Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science

<http://pic.sagepub.com/>

Analytical and numerical investigation of the optimum pressure distribution along a low-pressure axial fan blade

M Pascu, M Miclea, P Epple, A Delgado and F Durst

Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science 2009 223: 643

DOI: 10.1243/09544062JMES1023

The online version of this article can be found at:

<http://pic.sagepub.com/content/223/3/643>

Published by:

<http://www.sagepublications.com>

On behalf of:

[Institution of Mechanical Engineers](#)

Additional services and information for *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science* can be found at:

Email Alerts: <http://pic.sagepub.com/cgi/alerts>

Subscriptions: <http://pic.sagepub.com/subscriptions>

Reprints: <http://www.sagepub.com/journalsReprints.nav>

Permissions: <http://www.sagepub.com/journalsPermissions.nav>

Citations: <http://pic.sagepub.com/content/223/3/643.refs.html>

>> [Version of Record](#) - Mar 1, 2009

[What is This?](#)

Analytical and numerical investigation of the optimum pressure distribution along a low-pressure axial fan blade

M Pascu^{1*}, M Miclea¹, P Eppe¹, A Delgado¹, and F Durst²

¹Institute of Fluid Mechanics LSTM, Friedrich-Alexander University, Erlangen-Nuremberg, Erlangen, Germany

²Fluid Measurements and Projects FMP Technology GmbH, Erlangen, Germany

The manuscript was received on 28 January 2008 and was accepted after revision for publication on 11 July 2008.

DOI: 10.1243/09544062JMES1023

Abstract: In the field of axial flow turbomachines, the two-dimensional cascade model is often used experimentally or numerically to investigate fundamental flow characteristics and overall performance of the impeller. The core of the present work is a design method for axial fan cascades aiming to derive inversely the optimum blade shape based on the requirements of the impeller and not using any predefined aerofoil profiles.

While most design strategies based on the aerofoil theory assume constant total pressure at all streamlines, i.e. free-vortex flow, this paper investigates the possibility of varying the total pressure along the blade and based on that, an analytical expression of the outlet blade angle is determined. When computing the blade profile at a specified radius, critical parameters reflecting on the flow characteristics are observed and adjusted (i.e. sufficient lift and controlled deceleration of the flow on the contour) so that the resulting profile is derived for minimum losses.

The validation of this design strategy is given by the numerical results obtained when employed as an optimization tool for an industrial fan: 10–20 per cent absolute increase in the static efficiency of the optimized impeller.

Keywords: non-free-vortex flow, optimum pressure distribution, design strategy, blade profile

1 INTRODUCTION

The rapid design of a blade that performs well and satisfies machining requirements is one of the goals in designing fluid machinery blades, and computational fluid dynamics (CFD) is nowadays used as a common tool to predict the performance of such devices. In the field of turbomachinery, the design of new blades can be achieved through direct or inverse design methods. While the direct methods are frequently used to predict the performance of the impeller under design and off-design conditions by solving Reynolds averaged Navier–Stokes equations [1], the inverse methods compute the geometry of the blades according to design specifications of flow features given as input and the necessary boundary conditions are given empirically, according to statistical data [2].

Of high complexity in calculation are the three-dimensional inverse methods that are formulated with different choices of prescribed quantities. The first would be the pressure distribution along the blade suction and pressure surfaces [3]. The preferred method, however, uses the blade pressure loading (the static pressure difference between the suction and pressure surfaces) and blade thickness distribution, resulting in the calculation of the mean camber line [4]. Other fully three-dimensional inverse methods compute the mean through-flow [5] or make intensive use of the algorithms based on the potential theory to predict flow instabilities [6].

In the field of axial fans, the major issue is designing high-efficiency fans at a given flowrate and for given pressure-duties, and in this case, design techniques are typically based on engineering experience and may involve much trial and error before an acceptable design is found. The most common approach is making use of the aerofoil theory, which is invariably associated with the free-vortex flow energy distribution along the radius, i.e. the blade profiles are designed for the same total pressure in span-wise

*Corresponding author: Institute of Fluid Mechanics LSTM, Friedrich-Alexander University, Erlangen-Nuremberg, Cauerstrasse 4, Erlangen, 91058, Germany.
email: maria.pascu@lstm.uni-erlangen.de

direction. The work of Wallis [7] is also concerned with such design assumption: the considered system is of the free-vortex flow type and several important parameters (e.g. lift/drag ratio) are fixed at reasonable values. Similarly, by employing the free-vortex flow, Dugao *et al.* [8] achieved considerable improvement in the efficiency of the fan, with noise reduction as a second advantage.

However, even though this assumption is preferred by most designers, there is nothing in the aerofoil theory to prevent any desired pressure distribution along the blade [9] and combining an optimization algorithm with the arbitrary vortex flow might enable the designer to investigate a larger range of design alternatives in an efficient manner. In this sense, the work of Sørensen *et al.* [10, 11] is mainly concerned with employing an arbitrary vortex flow model for fan analysis, allowing the investigation of the fan under various operating conditions. The advantage of employing such a design assumption in the sense of better fan performance throughout a larger range of flows was shown. Also by employing the non-free-vortex flow analysis at an early design stage, Albuquerque *et al.* [12] developed a model found effective in the optimization of axial flow hydraulic turbines.

In the field of axial fans, a major challenge is constituted by the large range of geometries available and their fixed use, and often, the design methods in this area are developed for a single class of application. Therefore, it seems desirable to develop a design strategy that, for a specified fan application, determines if improvements in the performance can be achieved by employing the free-vortex flow or some variation is better suited. Hence, the motivation of the present work is to determine the optimum pressure distribution along an axial fan blade, by considering both free and non-free-vortex flow assumptions and parameterizing the solution according to the fan specifications.

The initial design considerations and the core of the elaborated design strategy are stated in section 2, the derivation of the optimized profiles is presented in section 3, and in sections 4–6, the results obtained when optimizing a reference impeller (i.e. an axial fan of a hub ratio of 0.5 currently used in the automotive industry for cooling purposes) are presented.

2 OPTIMUM BLADE PROFILE

2.1 Theoretical considerations of the flow motion in an axial impeller

The complex three-dimensional flow in axial impellers (where the flow particles enter and leave the impeller at the same radius, i.e. constant peripheral velocity) can be treated as the superposition of a number of

Fig. 1 Sketch of the two-dimensional cascade according to Lewis [13]

two-dimensional flows that lead to more manageable blade design and profile selection techniques, i.e. two-dimensional cascades. If a cylindrical cut is made through the impeller at specified radius and the cylinder is then developed onto a plane, a row of blade profiles will result [13], as depicted in Fig. 1.

The advantage of this simple approach is that the Euler equation for turbomachinery can be applied to each cascade section independently to determine the inlet and outlet velocity triangles for that particular blade section

$$\Delta p_t = \frac{\rho}{2} \left[\underbrace{(w_1^2 - w_2^2)}_{\text{static component}} + \underbrace{(c_2^2 - c_1^2)}_{\text{dynamic component}} \right] \quad (1)$$

The first design assumption that one can make, confirmed by both experiments and CFD calculations, is that at the design stage, axial entry of the fluid can be assumed and this is shown in the inlet velocity triangle in Fig. 2 (zero pre-swirl upstream of the impeller and hence $w_{1u} = u$). Based on this assumption, the expression of the inlet blade angle can be determined as

$$\tan \gamma_1 = \frac{w_m}{w_{1u}} \quad (2)$$

Further analysis of the cascade pressure difference in equation (1) yields to

$$\begin{aligned} \Delta p_t &= \frac{\rho}{2} (w_{1u}^2 - w_{2u}^2 + c_{2u}^2) \\ &= \frac{\rho}{2} [u^2 - w_{2u}^2 + (u - w_{2u})^2] \\ &= \rho u (u - w_{2u}) = \rho u c_{2u} \end{aligned} \quad (3)$$

2.2 Blade angle distribution

In the previous paragraph, axial entry of the fluid in the impeller was assumed. This consideration makes

Fig. 2 Inlet (for axial entry) and outlet velocity triangles at specified blade section

the derivation of the inlet blade angle simple

$$\tan \gamma_1 = \frac{w_m}{u} \frac{Q}{2A\pi r n} \quad (4)$$

For the determination of the outlet blade angle, one should come back to the purpose of the design, namely to maximize flow and minimize losses. When further deriving the expression of the static component of the pressure difference for one profile cascade in equation (1), then

$$\begin{aligned} \Delta p_s &= \frac{\rho}{2} (w_{1u}^2 + w_{1m}^2 - w_{2u}^2 - w_{2m}^2) = \frac{\rho}{2} (w_{1u}^2 - w_{2u}^2) \\ &= \frac{\rho}{2} (u^2 - w_{2u}^2) = \frac{\rho}{2} \left(u^2 - \frac{w_m^2}{\tan^2 \gamma_2} \right) \\ &= \frac{\rho}{2} \left(u^2 - \frac{Q^2}{A^2} \frac{1}{\tan^2 \gamma_2} \right) \end{aligned} \quad (5)$$

If referring to the system characteristic then maximum flowrate can be achieved for $\Delta p_s = 0$. Hence

$$u^2 - \frac{Q_{\text{MAX}}^2}{A^2} \frac{1}{\tan^2 \gamma_2} = 0$$

and this immediately yields to

$$Q_{\text{MAX}} = uA \tan \gamma_2 \quad (6)$$

When analysing equation (6), it can be readily observed that the maximum flowrate will occur for an angle of 90° (when $\tan \gamma_2 \rightarrow \infty$).

Analysing the outlet velocity triangle, one can immediately notice that $\gamma_2 = 90^\circ$ implies $w_{2u} = 0$ and $c_{2u} = u$. Since c_{2u} represents the swirl component of the absolute flow at the exit of the cascade, a good design should limit this component because it is a direct measure of the cascade losses. Moreover, through the deceleration of this component, a recovery of the static pressure can be achieved. Since the hub section has the highest work coefficient (loading) and the profile should not turn more than axial here, the outlet

angle of 90° is best suited. For the other blade sections, i.e. middle and tip, the outlet blade angle will be determined based on this assumption, as shown in the following paragraph.

2.3 Pressure distribution along the blade and its influence on the blade shape

As already mentioned in the introductory paragraph, most design methods for axial fans assume that the same total pressure rise is applied at all sections along the blade, i.e. the free-vortex flow. Considering various sections of the blade, hub, and an intermediary section, as indicated in Fig. 3(a), this can be written as

$$\frac{\Delta P_{t,r}}{\Delta P_{t,h}} = 1 \quad (7)$$

According to Carolus [14], employing such an assumption introduces, from the design stage, a correct blade loading, at the designated section. A proper blade loading at the tip section (where the radius is larger) demands considerable smaller values of c_{2u} (responsible for the swirl) than at the hub section (where the radius is smaller). This means that close to the hub the absolute flow, characterized by higher c_{2u} , has to be redirected to the tip section and thus keeping the product $c_{2u}r$ constant. The desired effect of such consideration is that the flow detachments will occur earlier, and hence, the hub section will be more stressed than the tip.

This assumption is directly connected to the swirl component at the exit of the cascade and it is employed whenever aerofoil profiles are used in designing an axial flow machine. However, there is nothing in the aerofoil theory to prevent the desired head distribution along the blade radius.

By changing the pressure distribution along the blade (and thus the outlet blade angle at the corresponding radius), designs providing higher overall performances may be achieved with small impact on

Fig. 3 (a) Two-dimensional sketch of an axial impeller; (b) view of specified blade section in (x, y) coordinate system

the dimensions of the swirl. Hence, it is the designer's task to investigate, for the specified design parameters and operating conditions, the optimum pressure distribution that balances the efficiency and the swirl component at the same time.

For this it was assumed that starting from the hub and advancing to the tip, the Eulerian pressure difference variation can be expressed as a function of the radius, as indicated below

$$\frac{\Delta P_{t,r}}{\Delta P_{t,h}} = f(r) \quad (8)$$

When employing the expression of the total pressure difference indicated by equation (3), a recurrent relationship between the outlet blade angles, at different blade sections, can be determined

$$\begin{aligned} f(r)2\pi r_h n \left(2\pi r_h n - \frac{w_m}{\tan \gamma_{2h}} \right) \\ = 2\pi r n \left(2\pi r n - \frac{w_m}{\tan \gamma_{2r}} \right) \\ r \frac{w_m}{\tan \gamma_{2r}} = 2\pi n [r^2 - f(r)r_h^2] + f(r)r_h \frac{w_m}{\tan \gamma_{2h}} \\ \frac{1}{\tan \gamma_{2r}} = \frac{2\pi n}{r w_m} [r^2 - f(r)r_h^2] + f(r) \frac{r_h}{r} \frac{1}{\tan \gamma_{2h}} \end{aligned} \quad (9)$$

Equation (9) can be further simplified with another assumption made in the previous section, namely that at the hub, the outlet blade angle is 90°

$$\frac{1}{\tan \gamma_{2r}} = \frac{2\pi n}{r w_m} [r^2 - f(r)r_h^2] \quad (10)$$

It can be noticed in equation (10) that the expression of the outlet blade (metal) angle can be parameterized according to the prescribed variation of the total pressure difference in span-wise direction. The use of such formulation can be further extended if the equation can be expressed in terms of relevant geometrical parameters for the design process, such as the hub ratio.

The actual application of such impellers (engine cooling fans) is an important factor for the parameterization of the pressure variation, since such devices are normally low-pressure high-flow delivering machines. Hence, the pressure should not be increased to unrealistic values in the range of which the impeller will never operate anyway. Also rapid variations that might result from employing exponential or higher order polynomial laws should be avoided.

Another major constraint in the parameterization process of the function $f(r)$ is the immediate impact that this expression has on the value of the outlet blade angle. The angle, at the specified blade radius, delivered by $f(r)$ should be a realistic value (no negative values) and should be incorporated in the geometry of the full blade, i.e. to respect the trend imposed by the neighbouring sections.

Furthermore, it was the authors' aim to derive an expression in which, by changing one factor, the desired pressure variation along the blade can be achieved.

Solving all these constraints simultaneously finally led to the following expression for the function $f(r)$

$$f(r) = x \left(\frac{r_{\text{tip}}}{r_{\text{hub}}} \right)^x (r - r_h)^{1.35} + 1 \quad (11)$$

The parameter to be changed during the design process is x . It can be immediately noticed that for $x = 0$ the classical assumption of the constant pressure (free-vortex flow) is employed, since $f(r) = 1$. For $x = 1$, the pressure variation is linear with the inverse of the hub ratio, for $x = 2$ parabolic, and so on.

Equation (10) then becomes

$$\frac{1}{\tan \gamma_{2r}} = \frac{2\pi n}{r w_m} \left\{ r^2 - \left[x \left(\frac{r_{\text{tip}}}{r_{\text{hub}}} \right)^x (r - r_h)^{1.35} + 1 \right] r_h^2 \right\} \quad (12)$$

The expression in equation (12) allows the determination of the outlet blade angle, at the specified radius, based on a pressure variation from hub to tip.

Essentially, this variation has a major influence on the resulting blade shape, and as a consequence, an optimum profile demands a proper pressure prescription. Hence, several test cases of axial fan designs will be investigated, including the classical constant pressure assumption, in order to determine, for specified dimensions and operating conditions, the optimum pressure variation in span-wise direction, which delivers high efficiency and small losses due to the swirl component.

2.4 Blade shape computation

In a cascade, the action of the fluid on the profile can be considered similar to that taking place on an aerofoil in a wind tunnel, provided that the velocity of the undisturbed flow w_∞ is an average of the inlet and outlet relative velocities [15].

The ratio of the blade chord length to the blade spacing l/t is an important design parameter, being an index of the solidity. The chord spacing ratio generally increases from tip to hub due to aerodynamical reasons. When attaching a coordinate system to the profile cascade, the blade profile, i.e. blade shape, will have the aspect indicated in Fig. 3(b). Basically, the blade shape at a specified section can be readily computed if imputing the y coordinates and angle distribution $\gamma(y)$, since

$$\tan \gamma(y) = \frac{\Delta y}{\Delta x} \quad (13)$$

To effectively apply any driving action on the fluid, the blade angle is increased from γ_1 to γ_2 . The difference between the two ($\gamma_2 - \gamma_1$) is a measure of the blade curvature along any given blade section. The increase from γ_1 to γ_2 can be estimated by any type of variation, either linear or higher degree polynomial. According to Pascu and Epple [16], for axial fans a parabolic distribution of the blade angles is appropriate. It was shown by means of streamline analysis that this assumption, made at early design stage, was confirmed by the actual flow angles, thus achieving very good flow-blade congruency.

When considering such a distribution $\gamma(y) = Ay^2 + By + C$

$$\begin{aligned} y = y_1 &\rightarrow \gamma = \gamma_2 \\ y = y_m &\rightarrow \gamma = \gamma_m \\ y = y_2 &\rightarrow \gamma = \gamma_1 \end{aligned} \quad (14)$$

The blade shape computation is now carried out with one degree of freedom given by (γ_m, y_m) . The geometry is iterated until the axial chord constraints are matched as well as corrected for abrupt flow parameters. The entire procedure for computing the blade

Fig. 4 Summarized derivation of the blade shape

shape at specified radius can be summarized for better understanding as presented in Fig. 4.

2.5 Further design assumptions based on the profile theory

Considering the flow over a profile, the fluid approaches the profile from upstream with the velocity w_1 at an angle γ_1 and leaves the profile with the velocity w_2 at an angle γ_2 , as indicated in Fig. 5.

In Fig. 5, ε is defined as the drag to lift ratio. In the literature, ε is often referred as the 'gliding' angle of the profile and several optimum values are proposed based on extensive experimental results [17]. The value of ε is an important design choice since it denotes the impeller losses by friction. Compared to the ideal case of the friction-less flow, when $\varepsilon = 0$, the drag force introduced by ε in the direction of w_∞ (Fig. 5) calls for a loss in the pressure difference. Returning again to the application of the design, it is the design choice of the authors to initially consider $\varepsilon = 0$, since the computed profiles should not create additional losses in the flow around them. The next design assumption is that, at all points on the computed profiles, flow-blade congruency is achieved, i.e. flow and computed blade angles are identical and this implies zero incidence and deviation angles.

Another design parameter often used in the profile theory is the camber angle, θ . Basically, the result of the design solver, as indicated in Fig. 4, is the camber line, and by attributing some thickness around it, a

Fig. 5 Flow over a profile cascade

Fig. 6 Performance curves for a constant thickness blade and an airfoil, Eckert [18]

profile is obtained. The thickness distribution is again an important design choice and one has to carefully weigh if profiling really pays. Most design methods for axial fans use a variable thickness for the profile, according to the thickness distribution functions existent in the literature, i.e. NACA and C4 British series [19]. However, such methods were originally proposed for axial flow water turbines where the operating regime requires high pressures. For axial fans, where the pressures are in the range of thousands of Pascals, a variable thickness along the camber is unnecessary since, according to Eckert [18], when measuring two impellers, one with aerofoil profiles and the other one with constant thickness, their performance is identical, as shown in Fig. 6.

Hence, for the calculated profiles, constant thickness will be applied, thus introducing the last assumption at design stage, i.e. due to constant thickness distribution, there will be no point of maximum thickness, and hence, the camber angles, at all points on the profile, will coincide with the computed blade angles.

3 CAD MODELS OF THE NEW DESIGNS

An alternative design method for axial flow impellers has been presented so far. The method is based on computing the appropriate blade profile at each section of the impeller, so that optimum flow conditions are achieved. Essentially, the blade is computed as a succession of several cascades characterized by different radius and chord length and the profiles at different radii are derived based on the variation of the pressure along the blade. Hence, calculating the optimum blade profile is directly dependent on the determination of the optimum pressure variation,

according to the specified dimensions and operating conditions of the impeller.

In order to determine the optimum pressure distribution, several new designs are derived by simply varying the term x in equation (12):

Design I: $x = 0$ (corresponds to the classical assumption of free-vortex flow and hence constant pressure at all radii is assumed)

Design II: $x = 1$

Design III: $x = 2$

During the preparation of the present work, the CAD model of a reference impeller currently used as engine cooling ventilator in the automobile industry was available, and for optimization purposes, all the new designs were derived for the same dimensions and design parameters as this impeller:

- (a) hub radius, $r_{\text{hub}} = 147$ mm;
- (b) tip diameter, $r_{\text{tip}} = 280$ mm;
- (c) rotational speed, $n = 3000$ r/min;
- (d) design flowrate, $Q_{\text{design}} = 4$ m³/s;
- (e) number of blades, $z = 8$.

The new designed blades were obtained by computing the blade shape according to the prescribed pressure distribution for several cascades: hub, tip, and two intermediary sections, namely 187 and 227 mm. The solidity of the reference is variable with the radius, as indicated in Fig. 7, and it is kept constant from one design to the other.

The computer camber lines, for each pressure distribution, are presented in Fig. 8.

Obviously, for the hub section, where the outlet angle is always fixed to 90°, the aspect of the camber will not change no matter the pressure distribution. The differences in the computed cambers appear from the following sections, inducing a difference between the prescribed angles as well, as indicated in Fig. 9.

Constant thickness of 3.2 mm was applied to all models. Three impellers were generated, each having

Fig. 7 Variation of the solidity with the radius

Fig. 8 Differences in the computed camber lines for different pressure distributions

Fig. 9 Variation of the prescribed angle distributions with different pressure variations

Fig. 10 CAD model of the impeller

eight blades and corresponding to the dimensions indicated previously. The three impellers were placed inside of pipe and the tip clearance was of ≈ 10 mm. To bring the simulation results closer to the real-life operating conditions, two extra components were added to the CAD model: at the inlet of the impeller a rig section long enough so that the flow entering the impeller domain can be considered fully developed (the length of the rig section was chosen $3D_{\text{pipe}}$); and at the outlet (ambient), another pipe with the length of $2D_{\text{pipe}}$ was included. An exemplary depiction of the CAD models is shown in Fig. 10.

4 MESH GENERATION AND NUMERICAL SIMULATIONS

The grid (Fig. 11) was generated in ANSYS ICEM 11.0 (<http://ansys.com/>) using tetrahedral mesh and its size was around 400 000 nodes for the flow domain corresponding to one blade. The general mesh size for

the domain fan was set to 8, with a finer size around the blade tips and leading and trailing edges (2), and the suction and pressure surfaces of the blade (4). To resolve more accurately the flow at the wall portion, prismatic layers were employed. A coarser mesh was applied for the flow domain around the impeller and the pipe walls.

The numerical simulations were carried out with the commercial code ANSYS CFX 11.0. The simulation type was set initially to be *steady*. The fluid flow was set to be viscous (air properties as an ideal gas were considered). The flow was solved with the Navier–Stokes equations, assuming conservation of mass, momentum, and energy.

Typical values for the Reynolds number, calculated as $Re = w_m D_{\text{tip}} / \nu$ are around 800 000. One of the main problems in turbulence modelling is the accurate prediction of flow separation from a smooth surface. Standard two-equation turbulence models often fail to predict the onset and the amount of flow separation under adverse pressure gradient conditions. To avoid this problem, the model used in the present computations of the air flow is the shear stress transport model [20]. The model works by solving a turbulence–frequency-based model ($k-\omega$) at the wall and $k-\varepsilon$ model in the bulk flow. A blending function ensures smooth transition between the two models. The interface between the different frames of reference is taken to be a frozen rotor general grid interface (GGI). The frozen rotor model has the advantage of being robust, using less computer resources than the other frame change models, e.g. stage interface model, which is not suitable for applications with tight coupling of components and/or significant wake interaction effects and may not accurately predict

Fig. 11 Detail of the generated mesh

loading. The frozen rotor model treats the flow from one component to the next by changing the frame of reference while maintaining the relative position of the components. This model must be used for non-axisymmetric flow domains, such as impeller/volute or classifier/casing.

Inlet boundary with specified mass flow was applied to the rig domain, flow direction normal to the boundary condition, and medium turbulence intensity (5 per cent). A rotational speed of 3.000 r/min was applied to the fan domain. The outlet boundary was applied to the ambient domain by specifying a 25 °C temperature (air density 1.05 kg/m³). Typical Mach numbers were ~0.25 (it is known that for Mach numbers <0.3, the fluid can be considered as incompressible [21]). Several simulations were carried out for all models by varying the flowrate at 2, 3, 4, 5, 6, and 8 m³/s. The simulations were completely converged; all runs reached the convergence criterion (residual type RMS and residual target to default value 10⁻⁴). The convergence criterion value can be increased, but this means also a considerable increase in the required computational time and not at the cost of significant differences in results. Besides, during the simulations, the most important physical quantities, such as flowrate, pressure, and efficiency, were monitored and it was observed that all these relevant quantities converged properly with the 10⁻⁴ residual target. All simulations were solved by parallel running on 16 CPUs and typical computation time required for full convergence is in wall clock seconds of ~5.0E+03.

5 NUMERICAL RESULTS

5.1 Flow analysis on the blades

Qualitatively, the flow behaviour can be tracked by means of velocity streamlines around the blades, thus picturing if the prescribed profile angles at design stage are indeed achieved. Such exemplary depictions are shown in Fig. 12 at one blade section. Apart from no obvious differences between the three designs, the streamlines also indicate a smooth flow that appears to follow completely the blade contours.

Pressure plots on the blade are helpful in identifying the stressed spots, since according to reference [22], they are responsible for losses in the system caused by sound sources, i.e. the leading edge of the blade and areas close to tip section, due to the tip clearance.

Although the prescribed pressure distribution for the three designs is essentially different (e.g. Design III has twice the pressure at the tip section than at the hub, while Design I has a constant pressure through out the blade length), when plotting the static pressure coefficient, no major differences between the three designs can be observed, as indicated in Fig. 13.

Another aspect to be analysed for the comparison of the flow over the investigated profiles is the velocity distribution, i.e. flow velocity in the span-wise direction, as shown in Fig. 14. At this point, the authors would like to emphasize that the results presented in both Figs 13 and 14 correspond to the full profile cascade at the indicated radius. For the early blade sections (i.e. hub and very close to the hub), the cascade

Fig. 12 Velocity streamlines around the blade for Q_{design} at $r = 227$ mm

Fig. 13 Static pressure coefficient distribution along the blade profiles for Q_{design}

Fig. 14 Span-wise velocity distributions along the blade profiles for Q_{design}

effect can be clearly observed in the velocity distributions, since more variation curves can be identified for the same design. The reason behind these differences is due to the wall influence from the hub and this basically means that, in these regions, the flow over one profile is not symmetric and obvious influences from the neighbours occur. This, however, is not the case for the upcoming sections, where hardly any differences can be observed, meaning that from mid sections to near-tip, there is hardly any cascade influence (see corresponding plots for $r227$ and $r270$). The wall influence in the flow also explains why, for the extreme sections (hub and tip), negative velocities in span-wise direction appear, thus indicating recirculations of the flow on the suction surface of the blades in these regions. While at the hub, the velocity gradient is not too large, and close to the tip sections, strong negative values can be observed, due to the tip clearance phenomenon. Such gradients can be avoided by reducing the gap between the tip and the pipe walls.

As in the case of the computed C_p , there is no major difference value wise in the velocity distribution between the three designs at the investigated section, thus confirming the similar flow pictures depicted in Fig. 12.

5.2 Performance assessment of the new designs: optimum pressure distribution

A correct assessment of a turbomachine depends on defining appropriate performance indicators. Such an indicator is the efficiency of the impeller, which for fans can be defined as

$$\text{Efficiency} = \frac{\text{total (hydrodynamic) energy input to fluid in unit time}}{\text{power input to coupling of shaft}}$$

In other words, the efficiency of the impeller can be written as the ratio between the hydraulic power and the shaft power

$$\text{Efficiency} = \frac{\text{hydraulic power}}{\text{shaft power}} \quad (15)$$

Ideally, the two parameters should be equal so that the efficiency of the machine is 1. However, such a definition becomes useless at design stage since its value will never change no matter the design [23]. Hence, more precise performance indicators have to be defined.

The shaft power developed by the impeller can be written as the product between the angular speed and the required torque

$$P_{\text{shaft}} = \omega T$$

If the static pressure difference between the inlet and the outlet of the impeller is measured, then the

hydraulic power is

$$P_{\text{hydraulic}} = \Delta P_s Q$$

When the total pressure difference across the impeller is considered, then the hydraulic power becomes

$$P_{\text{hydraulic}} = \Delta P_t Q$$

Accordingly, two efficiencies, conveniently named total-to-static and total-to-total efficiencies, can be

Fig. 15 Evaluation of the performance of the three designs: (a) static efficiency; (b) total efficiency; (c) polytopic efficiency

defined

$$\eta_{t-s} = \frac{\Delta P_s Q}{\omega T}$$

$$\eta_{t-t} = \frac{\Delta P_t Q}{\omega T} \quad (16)$$

Also of high relevance for practical use is the polytropic efficiency, defined as $\eta_{poly} = ((\gamma - 1) \ln \pi) / (\gamma \ln \tau)$ where the calculated air ratio is $\gamma = 1.4$, $\pi = p_2/p_1$, and $\tau = T_2/T_1$.

After processing the results of the numerical simulations and evaluating the performance of the three designs according to equation (16), the performance charts in Fig. 15 can be plotted. It can be readily observed that, while at the beginning of the investigated flow range and at the design point, all three designs reach similar performances, for higher flows the advantage of the third design becomes obvious, especially towards the end of the flow range where Design III achieves the following efficiency increases when compared with Design I: 8 per cent static efficiency, 9 per cent in total efficiency, and 10 per cent in polytropic (all absolute differences).

At this point, the advantage of considering at design stage the possibility of a non-free vortex flow (i.e. assuming that in span-wise direction the total pressure is not constant) becomes obvious. Employing such a design assumption impacts directly into the extension of the flow range under which the fan can effectively operate and as indicated in Figs 13 and 14, the flow around the computed profiles, both pressure and velocity wise, is similar from one profile to another. Hence, for the investigated hub ratio, the authors suggest that the optimum pressure distribution along the blade should have a parabolic distribution with the ratio r_{tip}/r_{hub} , as follows:

$$\frac{\Delta P_{t,r}}{\Delta P_{t,h}} = 2 \left(\frac{r_{tip}}{r_{hub}} \right)^2 (r - r_h)^{1.35} + 1 \quad (17)$$

A further increase in the pressure distribution (more than parabolic with r_{tip}/r_{hub}) is not recommended, since a fourth design, corresponding to $x = 2.5$, was investigated and decreases in all performance indicators, compared to $x = 2$, were observed.

5.3 New design versus industrial impeller

As mentioned previously, for optimization purposes, all the new designs that were investigated were derived for the corresponding dimensions and operational parameters of an industrial impeller. Since it was found in the previous section that Design III achieves overall higher performances, this model will be employed from now on for further analysis during the optimization process.

However, before comparing the performances of the reference and suggested design, it becomes interesting

Fig. 16 Total pressure distribution in span-wise direction for the investigated models

Fig. 17 Streamlines around the blade profiles for Q_{design} at $r = 227$ mm: (a) reference impeller; (b) Design III ($x = 2$)

to analyse the differences in design concepts for the two models.

In Fig. 16, the total pressure distributions in span-wise direction prescribed at design stage, for the reference blade and the new models, developed according to the presented design strategy, are depicted. As already stated, what has been used as reference is an axial fan which is currently manufactured for application in the automotive industry for engine cooling purposes. The CAD model of this fan was

made available by the manufacturer for optimization purposes. After reading out the geometry of the blade, the corresponding design parameters were determined and the pressure distribution along the blade was determined. Figure 16 highlights the main goal of the proposed design strategy: to design blade profiles strictly according to the operational requirements of the impeller and not to use predefined ones, which may have been computed for completely different flow conditions, thus ensuring that what has been prescribed at design stage can actually be achieved during operation. It remains to be seen, however, which of the two models (reference and Design III) actually performs better under identical operating conditions, and for this, another set of numerical simulations was carried out.

A first analysis of the numerical results is a qualitative one, i.e. the flow aspect around the blades. By plotting the velocity streamlines for both models at the same radius, it was observed that, while the flow in the industrial impeller is characterized by obvious detachments, smooth streamlines completely following the blade profiles result in the case of new design, as exemplary depicted in Fig. 17.

Furthermore, when comparing the performance curves of the two models as plotted in Fig. 18, significant differences were observed: 7–28 per cent in static efficiency, 2–9 per cent in total efficiency, and 4–17 per cent in polytropic efficiency absolute increases for Design III ($x = 2$).

6 CONCLUSIONS

An integrated optimization procedure based on a design strategy focusing on computing the optimum blade profile at the specified section according to the operating requirements of an axial fan impeller was presented.

While most design methods for axial fans that rely entirely on the extensive data on aerofoil profiles assume constant total pressure at all streamlines based on the free-vortex flow theory, the proposed strategy computes the blade shape after determining the optimum pressure distribution along the blade.

An analytical investigation was carried out and a parametric expression for the total pressure distribution on the blade, based on the dimensions of the impeller, was computed.

Three new designs matching the constructive measures of an industrial impeller used in the automobile industry for cooling purposes were derived, each design being characterized by different pressure variation (including the classical assumption of constant pressure). All three designs were numerically investigated for boundary conditions reproducing the actual working conditions and it was found that for the investigated class of impellers (hub ratio of 0.5),

Fig. 18 Performance comparison for Design III ($x = 2$) and the industrial impeller: (a) static efficiency; (b) total efficiency; (c) polytropic efficiency

the optimum pressure in the span-wise direction is described by a polynomial that increases the pressure from hub to tip and that employing a non-free-vortex flow assumption at the design stage impacts directly in the extension of the flow range under which the fan can effectively operate.

This conclusion was further employed as a starting premise for the optimization process of the industrial impeller. Flow pictures in the two impellers clearly show that while the flow around the reference blade is characterized by obvious flow detachments, the streamlines around the new design follow completely the blade profile and the flow-blade congruency is achieved.

For a further comparison, appropriate performance indicators were defined, i.e. the static, total, and polytropic efficiencies. These plots revealed the improved performance of the new designs with considerable increases compared with the reference impeller. These results stand for a fully optimized impeller and underline the validity of the proposed design strategy. However, the optimization process is completely based on a design method focusing on theoretical aspects of the flow characteristics in an axial impeller. Further improvements to the new design can be achieved by employing genetic algorithms [24] or by improving the geometry of the system [25], and higher performances of the impeller can be obtained, bringing the impeller closer to the ideal machine. It has to be kept in mind, however, that these later methods are more costly and require a good starting solution, which can be given by the present method.

ACKNOWLEDGEMENTS

The authors would like to acknowledge the financial support of the Bavarian Science Foundation.

REFERENCES

- 1 He, L. and Sato, K. Numerical solution of incompressible unsteady flows in turbomachinery. *ASME J. Fluids Eng.*, 2001, **123**, 680–685.
- 2 Goto, A., Nohmi, M., Sakurai, T., and Sogawa, Y. Hydrodynamic design system for pumps based on 3-D CAD, CFD and inverse design method. *ASME J. Fluids Eng.*, 2002, **124**, 329–335.
- 3 Demeulenaere, A. and Van den Braembussche, R. Three-dimensional inverse method for turbomachinery design. *ASME J. Turbomach.*, 1998, **120**, 247–255.
- 4 Jiang, J. and Dang, T. Design method for turbomachines blades with finite thickness by the circulation method. *ASME J. Turbomach.*, 1997, **119**, 539–543.
- 5 Peng, G. A practical combined computation method of mean through-flow for 3D inverse design of hydraulic turbomachinery blades. *ASME J. Fluids Eng.*, 2006, **127**, 1183–1190.
- 6 Bakir, F., Kouidri, S., Belamri, T., and Rey, R. On a general method of unsteady potential calculation applied to the compression stages of a turbomachine – part i: theoretical approach. *ASME J. Fluids Eng.*, 2001, **123**, 780–786.
- 7 Wallis, R. A. Optimization of axial flow fan design. *Mechanical and Chemical Engineering Transactions*, MC4, No. 1, 1976, pp. 31–37 (The Institution of Engineers, Australia).
- 8 Dugao, Z., Jiang, Z., and Song, J. Optimization of an axial flow-fan used for mining local-ventilation. *Comput. Ind. Eng.*, 1996, **31**(3), 691–696.
- 9 Stepanoff, A. J. *Turboblowers. Theory, design and application of centrifugal and axial compressors and fans*, 1955 (John Wiley & Sons, Inc., New York).
- 10 Sørensen, D. N., Thompson, M. C., and Sørensen, J. N. Toward improved rotor-only axial fans – part I: a numerically efficient aerodynamic model for arbitrary vortex flow. *ASME J. Fluids Eng.*, 2000, **122**, 318–323.
- 11 Sørensen, D. N., Thompson, M. C., and Sørensen, J. N. Toward improved rotor-only axial fans – part II: design optimization for maximum efficiency. *ASME J. Fluids Eng.*, 2000, **122**, 324–329.
- 12 Albuquerque, R. B. F., Manzaneres-Filho, N., and Oliveira, W. Conceptual optimization of axial-flow hydraulic turbines with non-free vortex design. *Proc. IMechE, Part A: J. Power and Energy*, 2007, **221** (A5), 713–725. DOI: 10.1243/09576509JPE394.
- 13 Lewis, R. I. *Turbomachinery performance analysis*, 1996 (Arnold, London).
- 14 Carolus, Th. *Ventilatoren*, 2003 (Teubner, Wiesbaden).
- 15 Bohl, W. and Elmendorf, W. *Stroemungsmaschinen 1*, 2004 (Vogel Buchverlag).
- 16 Pascu, M. and Eppele, P. Numerical investigation of the validity of axial blade design. In *ASME Proceedings of the International Mechanical Engineering Congress and Exposition*, 2007, paper IMECE 2007–42960.
- 17 Eck, B. *Fans*, 1st edition, 1973, pp. 234–251 (Pergamon Press).
- 18 Eckert, B. *Axial-und Radialkompressoren*, 1953 (Springer, Berlin).
- 19 Wallis, R. A. *Axial flow fans*, 1961 (Newnes London).
- 20 Menter, F. R. Zonal two-equation $k-\omega$ turbulence models for aerodynamic flows, 1993, AIAA paper 96-2906.
- 21 Anderson, J. D. *Fundamentals of aerodynamics*, 1984 (McGraw-Hill Book Company, New York).
- 22 Reese, H., Kato, C., and Carolus, T. Large eddy simulation of acoustical sources in a low axial-flow fan encountering highly turbulent inflow. *ASME J. Fluids Eng.*, 2007, **129**, 263–272.
- 23 Eppele, P., Ilic, C., and Durst, F. Combined analytical and numerical radial fan performance optimization. In *Conference on Modeling Fluid Flow*, Budapest, 2006, pp. 1110–1117.
- 24 Van Den Braembussche, R. A. Modern turbomachinery component design. *Conference on Modeling Fluid Flow (CMFF '06)*, Budapest, 2006, pp. 33–47.
- 25 Neal, D. R., and Foss, J. F. The application of an aerodynamic shroud for axial ventilation fans. *ASME J. Fluids Eng.*, 2007, **129**, 764–772.

APPENDIX

Notation

A	area (m ²)
c	absolute velocity (m/s)
l	blade chord (m)
n	rotational speed (r/min)
P	power (W)
Q	flowrate (m ³ /s)
r	radius (m)
T	torque (Nm)
w	relative flow velocity (m/s)
w_m	meridional component of the relative velocity (m/s)
w_u	tangential (peripheral) component of the relative velocity (m/s)

γ	blade angle (°)
ΔP	pressure difference (Pa)
η	efficiency
π	pressure ratio between the fan outlet and the inlet static pressures
ρ	density (kg/m ³)
τ	temperature ratio between the fan outlet and inlet temperatures
ω	angular velocity (rad/s)

Subscripts

1	blade inlet
2	blade outlet
d	dynamic
h	hub
s	static
t	tip, total