Available online at www.sciencedirect.com
SCIENCE DIRECT®
Applied Thermal Engineering 26 (2006) 846–852
APPLIED THERMAL ENGINEERING
www.elsevier.com/locate/apthermeng

Numerical investigation of fan performance in a forced draft air-cooled steam condenser

J.R. Bredell [a,*], D.G. Kröger [a,1], G.D. Thiart [b,2]

[a] Department of Mechanical Engineering, University of Stellenbosch, Private Bag X1, Matieland 7602, South Africa
[b] School for Science and Technology, Faculty of Military Science, Private Bag X2, Saldanha 7395, South Africa

Received 9 May 2005; accepted 19 September 2005
Available online 2 December 2005

Abstract

Inlet air flow distortions in a large air-cooled steam condenser (ACSC), caused by structures, wind and other fans may result in a significant reduction in fan performance, or volumetric effectiveness, as well as fan blade vibration. This phenomenon has an adverse effect on the heat rejection capacity of an ACSC due to a decrease in air mass flow rate. In this study the effect of inlet flow distortions on the flow rate through the fans in an ACSC is numerically investigated by modelling the flow field in a section of such a system using the computational fluid dynamics code, FLUENT. The effect of platform height on the volumetric effectiveness of two different types of axial flow fans is considered. The two fans have the same diameter, number of blades and rotational speed, but feature different blade designs, and hub-tip-ratios of respectively 0.153 and 0.4. Numerical simulations show all-round superior performance in terms of volume flow rate for the fan with a hub-tip-ratio of 0.4. It is furthermore confirmed that the addition of a walkway can significantly increase the flow rate through the fans located near the edge of the fan platform.

Keywords: Air-cooled; Inlet flow distortion; Axial flow fan; Fan performance; CFD

1. Introduction

Forced draft air-cooled steam condensers (ACSCs) consisting of an array of fan units are used in direct air-cooled power plants to condense steam in a closed steam cycle. Axial flow fans located below an A-frame configuration of finned tube heat exchanger bundles, force a stream of ambient cooling air through the system. In so doing, heat from the condensing steam is rejected to the environment via the finned tubes. Owing to the dynamic interaction between the steam turbines and the ACSC, a change in the heat rejection rate of the ACSC will directly influence the efficiency of the steam turbines. Understanding and predicting the factors or mechanisms that can reduce the heat rejection rate of an ACSC is therefore essential. This study involves the numerical modelling of the flow about and through a middle section of a long ACSC bank consisting of six fan rows, as shown in Fig. 1, in order to determine the effect of inlet flow distortions on fan flow rate. Although an ACSC is currently analysed, this investigation is equally relevant to forced draft air-cooled heat exchangers (ACHEs) due to close similarity between the two different systems.

In an experimental study on a scale model of a forced draft ACHE Salta and Kröger [1] observed significant changes in air flow rate by varying the platform height. The test model used was in effect representative of a middle section of a long ACHE bank consisting of up

* Corresponding author. Tel.: +27 21 808 4376; fax: +27 21 808 4958.
E-mail addresses: jrbredell@sun.ac.za (J.R. Bredell), dgk@sun.ac.za (D.G. Kröger), thiart@ma2.sun.ac.za (G.D. Thiart).
1 Tel.: +27 21 808 4259; fax: +27 21 808 4958.
2 Tel.: +27 22 702 3153; fax: +27 22 702 3060.

1359-4311/$ - see front matter
doi:10.1016/j.applthermaleng.2005.09.020

Nomenclature

d	diameter (m)
H	height (m)
N	rotational speed (rpm)
n	number
P	power (kW)
p	pressure (Pa)
V	volume flow rate (m^3/s)
W	width or pitch (m)
X	dimensionless platform height (–)

Greek symbols

Δ	differential
δ	increment
η	efficiency (%)
ρ	density (kg/m^3)

Subscripts

F	fan
FB	fan blade
FP	fan platform
FR	fan rows
Fs	fan static
h	hub
i	inlet
id	ideal
sys	system
w	walkway

to 12 fan rows, similar to Fig. 1. It was found that lowering the platform height resulted in a reduction in flow rate through the fans, most notably the edge fans, due to flow distortions and separation at the bell-mouth fan inlets. The effect of a solid walkway extending horizontally from the fan platform, on the flow rate, was also considered.

Duvenhage et al. [2] numerically investigated flow distortions at the cylindrical fan inlets of a long forced draft ACHE consisting of two fan rows. The reduction in volume flow rate through the system associated with a decrease in platform height was successfully modelled. Numerical results showed a trend similar to the experimental data of Salta and Kröger [1], despite different geometry and fan inlets used. Comparison of experimental results with those obtained by Spiers [3] and Russel and Peachy [4], who also used cylindrical fan inlets, lead Duvenhage et al. [2] to conclude that the type of fan used, affects the flow rate through an ACHE operating under distorted inlet conditions.

In a more recent study Meyer [5] numerically investigated the effect of inlet flow distortions on the flow rate through an ACHE consisting of two and four fan rows at different platform heights. Although the trend in the

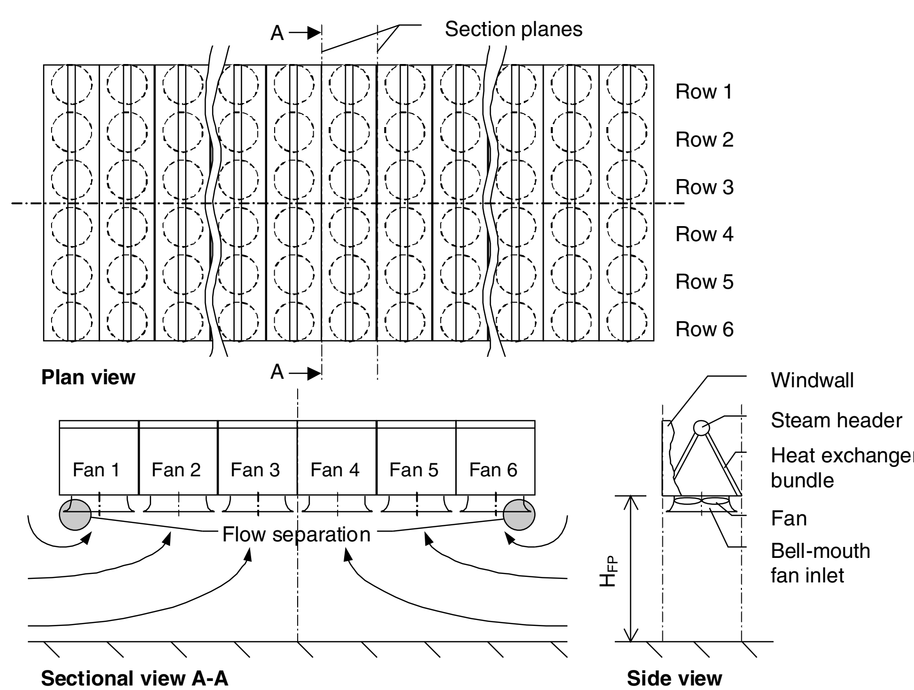

Fig. 1. Schematic of a representative section of a long ACSC depicting the induced cross-draft and resulting flow separation at the inlets of the periphery fans.

cumulative flow rates through all the fans were the same as measured by Salta and Kröger [1], close scrutiny of individual fan flow rates revealed that the fans used by Salta and Kröger [1] were more sensitive to flow distortions than the fans used by Meyer [5]. Different fan inlet sections were considered and it was found that by removing the fan inlet section of the edge fan reduced inlet flow losses and consequently increased the flow rate through this particular fan. The flow rate enhancing effect of a walkway was also numerically confirmed.

Based on previous experimental and numerical research it can be concluded that flow distortions result in a reduction in air flow rate due to increased inlet losses and a reduction in fan performance. Bruneau [6], who designed an axial flow fan for application in an ACSC, explains this reduction in fan performance in more detail, and states that, "Depending on the severity of the distortions (i.e., non-uniform inlet profiles) the fan blade can stall, accompanied by losses in aerodynamic efficiency and peak pressure rise, as well as blade vibrations". It is furthermore deduced that the air flow rate through an ACHE (or ACSC) subjected to inlet flow distortions, is dependent on the type or fan used. The objective of this study is to critically compare the flow rate delivered by two different fans in a section of an ACSC under various degrees of inlet flow distortion.

2. Computational aspects

The flow field in the section of an ACSC is modelled using the CFD code, FLUENT, which numerically solves the finite volume based, steady-state discretized differential equations for the conservation of mass and momentum, relevant to incompressible viscous fluids. The SIMPLE algorithm [7] for pressure correction and the k–ε model for turbulence [8] are employed. A typical fan unit in the ACSC under investigation is simplified as shown in Fig. 2. It can be seen that objects such as inlet screens, support beams, electrical fan drives and ducting are not modelled. The flow losses resulting from these objects are however taken into account in the effective system resistance, which also includes flow losses imparted by the heat exchanger bundles. The parabolic effective system resistance curve corresponding to a fan unit in the ACSC (shown in Fig. 6) was modelled in FLUENT using a porous media condition. By specifying infinitely high porosity in the plane of the porous media, the discharge flow was aligned normal to this plane. The plenum chambers of adjacent fan units are separated, so that the air flow delivered by each fan is not mixed. The A-frame plenum chamber was simplified as a rectangular box. This simplification is justified considering that the actual plenum chamber is essentially a free outlet, and that the focus of this study is the investigation of the flow field near the fan inlets, and not the flow inside the plenum chamber. Regarding the latter, reference is made to Meyer and Kröger [9] who investigated plenum chamber aerodynamic behaviour in an ACHE.

Dimensions pertaining to the fan installation of the ACSC are shown in Fig. 3. Two different rotor-only axial flow fans are investigated. The first fan, hereafter referred to as the A-fan, has the following main features: diameter $d_F = 9.145$ m, number of fan blades $n_{FB} = 8$, hub-tip-ratio $d_h/d_F = 0.153$, rotational speed $N = 125$ rpm. Such fans are commonly found in industrial cooling applications. The second fan, hereafter referred to as the B-fan, has the same diameter, number of blades and rotational speed as the A-fan, but features an enlarged hub with a hub-tip-ratio of $d_h/d_F = 0.4$. Bruneau [6] designed the B-fan with the purpose of application in an existing ACSC. The original fan installation was of the low hub-tip-ratio type, similar to the A-fan. A characteristic of these low hub-tip-ratio type fans is the recirculation or backflow that occurs near the hub. In the design of the B-fan, the fan efficiency at the required volume flow rate and pressure rise, was improved by eliminating the reverse flow effect at the hub region by increasing the hub size. To prevent confusion me B-fan in the current investigation refers to the B2-fan in Bruneau [6].

An actuator disc model, as discussed by Thiart and von Backström [10], was employed to numerically simulate and predict the performance of the axial flow fans. The aforementioned model entails the calculation of momentum source terms based on blade element theory,

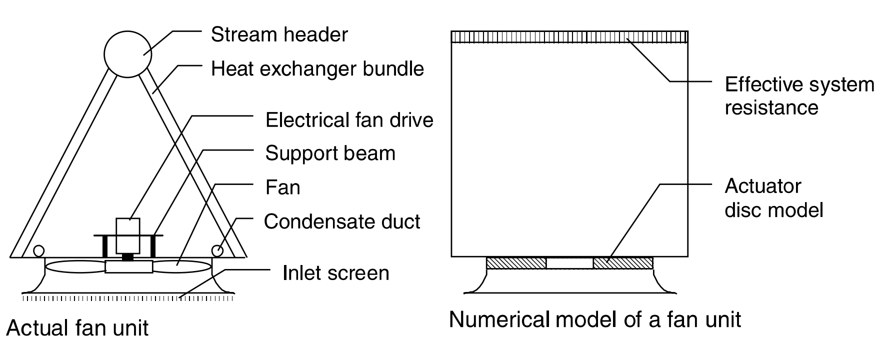

Fig. 2. Numerical model of a typical fan unit in the ACSC.

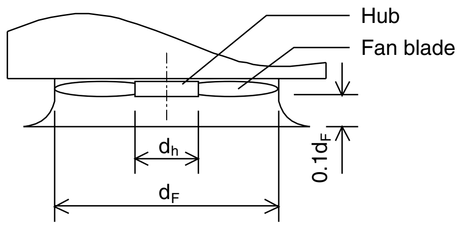

Fig. 3. Fan installation.

thereby modelling the effect of an axial flow fan without actually modelling the fan blades as rotating solid surfaces in the flow domain. In order to confirm the accuracy of the actuator disc model, the numerically predicted free inlet, free outlet fan characteristics for a scale model of the B-fan ($d_F = 1.542$ m, $N = 750$ rpm) are compared to experimental data (as per British Standard BS 848 Part 1 [11]) obtained by Stinnes [12]. Fig. 4 shows good consistency between numerical predictions and experimental data, for both the fan static pressure and fan shaft power. Further detail regarding the validation of the actuator disc model, in addition to the fan blade geometry and fan blade characteristics of respectively the A-fan and B-fan, are reported by Bredell [13].

Assuming that under windless conditions the flow in the section of the ACSC does not cross the sectional planes, these planes were accordingly modelled as slip-walls. Owing to the symmetry plane through the longitudinal axis of the fan platform, it was only required to model one half of the section under consideration, i.e., fans 1–3. Detail regarding the geometric layout of the computational grid and specified boundary conditions are shown in Fig. 5. The atmospheric inlet boundary, which is placed far away so as not to affect the flow near the fan inlets, is modelled by means of a total pressure

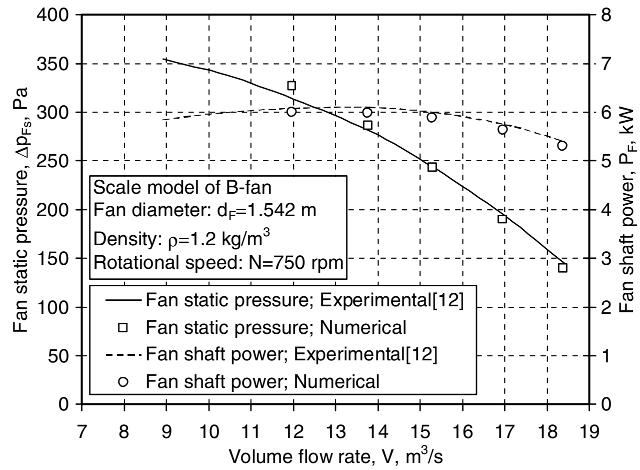

Fig. 4. Numerical prediction of the fan characteristics for a scale model B-fan.

boundary condition. A static pressure boundary located downstream of the fan units serves as a flow outlet. The ground, bell-mouth fan inlets and plenum chamber walls are modelled as zero-slip walls. Depending on the platform height, the computational grid consisted of between 450,000 and 500,000 cells or control volumes. For the ACSC under consideration the required volume flow rate through a single fan unit was found to be approximately 650 m^3/s for an ambient air density of 1.085 kg/m^3. In order to compare the A-fan and the B-fan, both fans are required to have the same operating point under ideal inlet conditions. To satisfy the aforementioned condition the fan blade settings of both fans are accordingly adjusted, as depicted through the numerically determined free inlet, free outlet fan characteristics shown in Fig. 6.

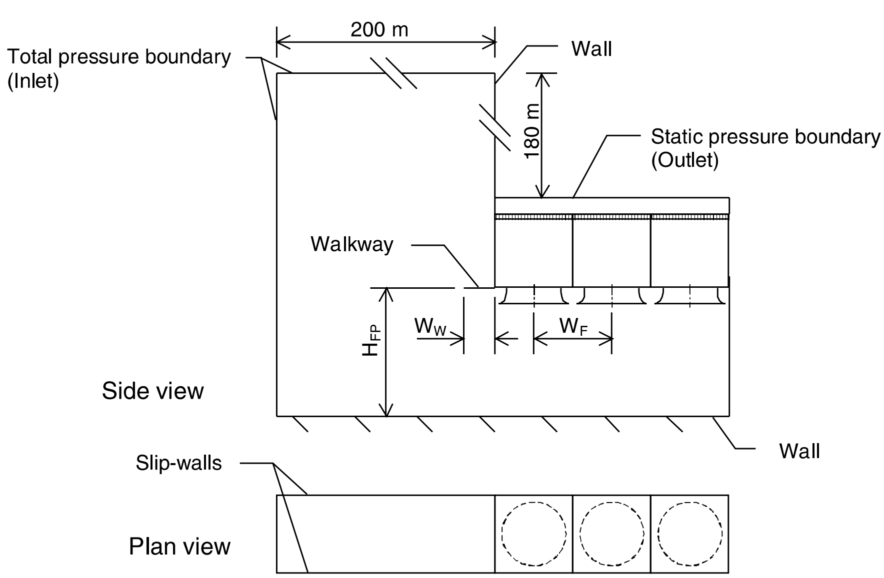

Fig. 5. Computational geometry and boundary conditions.

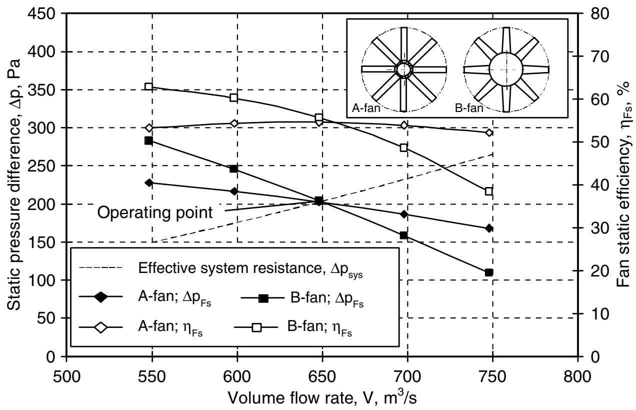

Fig. 6. Numerically determined fan characteristics and ideal operating point.

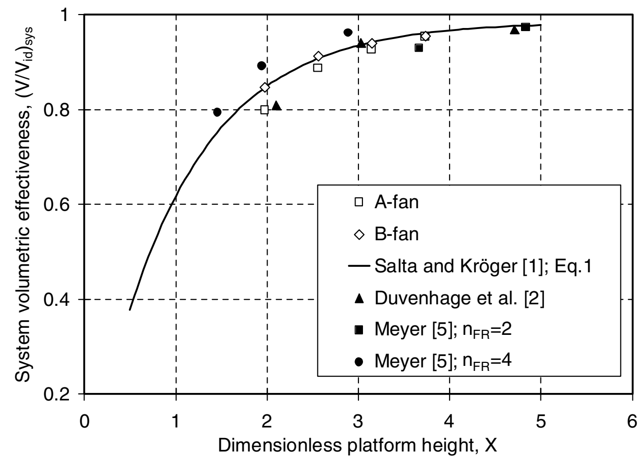

Fig. 7. System volumetric effectiveness.

3. Results and discussion

In order to compare the flow rate of fans operating under distorted inlet conditions to the ideal case with no inlet flow distortions, the following parameters are defined. The volumetric effectiveness of a fan (or fan-heat exchanger combination), V/V_{id}, is defined as the ratio of the flow rate through a particular fan unit in a multi-fan system simulation (V), to the reference flow rate of the same freestanding fan unit (V_{id}). The system volumetric effectiveness, $(V/V_{id})_{sys}$, is the average of the individual volumetric effectiveness's of all the fans in a multi-fan system.

3.1. Effect of platform height

Salta and Kröger [1] presents the following empirical relation to describe the reduction in system volumetric effectiveness, $(V/V_{id})_{sys}$, with a decrease in platform height, H_{FP}, for an ACHE or ACC with n_{FR} fan rows

$$(V/V_{id})_{sys} = 0.985 - \exp^{(-X)} \tag{1}$$

where X is a dimensionless platform height and defined by

$$X = \frac{(1 + 45/n_{FR})H_{FP}}{(6.35 d_F)} \tag{2}$$

The numerically predicted system volumetric effectiveness for the A-fan and the B-fan in the current investigation ($n_{FR} = 6$, $W_F/d_F = 1.29$, $d_h/d_F = 0.153$ for the A-fan and $d_h/d_F = 0.4$ for the B-fan, with no walkway added) as a function of X, are compared to the empirical relation of Salta and Kröger [1] ($n_{FR} = 2$, 4, 8 and 12, $W_F/d_F = 1.27$ and $d_h/d_F = 0.26$), in addition to numerical results of Duvenhage et al. [2] ($n_{FR} = 2$, $W_F/d_F = 1.18$ and $d_h/d_F = 0.232$) and Meyer [5] ($n_{FR} = 2$ and 4, $W_F/d_F = 1.27$ and $d_h/d_F = 0.4$), in Fig 7.

It can be seen that the numerically predicted system volumetric effectiveness for the A-fan and the B-fan display the same trend to that predicted by Eq. (1), and results obtained by Duvenhage et al. [2] and Meyer [5]. The system volumetric effectiveness of the A-fan is however notably less than that of the B-fan.

The individual fan volumetric effectiveness for the A-fan and the B-fan are shown in Fig. 8. It can be seen that the B-fan is significantly less affected by inlet flow distortions in terms of a reduction in volumetric effectiveness compared to the A-fan. The difference in performance of the A-fan and the B-fan can in part be explained by referring to the corresponding fan static pressure curves, shown in Fig. 6. The B-fan features a relatively large hub and inherently exhibits a steeper fan static pressure gradient compared to that of the A-fan. Consequently, for an additional flow loss or pressure drop caused by inlet flow distortions, denoted δp_i, the resultant change in volume flow rate for the A-fan and B-fan is respectively δV_A and δV_B, with $\delta V_A > \delta V_B$, as illustrated in Fig. 9. Note however that although the fan characteristics are influenced by cross-flow and inlet flow distortion, as respectively reported by Hotchkiss

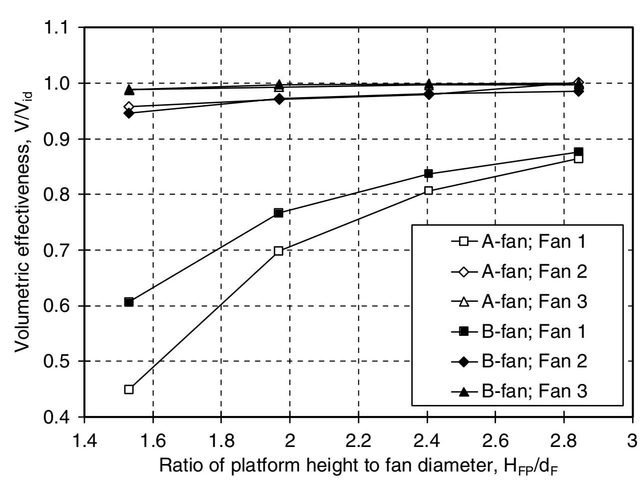

Fig. 8. Volumetric effectiveness of fans 1–3 for the A-fan and B-fan.

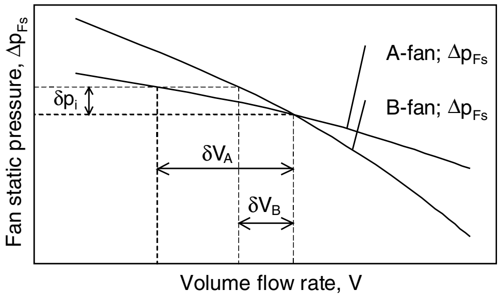

Fig. 9. Effect of inlet flow distortion on the volume flow rate of the A-fan and B-fan.

et al. [14] and Thiart and von Backström [10], the fan static pressure curve of the B-fan would still display a steeper gradient than that of the A-fan under similar inlet conditions. Stinnes [12] compared the experimentally determined performance characteristics of two fans having hub-tip-ratios of respectively $d_h/d_F = 0.26$ and $d_h/d_F = 0.4$, over a range of cross-flow inlet conditions. The current investigation presents similar results to that of Stinnes [12], where it emerged that the fan with $d_h/d_F = 0.4$ was less sensitive to cross-flow than the fan with $d_h/d_F = 0.26$.

Although a dimensionless platform height of $X = 1.53$ is perhaps unrealistically low for a practical ACSC, Duvenhage and Kröger [15] states that, "The influence of wind on fan performance appears to exhibit features similar to the influence of platform height on fan performance". The reduction in volumetric effectiveness under windless conditions at low platform heights, i.e., induced cross-draft conditions, may to a certain extent represent the effect of cross-winds on the fans located on the windward edge of the fan platform. The relevance of this investigation is therefore not only limited to the case of windless conditions.

3.2. Effect of a walkway

Previous experimental studies have shown that the volumetric effectiveness of the fans in an ACSC (or ACHE) can significantly be improved by adding a walkway to the edge of the fan platform as shown in Fig. 5. This was found to be especially applicable to the edge fans at low platform heights.

Salta and Kröger [1] recommend a walkway width of between $W_w/d_F = 0.159$ and $W_w/d_F = 0.476$. The improvement in volumetric effectiveness of the edge fan, for both the A-fan and the B-fan, with the addition of a walkway with a width of $W_w/d_F = 0.33$, is depicted in Fig. 10. Based on the critical evaluation and comparison of the velocity vectors and pressure distributions, given by Bredell [13], it can be deduced that the increase in flow rate is mainly as a result of reduced cross-flow velocity and abatement of separation or distortion occurring at the inlet of the edge fan.

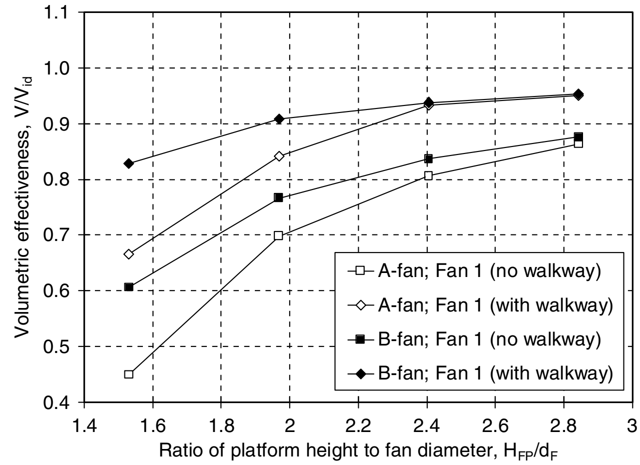

Fig. 10. Effect of a walkway on volumetric effectiveness of the edge fan.

4. Conclusion

Due to the proximity of the ground, an unfavourable cross-flow velocity component is induced underneath the fan platform of an ACSC. Depending on the height of the fan platform, the induced cross-draft may result in flow separation and distortion at the fan inlets. The reduction in volumetric effectiveness with a decrease in platform height was successfully modelled and results were found to be consistent with previous experimental and numerical investigations. Based on interpretation of the graphical representation of the associated flow field (i.e., vector, pressure and streamline plots) the conclusion was drawn that inlet flow distortions have an adverse effect on fan flow rate due to a combination of the following factors or mechanisms:

- Increased inlet flow losses (i.e., flow separation at the edge of the fan inlet) resulting in a decrease in flow rate.
- Maldistribution of air into the fan (i.e., non-uniform inlet profiles) resulting in fan blade stall and reduction in aerodynamic efficiency.
- Off-axis inflow conditions, which decreases the static pressure rise generated by the fan.

Because of the complex nature of the flow it is difficult to isolate and quantify the relative contribution of each of the abovementioned factors. This could perhaps be addressed in future research. The prototype B-fan ($d_h/d_F = 0.4$) was found to be less sensitive to inlet flow distortion compared to the A-fan ($d_h/d_F = 0.153$). The conclusion was drawn that the type of fan used will influence the effectiveness of a forced draft ACSC (or ACHE) subjected to inlet flow distortions. Furthermore, it was shown that by adding a solid walkway to the edge of the fan platform, significantly improved the performance of the edge fan in the case of the A-fan, as well as the B-fan.

References

[1] C.A. Salta, D.G. Kröger, Effect of inlet flow distortions on fan performance in forced draft air-cooled heat exchangers, Heat Recovery Systems & CHP 15 (1995) 555–561.

[2] K. Duvenhage, J.A. Vermeulen, C.J. Meyer, D.G. Kröger, Flow distortions at the fan inlet of forced draft air-cooled heat exchangers, Applied Thermal Engineering 16 (1996) 741–752.

[3] R.R.M. Spiers, Inlet tests on a full-size air-cooled heat exchanger, National Engineering Laboratories, NEL/HFTS 12, H.F.T.S. Paper RS361, 1981, pp. 86–104.

[4] C.M.B. Russel, J. Peachy, Air inflow effects on fan performance in air-cooled heat exchangers, Int. Conf. on Fan Design & Applications, Guilford, England, 7–9 September 1982.

[5] C.J. Meyer, Numerical investigation of the effect of inlet flow distortions on forced draft air-cooled heat exchanger performance, Applied Thermal Engineering 25 (2005) 1634–1649.

[6] P.R.P. Bruneau, The design of a single rotor axial flow fan for a cooling tower application, MSc Eng (Mechanical) Thesis, Department of Mechanical Engineering, University of Stellenbosch, South Africa, 1994.

[7] S.V. Patankar, Numerical Heat Transfer and Fluid Flow, McGraw-Hill, New York, 1980.

[8] B.E. Launder, D.B. Spalding, The numerical computation of turbulent flows, Computer Methods in Applied Mechanics and Engineering 3 (1974) 269–289.

[9] C.J. Meyer, D.G. Kröger, Numerical investigation of the effect of fan performance on forced draft air-cooled heat exchanger plenum chamber aerodynamic behaviour, Applied Thermal Engineering 24 (2004) 359–371.

[10] G.D. Thiart, T.W. von Backström, Numerical simulation of the flow field near an axial flow fan operating under distorted inflow conditions, Journal of Wind Engineering and Industrial Aerodynamics 45 (1993) 189–214.

[11] British Standards Institution, BS 848: Part 1: 1980, Fans for General Purposes, Part 1: Methods for Testing Performance, 1980.

[12] W.H. Stinnes, The performance of axial fans subjected to forced cross-flow at inlet, MSc Eng (Mechanical) Thesis, Department of Mechanical Engineering, University of Stellenbosch, South Africa, 1998.

[13] J.R. Bredell, Numerical investigation of fan performance in a forced draft air-cooled steam condenser, MSc Eng (Mechanical) Thesis, Department of Mechanical Engineering, University of Stellenbosch, South Africa, 2005.

[14] P.J. Hotchkiss, C.J. Meyer, T.W. von Backström, Applied Thermal Engineering, in press (available on-line).

[15] K. Duvenhage, D.G. Kröger, The influence of wind on the performance of forced draft air-cooled heat exchangers, Journal of Wind Engineering and Industrial Aerodynamics 62 (1996) 259–277.