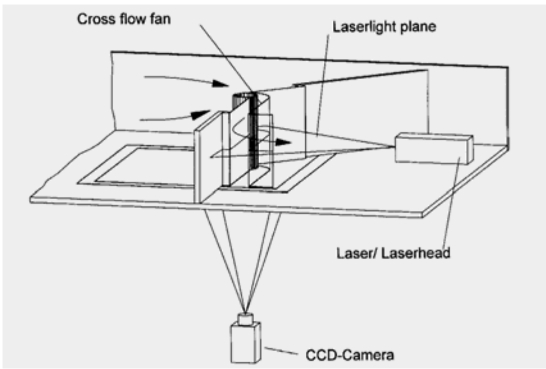

Figure 8. PIV measurements

5. **Results**

5.1. **Test case.** This geometry was investigated experimentally by Tuckey et al. [2]. The CFF is designed as the air supply for an open-circuit wind tunnel. The resulting rotor parameters and operating conditions are shown in Table 1, the geometrical values correspond to Figure 9. This table also shows the dimensions and the operational conditions of the model fan in the test installation. The geometrical scale factor is $0,16$.

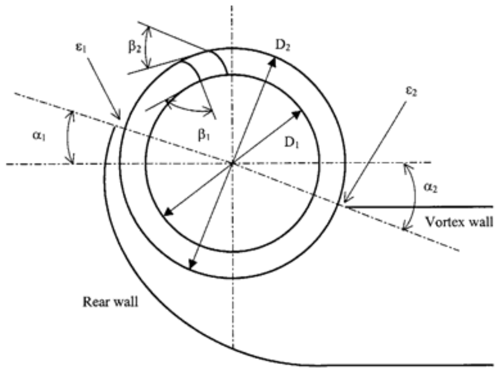

Figure 9. Test geometry [2]

Figure 10 represents the results of measurements of the static pressure coefficient vs. the volume rate coefficient of both fans.

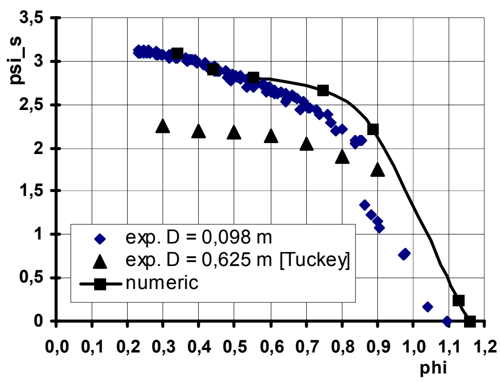

Figure 10. Comparison of experimental and numerical results

The fact that the pressure coefficient of the model is higher than that of the large fan in the left part of the characteristics may be caused by the higher number of blades.

Table 2. Geometric and operating data

Literature [2]:	
impeller diameter D_2	$0.625\,m$
diameter ratio D_1/D_2	0.78
blade angles	$\beta_1 = 90^o,\ \beta_1 = 26^o$
number of blades	24
chord length S	$0.084\,m$
blade profile	circular arc
rotating speed	$n = 6.67\,sec^{-1}$
Re (chord length)	$Re_S = 70.000$
Re (global)	$Re_D = 550.000$
volume flow coefficient	$0.2 < \varphi < 0.9$
stat. pressure coefficient	$1,6 < \psi_s < 2,2$
Model:	
impeller diameter D_2	$0.098\,m$
diameter ratio D_1/D_2	0.65
blade angles	$\beta_1 = 90^o,\ \beta_1 = 26^o$
number of blades	36
chord length S	$0.01\,m$
blade profile	circular arc
rotating speed	$n = 25\,sec^{-1}$
Re (chord length)	$Re_S = 5.000$
Re (global)	$Re_D = 50.000$
volume flow coefficient	$0,3 < \varphi < 1,1$
stat. pressure coefficient	$0 < \psi_s < 3$

The line marks the results of the numerical calculations. Corrections for 3D losses, caused by friction of the discs and volumetric losses in the lateral gaps are not taken into consideration during the numerical calculations.

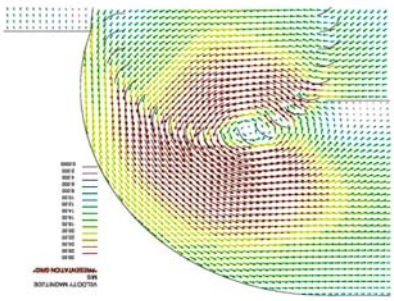

Figure 11. Velocity field, numeric

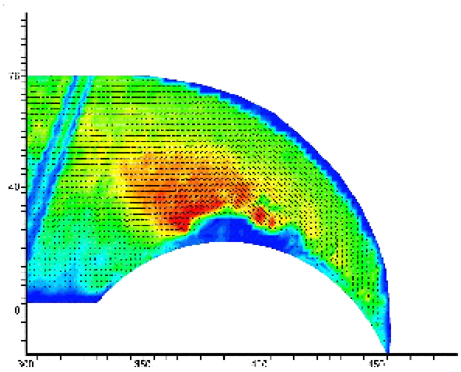

Figure 12. Velocity field, PIV

Figure 11 shows a calculated velocity field, which gives a good impression of the steering vortex. It corresponds with the PIV measurements in Figure 12 in a satisfactory way.

5.2. **Variation of operational conditions.** Within these investigations a variation of the point of operation is calculated and measured for a typical CFF configuration with a 90 degree deflection. Figures 13 and 14 show the CFD results for two different values of the volume flow coefficient. In Figures 15 and 16 the corresponding flow fields measured with PIV are shown. Calculated and measured flow fields, especially the formation of the steering vortex are in good agreement.

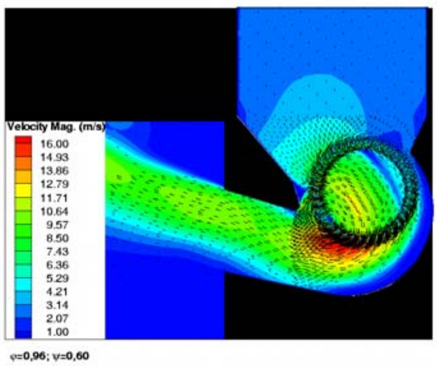

Figure 13. Velocity field, calculated, $\varphi = 0,96$

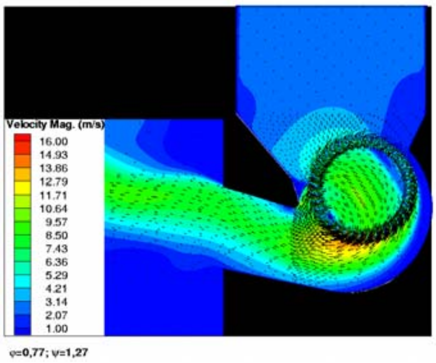

Figure 14. Velocity field, calculated, $\varphi = 0,77$

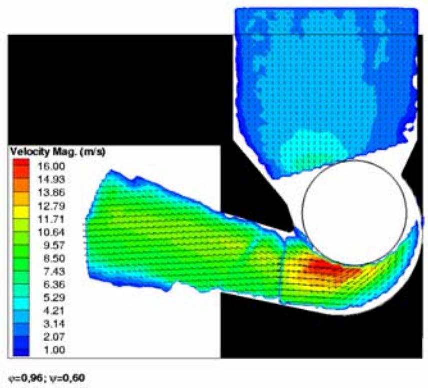

Figure 15. Velocity field, PIV, $\varphi = 0,96$

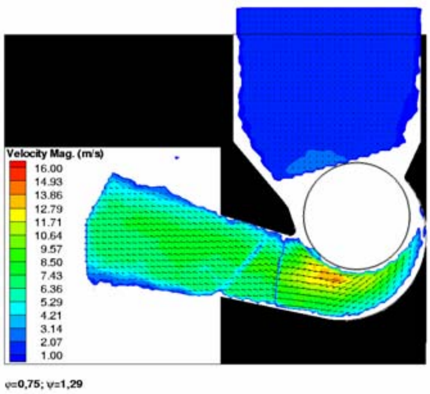

Figure 16. Velocity field, PIV, $\varphi = 0,75$

6. **Summary**

PIV measurement techniques in combination with CFD are useful tools to understand the flow fields in fluid machinery. Particularly with regard to CFF, whose functional principle cannot be described sufficiently with the common theories of fluid machinery, these techniques can help in the process of design and dimensioning. Further investigations will allow a better look into the zones more immediate to the impeller. It is also planned to arrange PIV measurements inside the impeller to detect vortices more exactly.

Acknowledgement. This project is initiated and promoted by the German Fan Manufacturer's Research Association (FLT).

REFERENCES

1. Eck, B.: *Ventilatoren.* Auflage, Berlin: Springer-Verlag, 5, 1972.
2. Tuckey, P.R., Holgate, M.J. and Clayton, B.R.: *Performance and aerodynamics of a cross flow fan design & applications.* Guildford, England, 1982.
3. Willert, C.E., Raffel, M. and Kompenhans, J.: *Particle image velocimetry, A practice guide*, Springer-Verlag, Germany, 1998.
4. Dornstetter, S.: *Numerische und experimentelle Untersuchungen an Querstromventilatoren*, Dissertation Universität Karlsruhe(TH), 2002.
5. Gabi, M., Dornstetter, S. and Klemm, T.: Investigation of the flow field in cross-flow fans by particle imaging velocimetry, Proc. 10th Int. Symp. On Flow Visualisation, Kyoto, Japan, 2002.