PERIODICA POLYTECHNICA SER. MECH. ENG. VOL. 46, NO. 2, PP. 107–116 (2002)

COMPARATIVE INVESTIGATION ON AXIAL FLOW PUMP ROTORS OF FREE VORTEX AND NON-FREE VORTEX DESIGN

János VAD*, Ferenc BENCZE*, Helmut BENIGNI**, Wolfgang GLAS** and Helmut JABERG**

*Department of Fluid Mechanics
Budapest University of Technology and Economics
H–1521 Budapest, Hungary
**Institute for Hydraulic Fluid Machinery
Technical University of Graz, Austria

Received: January 8, 2001

Abstract

Comparative computational investigation has been carried out on two axial flow pump rotors having identical basic geometrical and flow rate parameters but differing in the design methods. The two rotors were designed for free vortex (FV) and non-free vortex (NFV) operation. The global as well as pitchwise-averaged and pitchwise resolved blade passage flow characteristics were investigated in detail. It has been concluded that NFV design is an effectual method for increase of specific performance but the increased risk of cavitation requires careful blade optimisation. It was found that the design blade circulation must kept constant near the hub in order to diminish the risk of corner stall. Furthermore, it has been anticipated that for the NFV rotor the efficiency drops more intensely with increase of tip clearance and the noise level may be lower than for the FV rotor.

Keywords: axial flow turbomachinery, free vortex design, non-free vortex design, rotor flow, computational fluid dynamics.

1. Introduction

The classic free vortex (FV) design method of axial flow turbomachinery rotors prescribes an ideal total head rise constant along the radius (KELLER, 1937). The FV operation was considered in early designs as an optimum case since principally no spanwise gradient of blade circulation is present in the rotor and there is no shed vortex from the blade. Due to the absence of the shed vorticity, cylindrical stream tubes can be assumed in FV design and the optimisation of blade sections can be carried out reliably on the basis of two-dimensional (2D) cascade concepts. The FV blading is highly twisted because blade inlet, exit and stagger angles vary considerably with radius. The maximum turning and risk of stall near the hub causes a limitation in the design mean total head rise coefficient for a FV rotor.

As summarised by LAKSHMINARAYANA (1996), ‘the myth that the free-vortex blading has the lowest losses has been replaced by a more systematic optimisation’ in modern axial flow turbomachinery design. The choice of the design tangential velocity distribution appears as a basic aspect of systematic optimisation. In order to increase the rotor performance, the demand of spanwise constant (FV) blade circulation must be released and a spanwise increasing blade circulation (and ideal total head rise) is to be prescribed. This method is called in present paper as non-free vortex (NFV) design. The NFV method ensures an increased contribution of blade sections at higher radii to overall rotor performance, even though the spanwise circulation gradient by NFV design results in shed vorticity from the blade. Modern high specific performance fans, blowers and compressors (especially automotive ones, e.g. for gas turbines in airplane jet propulsion engines) are designed in several cases with the NFV method (CUMPSTY, 1989; LAKSHMINARAYANA, 1996; WALLIS, 1983). Although for axial flow propeller pumps the FV method is mostly used, a NFV methodology appears also as an option (SAAFELD, 1954; KRISHNA, 1997). The NFV design methodology gives a potential for the increase of the specific pump performance, which is useful e.g. in mobile pumps of simultaneously high flow rate and high total head rise.

The shed vorticity from the blade due to NFV design results in a characteristic three-dimensional (3D) secondary flow inside the blade passages. This 3D flow comprises radial inward flow near the blade pressure side (PS) and radial outward flow near the blade suction side (SS), as outlined by SIEVERDING (1986) and pointed out experimentally e.g. by VAD and BENCZE (1998).

The development of 3D interblade flow in a NFV rotor introduces difficulties in blade optimisation. The present paper aims at making a fluid mechanical comparison between two pump rotors of FV and NFV operation. By this comparison, special effects due to 3D interblade flow are explored and applicability of NFV method in pump design is surveyed.

2. Test Rotors

In order to make possible a reasonable fluid mechanical comparison between rotors of FV and NFV operation, it was essential to design two isolated rotors

- identical in basic geometrical characteristics,
- having equal design mean flow coefficient but
- differing in the design mean total head rise coefficient.

The FV rotor was designed at the Institute for Hydraulic Fluid Machinery, Technical University of Graz, and the NFV one was designed at the Department of Fluid Mechanics, Budapest University of Technology and Economics.

The geometrical and flow parameters of the two comparative rotors are summarised in *Table 1*. The rotors have an identical blade number, hub-to-tip ratio, and relative tip clearance. Both rotors were designed for a mean flow coefficient $\Phi = 0.39$, with assumption of uniform axial inlet condition. No inlet swirl was considered. The blades have no sweep and no dihedral.

The FV rotor was designed for a spanwise constant blade circulation, performing a mean total head rise coefficient $\Psi = 0.17$ at the design flow rate (for the definition of Ψ, the reader is requested to study the list of symbols). The FV design

Table 1. Rotor geometrical and flow parameters

	FV rotor			NFV rotor		
blade number	5			5		
hub-to-tip ratio	0.51			0.51		
tip clearance (percent span)	0.3%			0.3%		
blade profile	NACA 65			NACA 65		
Φ	0.39			0.39		
Ψ	0.17			0.23		
	hub	midspan	tip	hub	midspan	tip
blade solidity ℓ/t	1.24	0.92	0.72	1.23	0.95	0.72
profile thickness (percent chord)	12%	7.5%	6.0%	12.0%	7.5%	6.0%
stagger angle, deg	55.3	33.6	23.0	50.0	37.4	29.4
camber angle, deg	32.2	16.0	13.6	27.6	25.7	27.0

methodology applied is summarised and further data on the FV rotor are reported in GLAS and KUHN (1998). For the NFV rotor, the prescribed blade circulation increases along the radius as a power function, thus resulting in a significantly higher mean total head rise coefficient $\Psi = 0.23$. The NFV design method is described in VAD and BENCZE (1998).

The blade solidity is practically equal for the two rotors. The blade profiles have been selected from the NACA 65 series and are identical for the two rotors at each radius. The camber line is non-circular for the FV rotor and is circular for the NFV rotor, in accordance with the different design methods applied. The blade stagger angle is calculated as the mean value of inlet and outlet blade angles, and is measured from the circumferential direction. The camber angle is calculated as the difference of inlet and outlet blade angles. It can be observed that the NFV blades are less twisted, and have higher stagger and blade curvature (except for the hub region) according to the higher performance required.

3. Computational Technique and Test Case

At the present state of research, the rotors have been tested by means of Computational Fluid Dynamics (CFD). Experimental verification of the CFD investigation is planned in the near future. The CFD studies have been carried out at the Institute for Hydraulic Fluid Machinery, Technical University of Graz, with use of CFD code TASCFlow, version 2.9.0.

The computational grid for the CFD studies consists of about 150.000 nodes. The nodes next to the walls are located at a dimensionless wall distance of $y^+ = 30$, which allows the application of a two-layer version of the standard k-epsilon model in order to accurately resolve the boundary layer. (y^+ is a dimensionless length coordinate used traditionally in boundary layer characterization. It is derived by multiplying the length coordinate by the ‘frictional velocity’ and dividing by the kinematic viscosity. The ‘frictional velocity’ is defined as square root of wall shear stress divided by fluid density.) A second order upwinding scheme assures the computational accuracy in the advection-dominated regions.

The inlet boundary conditions have been retrieved by a CFD simulation of the hub at the water test stand of the Institute for Hydraulic Fluid Machinery, Technical University of Graz, where the CFD results of the FV rotor have been verified by experiments.

The FV and NFV rotors have been tested for the design flow coefficient. In the CFD tests, the Reynolds number based on kinematic viscosity of water, blade chord and blade velocity at mid-span was $1.5 \cdot 10^6$.

4. Discussion of Results

The results of the CFD investigation have been represented and evaluated in a complex manner. In the following sections, most impressive fluid mechanical aspects are discussed.

4.1. Performance and Cavitation Characteristics

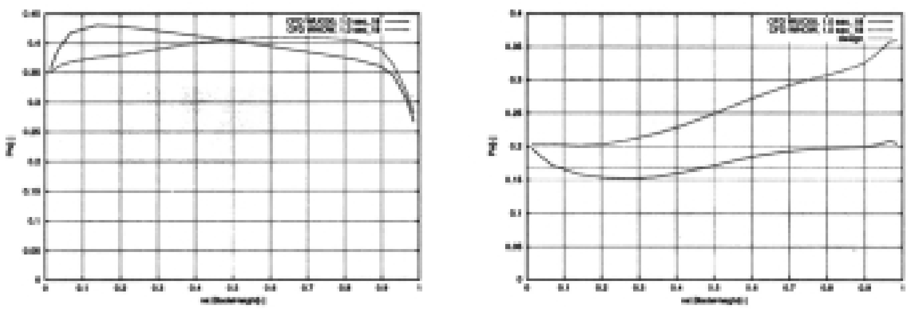

Fig. 1. Pitchwise-averaged flow (‘phi’) and swirl (‘psi’) coefficients donstream of the rotors on a plane perpendicular to rotor axis. Axial distance of the plane is 95 percent of the tip diameter from the blade stacking line. Solid line: FV rotor. Dashed line: NFV rotor

Fig. 1 presents pitchwise-averaged flow and swirl coefficients downstream of the two rotors. Behaving as expected by design (VAD and BENCZE, 1998), the NFV rotor performs swirl and axial velocity increasing with radius, ensuring higher rotor performance. A slight swirl gradient is present also in the case of the FV rotor.

This is due to the fact that a uniform axial inlet condition was assumed in design, however, the streamline deflection effect of the inlet nose cone resulted in spanwise decreasing axial inlet velocity.

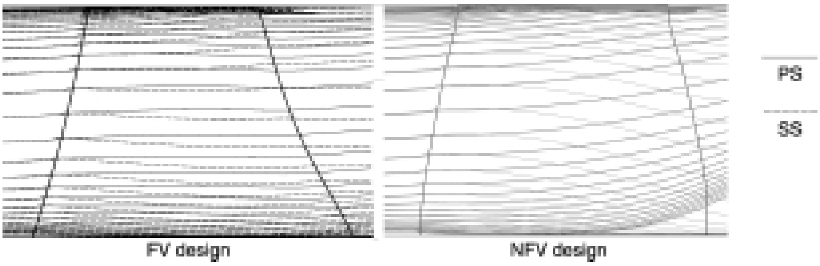

Fig. 2. Streamlines near the PS and SS at the edge of boundary layers

Fig. 2 shows streamlines passing the rotors near the PS and SS at the edge of blade boundary layers. For the NFV rotor, the radial outward flow on SS and radial inward flow on PS can be observed with a single glance. For consideration of such 3D effects, VAD et al. (2000) proposed the inclusion of systematic forward sweep of blades in NFV design. Three-dimensionality of blade passage flow is not characteristic for the FV rotor.

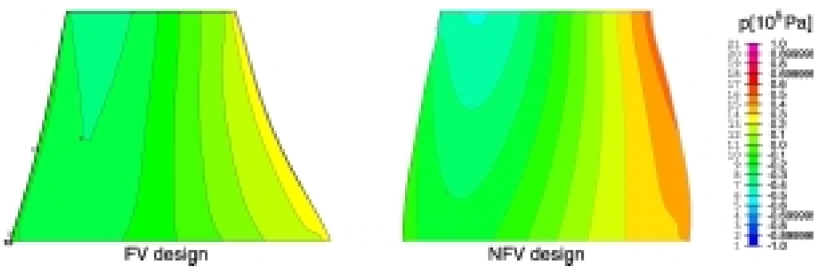

Fig. 3. Static pressure distribution on the SS blade surface

Figs. 3 and 4 show the static pressure distribution on the SS and PS blade surfaces of the two rotors, consecutively. The PS pressure distributions near the trailing edges (*Fig.*4) suggest that the trailing edge geometries require further optimisation. However, this necessity of improvement does not regard the fulfilment of goals of present paper. The pressure is represented as an overpressure (positive values) or as a depression (negative values) compared to a reference pressure. For reference pressure, the midspan mean static pressure one chord length upstream of the leading edges has been chosen. Both in the FV and NFV rotor, the SS is characterised predominantly by streamwise pressure gradients. The dominance of streamwise pressure gradients is observed also for the PS of the FV rotor. Contrarily, radial pressure gradients are dominant on the PS of the NFV rotor. The higher lift of NFV blade sections manifests itself in considerably increased pressure on the PS. The depression on the SS of the NFV rotor is also increased relative to the FV rotor, especially above midspan near the leading edge. In this region, the NFV blading under investigation is assumed to be apt to cavitation. With elaboration of the appropriate blade profile geometry, improvement of pump performance by NFV design can be possibly achieved with increase of PS pressures only, i.e. without increase of SS depression leading to cavitation.

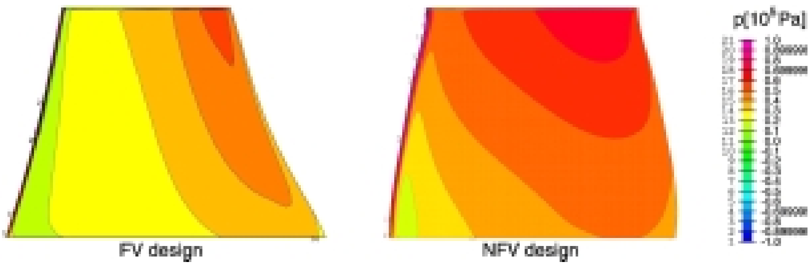

Fig. 4. Static pressure distribution on the PS blade surface

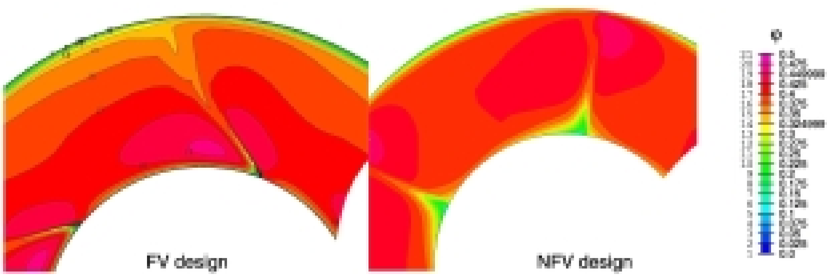

Fig. 5. φ local flow coefficient distribution downstream of the rotors on a plane perpendicular to rotor axis. Axial distance of the plane is 23 percent of tip diameter from the blade stacking line.

4.2. Flow Effects Near the Hub

MEIXNER (1994) investigated fan rotors for which the main intention for application of NFV design was not the increase of rotor performance (increased blade load at higher radii) but the reduction of the blade load near the blade root in order to reduce hub losses. In LAKSHMINARAYANA (1996), it is stated that even though the losses due to shed vorticity are minimised by FV design, the hub losses are likely increased given that the tangential velocity and flow turning has a maximum at the hub. Inspired by the above, one could think that with prescription of a reduced load near the hub of a NFV rotor, the hub losses and risk of hub corner stall can be reduced. In the following section, it is demonstrated that such consideration neglects the effects related to the 3D interblade flow and hub boundary layer.

It is conspicuous in *Fig. 2* that the streamlines entering the NFV rotor along the entire span near the SS are inclined radially outward due to NFV operation. As a consequence, a zone without any passing streamlines appears in the SS blade root – hub corner near the trailing edge. This zone is assumed to be a hub corner stall region. No similar phenomenon is observed for the FV rotor (*Fig. 2*). Behaving as expected, the hub stall zone in the NFV rotor manifests itself in locally decreased axial velocity (*Fig. 5*), increased swirl (*Fig. 6*) just downstream of the blade trailing edge. Such effects cannot be recognised in the FV rotor (*Fig. 5*, *Fig. 6*).

The CFD studies indicate hub corner stall in the NFV rotor and no stall in the FV rotor. This seems to be in contradiction with the fact that blade stagger and camber angle is lower at the hub of the NFV rotor (*Table 1*). In addition, the turning performed by the NFV rotor is generally higher near the hub, despite lower blade stagger and camber angle values (*Fig. 1*). This observation calls the attention that a simple 2D cascade approach, attempting the reduction of blade circulation by means of reducing blade stagger and camber angles near the hub, is insufficient in itself for blade root load moderation and reduction of hub losses in a NFV rotor.

To avoid hub corner stall, i.e. to hamper the development of a near-hub zone without any passing streamlines (*Fig. 2*), the streamlines entering the rotor on the SS near the hub should not drift radially outward. This means that the shed vorticity due to NFV operation should be eliminated near the hub. Accordingly, the design blade circulation should be kept constant with radius near the hub even for a NFV rotor. Such design consideration results in increased stagger and camber angle at the blade root, if the inlet condition is fixed.

This is in accordance with experiences of WISLER (1985), cited also in LAKSHMINARAYANA (1996), although their consideration was based on accommodation of blade geometry at the blade root to the hub boundary layer. The flow at the exit of the rotor in the hub region has inevitably a higher turning, and the above authors suggest that it is desirable to modify the blade geometry to accommodate this higher turning in order to reduce endwall losses. If the inlet blade angle is fixed, the accommodation of higher turning by blade geometry corresponds to increased stagger and camber angles at the blade root.

4.3. Flow Effects Near the Blade Tip

Fig. 7 represents streamlines approaching the blades at the mid-radius of tip clearance. It is conspicuous that the tip leakage flow being present in the FV rotor appears as a concentrated jet (with chance of rolling-up into a tip leakage vortex, as e.g. in INOUE et al., 1996). Contrarily, the leakage flow is rather diffused in the case of the NFV rotor with no rolling up. The casing wall boundary layer is more thickened in the NFV rotor and considerable increase can be observed in the absolute tangential velocity near the casing (*Fig. 1*). These observations are in accordance with results by FURUKAWA et al. (1998), although these authors established such behaviour not for a NFV rotor but for a FV diagonal rotor of very high blade loading. As they concluded, such flow characteristics in the blade tip region cause large reduction of efficiency with the size of tip clearance, and reduced level of noise (due to disappearance of tip clearance vortex core). The present survey anticipates that such behaviour is not only the characteristic of a FV rotor of high blade loading but also of a NFV rotor for which the blade load in the tip region is especially high.

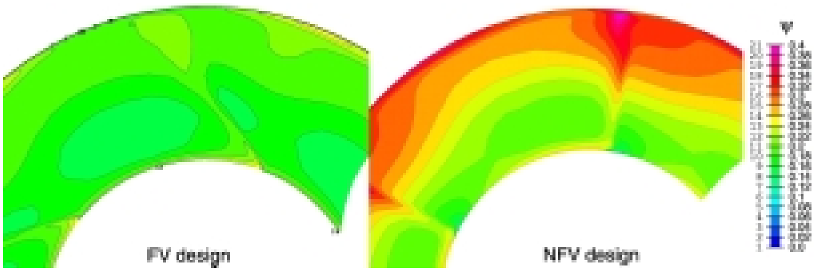

Fig. 6. ψ local flow coefficient distribution downstream of the rotors on a plane perpendicular to rotor axis. Axial distance of the plane is 23 percent of tip diameter from the blade stacking line.

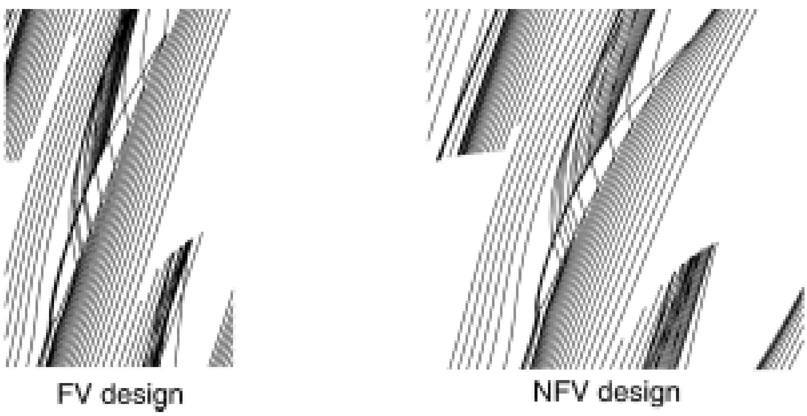

Fig. 7. Streamlines approaching the blades at the mid-radius of tip clearance

5. Summary

Comparative CFD investigation has been carried out on two axial flow isolated pump rotors of FV and NFV design in order to study the special flow effects due to NFV operation and to survey the applicability of NFV concept in pump design. The results are summarised as follows:

1. The investigation showed that NFV design is an effectual method for considerable increase of specific pump rotor performance. However, it was found that the risk of cavitation is increased in the NFV rotor above midspan on the suction side near the leading edge. Such undesirable effect can be possibly diminished by means of selection of appropriate blade section geometry.
2. It was found that the blade circulation must be kept constant with radius near the hub even in the case of a NFV rotor in order to diminish the risk of corner stall. This perception is in accordance with 3D flow effects in the NFV blade passage. It is in contrast with the 2D approach that the hub losses can be reduced by means of prescription of reduced blade circulation, i.e. reduced stagger and camber angles, near the blade root.
3. The flow in the NFV axial rotor near the blade tip showed characteristics similar to those concluded for a FV diagonal rotor of high blade loading (FURUKAWA et al., 1998). As a consequence, it is assumed that for a NFV rotor the efficiency drops more intensely with increase of tip clearance than for a FV rotor, and the noise level of the NFV rotor may be lower than that of the FV rotor.

Acknowledgement

Hungarian authors (J. Vad, F. Bencze) acknowledge the support of OTKA (Hungarian National Foundation for Science and Research) under contract T 025361 and MKM (Hungarian Ministry of Education) under contract FKFP 0356/1999. Author J. Vad has also been supported by a János Bolyai Hungarian National Research Grant, Ref. No. BO/00150/98. Austrian authors (W. Glas, H. Jaberg) acknowledge the support of the Austrian Science Fund (FWF), project P12072-ÖTE ‘swept blades’. The concerted research activity has been carried out within the framework of Hungarian-Austrian Intergovernmental Scientific and Technological (TéT) Cooperation Projects A-34/98 and A-10/01.

Nomenclature

c	absolute velocity
ℓ	blade chord
r	radius
$R = r/r_t$	dimensionless radius

$\overline{\Delta p_{t\,\mathrm{id}}}$	mass-averaged ideal total head rise in the annulus
t	blade pitch
u_t	tip velocity
Φ	mean flow coefficient: area-averaged axial velocity in the annulus divided by u_t
$\Psi = \overline{\Delta p_{t\,\mathrm{id}}}/\rho u_t^2$	mean total head rise coefficient
$\varphi = c_x/u_t$	local flow coefficient
$\psi = Rc_u/u_t$	local swirl coefficient
ρ	fluid density

Subscripts and Superscripts

t	blade tip
u, x	tangential, axial

References

[1] CUMPSTY, N. A. *Compressor Aerodynamics*, Longman Group, New York, 1989.

[2] FURUKAWA, M. – SAIKI, K. – NAGAYOSHI, K. – KUROUMARU, M. – INOUE, M., Effects of Stream Surface Inclination on Tip Leakage Flow Fields in Compressor Rotors, *ASME Journal of Turbomachinery*, **120** (1988).

[3] GLAS, W. – KUHN, K., Vergleich der Festigkeit von gepfeilten und ungepfeilten Schaufeln in axialen hydraulischn Turbomaschinen, Beitragstexte (V17 bis V39) zum *XXX. Kraftwerkstechnischen Kolloquium*, pp. 110–119, October 1998, Dresden, Germany.

[4] INOUE, M. – KUROUMARU, M. – FUKUHARA, M., Behavior of Tip Leakage Flow Behind an Axial Compressor Rotor, *ASME Journal of Engineering for Gas Turbines and Power*, **108** (1986).

[5] KELLER, C., *The Theory and Performance of Axial Flow Fans*, McGraw-Hill, New York-London, 1937.

[6] KRISHNA, H. C. R., ed., *Hydraulic Design of Hydraulic Machinery*, Avebury, 1997.

[7] LAKSHMINARAYANA, B., *Fluid Dynamics and Heat Transfer of Turbomachinery*, John Wiley and Sons, Inc., 1996.

[8] MEIXNER, H. U., Vergleichende LDA-Messungen an ungesichelten und gesichelten Axialventilatoren, PhD Thesis, University of Karlsruhe, VDI Verlag, 1994.

[9] SAAFELD, K., *Berechnung und Konstruktion von Propellerschaufeln*, KSB Interner Bericht, 1954.

[10] SIEVERDING, C. H., Secondary Flows in Axial Turbines, Lecture series, Von Karman Institute for Fluid Dynamics, Belgium, 1986.

[11] VAD, J. – BENCZE, F., Three-Dimensional Flow in Axial Flow Fans of Non-Free Vortex Design, *International Journal of Heat and Fluid Flow*, **19** December, (1998).

[12] VAD, J. – BENCZE, F. – CORSINI, A. – RISPOLI, F., Design Aspects of Three-Dimensional Flow in High Performance Axial Flow Ventilating Fans, *Ventilation 2000, 6th International Symposium on Ventilation for Contaminant Control*, June 2000, Helsinki, Finland.

[13] WALLIS, R. A., *Axial Flow Fans and Ducts*, John Wiley and Sons, New York, 1983.

[14] WISLER, D. C., Loss Reduction in Axial Flow Compressors Through Low Speed Testing, *ASME Journal of Engineering for Gas Turbines and Power*, **107** (1985).