Performance of low noise fans in power plant air cooled steam condensers

Sybrand J. van der Spuy[a], Theodor W. von Backström[b] and Detlev G. Kröger[c]

(Received: 09 January 2008; Revised: 13 May 2008; Accepted: 18 May 2008)

Axial fans are often installed in locations where the orientation and surrounding infrastructure can have a detrimental effect on the fan performance indicated by the manufacturer. This paper addresses various aspects of phenomena related to the installation of axial fans, one of these being the use of low-noise fans, and how these can be considered in the CFD performance evaluation of modern air-cooled power plant condensers.

Primary subject classification: 11.4.1; Secondary subject classification: 53.1

1 INTRODUCTION

In air-cooled power plant steam condensers, cooling is achieved by blowing air across the finned tube bundles arranged in the form of an A-frame above large-diameter axial flow fans (see Fig. 1). The fans are installed with the plane of rotation horizontally and are driven by electric motors through a gearbox. The fan and A-frame units are arranged in series to form a fan row, with a number of fan rows serving a single turbine unit in parallel. The result is that a power station will have a large array of fan units depending on the number of turbine units. The world's largest direct air-cooled power plant has an array of 288 axial fans, 9.1 m in diameter, located 45 m above ground level[1].

The performance characteristics of these fans have to be such that a prescribed air flow rate is guaranteed for specified flow resistances caused by the heat exchanger bundles and other obstructions, and by non-ideal flow patterns, while at the same time not exceeding prescribed noise levels. The required flow rate, coupled to the pressure losses, is regarded as the primary performance requirements of an installation, since it is directly linked to the effectiveness of the power generation process. The prescribed noise level is seen as a secondary requirement that is based on regulatory restrictions, often linked to the location of the installation. If a fan does not meet the primary performance requirement, it often exhibits increased noise levels due to increased unsteadiness in the flow[2]. Neise[2] referred to tests done with a 90° duct bend at various axial distances upstream of an axial flow fan. He reported that at short distances, the low frequency random noise components were increased by as much as 14 dB, while at the blade passing frequency an increase in the order of 7 dB was observed.

Recirculation of hot plume air and poor performance of the fans located near the edges of the array have been observed in large air-cooled steam condensers. In extreme cases, backflow of air through the fan occurred during windy periods[3,4]. The orientation of the fans means that their axes of rotation are vertical. The fans therefore have flow entering from a direction that is perpendicular to its axis of rotation. This causes fan inlet losses due to the separation of flow at the lip of the fan inlet as well as the off-axis inflow of air into the fan. Fans that are located near the edge or periphery of the array of air cooled condensers are severely affected by flow separation, while off-axis inflow occurs widespread through all fans installed in the array[5].

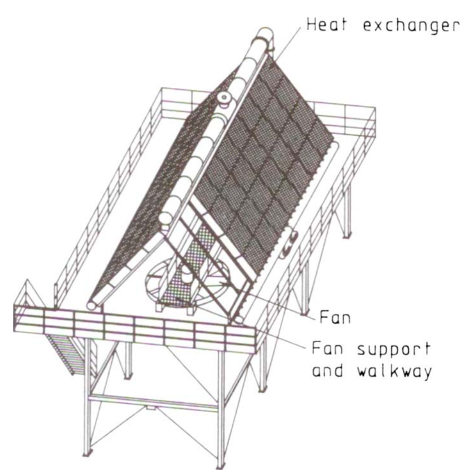

Fig. 1—Typical A-frame air-cooled condenser (Kröger[1]).

[a] University of Stellenbosch, Department of Mechanical Engineering, Private Bag X1, Matieland, 7602, SOUTH AFRICA; email: sjvdspuy@sun.ac.za

[b] University of Stellenbosch, Department of Mechanical Engineering, Private Bag X1, Matieland, 7602, SOUTH AFRICA; email: twvb@sun.ac.za

[c] University of Stellenbosch, Department of Mechanical Engineering, Private Bag X1, Matieland, 7602, SOUTH AFRICA; email: dgk@sun.ac.za

Stinnes et al.[6] derived a relatively simple, though highly effective, model to describe the decrease in performance due to off-axis inflow, based on a series of experiments during which fans were tested with inlet ducts at specific angles to the fan plane of rotation. A number of authors have modelled and investigated the inlet loss effect on fan performance using computational fluid dynamics (CFD)[3–5,7,8]. The circumferential variation in inlet conditions directly upstream of the fan rotation plane causes a significant cyclic variation in the loading of the fan blades and consequently acts as a source of fan blade fatigue and fan noise[2,9]. The use of CFD would potentially enable the plant designer or more specifically the fan designer, to make the necessary adjustments to the plant and fan design to minimize inlet losses. Unfortunately the use of CFD to model these conditions also has its limitations.

Due to the occurrence of backflow through some sections of the fan, conventional, simplified CFD fan models that only take into account the forward flow operation of a fan are not representative. Under these conditions the use of a full 3-dimensional CFD model of the fan or a novel "actuator disk model"[10] is recommended. These models are both however computationally intensive and therefore a simpler approach, referred to as the "pressure jump model", can be applied when flow distortions are less prominent. A number of fan installation and fan configuration effects have been investigated using this combination of methods. These will be discussed in more detail in this document.

2 THE ACTUATOR DISK MODEL

2.1 General Description

The actuator disk model and its application in CFD have been well researched and described in much detail by Meyer et al.[10] The actuator disk model simulates the effect of the individual fan blades on the flow field using blade element theory (see Fig. 2).

The lift and drag forces, δ_L and δ_D, [N] acting on a fan blade element of radial length δr [m] are calculated using the following equations:

$$\delta_L = \frac{1}{2}\rho |W_\infty|^2 C_L \times c \times \delta r \qquad (1)$$

$$\delta_D = \frac{1}{2}\rho |W_\infty|^2 C_D \times c \times \delta r \qquad (2)$$

where ρ is the air density [kg/m^3], W_∞ is the average relative velocity over the blade element [m/s], C_L and C_D are the coefficients of lift and drag (obtained from standard airfoil data based on an angle of attack α) and c is the average chord length of the blade element [m].

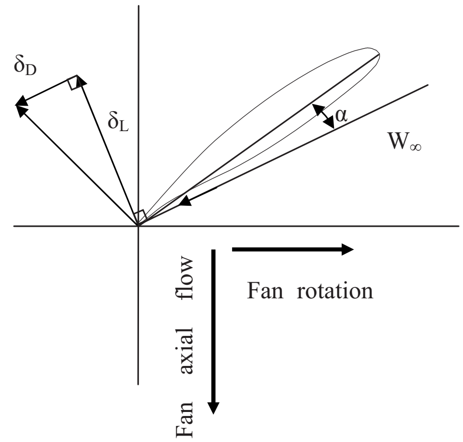

Fig. 2—Fan blade element.

Once the forces acting on the air stream are known, these are transformed into source terms that are inserted into the equation for linear momentum as follows:

$$\mathrm{s} - \nabla p + \nabla \cdot \tau_{ij} = \rho \frac{d\mathrm{V}}{dt} \qquad (3)$$

where s is the per volume force vector of source terms [N/m^3], ∇p is the gradient of static air pressure [N/m^3], τ_{ij} is the shear stress tensor [N/m^2] and V is the absolute velocity vector of the flow field [m/s].

2.2 Fan Model Validation

The fan considered in this analysis was a 9.145 m diameter, 8-bladed, cooling fan with a hub-to-tip ratio of 0.15, operating at 125 RPM (referred to as the A-fan). Details of the fan blade chord distribution, angle distribution and profile lift and drag coefficients are presented by Bredell[11]. Bredell calculated the lift and drag coefficients for the blade profile over a range of $-180°$ to $+180°$ using CFD. This enabled the actuator disk model to solve the momentum source terms for flow coming from any direction (including backflow) through the rotor disk. The actuator disk model used in this analysis was validated by comparing results from the supplier fan curve to results obtained using the actuator disk model (see Fig. 3), where Υ_{pt} refers to a setting angle at the blade tip based on the line tangent to the bottom of the blade profile. The results obtained from the actuator disk model were calculated according to the guidelines of the test standard used by the supplier, namely BS 848 part 1 (1980), type A[12]. The results show excellent correlation between the supplier and simulated data in the operating range of the fan (between 500 m^3/s and 700 m^3/s) for the fan static pressure.

All CFD simulations were performed using FLUENT™ version 6.2.16. To model the test condi-

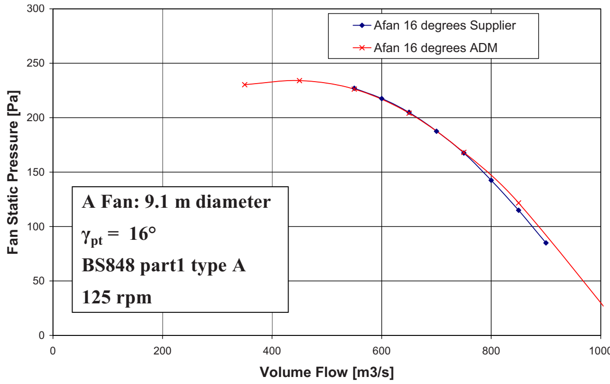

Fig. 3—Actuator disk model validation (fan static pressure).

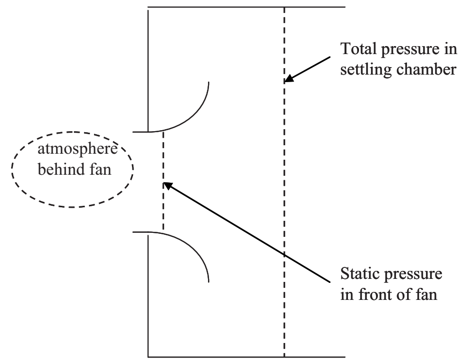

Fig. 4—Derivation of static-to-static pressure jump.

tions dictated by the British Standards, the inlet boundary was specified to be a mass flow inlet, while the outlet boundary was specified to be a total pressure boundary (pressure value set to atmospheric). To allow for dissipation of the fan exhaust dynamic component, the exhaust atmosphere was modelled to have a diameter of 4 × fan diameter and a length of 8 × fan diameter. The CFD model contained 550000 cell volumes. The validated CFD model used the realizable k-ε model[13] to simulate turbulence and the QUICK[14] interpolation scheme to calculate variables at the cell faces. The simulation was allowed to run for 3000 iterations or a residual value of 10^{-4}. Although the simulations were stable, convergence at flow rates less than 500 m^3/s were not good.

3 THE PRESSURE JUMP METHOD

3.1 General Description

The motivation behind considering the use of a pressure jump method lies in its potential ability to model an array of axial fans accurately using a reduced number of cell volumes in CFD. The pressure jump method detailed in this document is in essence the same technique as that used by van Staden[4] to model the performance of axial fans. The difference between the method detailed in this document (referred to as the "pressure jump method") and the one used by van Staden is however the way in which the effect of the fan is implemented into the CFD code. The pressure jump method assumes a static-to-static pressure jump that occurs at the location of the fan rotation plane. This static-to-static pressure value is added to the static pressure term of the linear momentum equation in the flow field directly upstream of the fan rotation plane, shown in Eqn. (3).

Hotchkiss et al.[5] and Stinnes et al.[6] found that under cross-flow conditions (that lead to off-axis inflow) the "fan static pressure" is reduced in magnitude by the dynamic pressure associated with the cross-flow component immediately upstream of the fan ("fan static pressure", as referred to by typical fan supplier data and simulated by the actuator disk model is actually fan total-to-static pressure). The cross flow component affects the static pressure in front of the fan and not the actual value of static-to-static pressure increase. This is shown by Hotchkiss et al.[5] to be attributed to the fact that the cross flow effect on flow angles and velocities over the fan blades effectively cancels out when considering a fan rotor with blades running with and against the direction of cross flow. Based on these results the pressure jump method should yield accurate results when analysing fans subjected to cross flow only. The same can however not be said for flow separation that occurs over a localised area in front of the fan rotation plane. It is therefore expected that, although the pressure jump method would identify possible problematic intakes at the side of an axial fan array, the results would not be accurate and a more accurate analysis would be required.

The fan supplier data was compiled for a type-A fan installation (see Fig. 4). The fan pressure data is derived from an average static pressure value that is measured in a plane, relative to atmosphere, in a settling chamber, 1.25 fan diameters upstream from the fan, where the axial velocity is specified to be less than 2 m/s. The static pressure measured in this location is assumed to equal the total pressure in this location. The total pressure loss between the measurement plane and the fan rotation plane is considered negligibly small because of the smooth bell mouth inlet (as specified by BS 848[12]).

To calculate the static pressure directly upstream of the fan rotation plane, as required for the pressure jump method, the dynamic pressure in the fan rotation plane is added to the "fan static pressure" curve. During the validation of the pressure jump method, it was found that the initial assumption of zero total pressure losses between the measurement plane and fan rotation plane

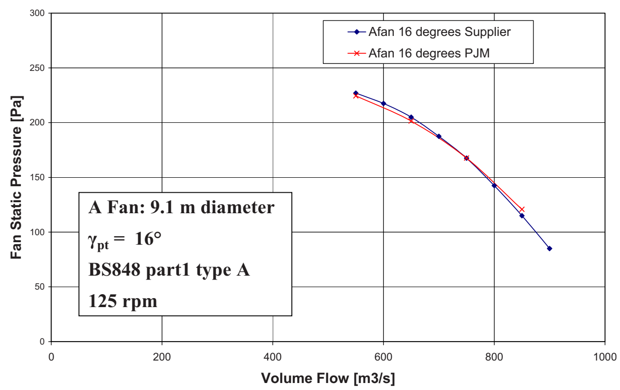

Fig. 5—Pressure jump method validation (fan static pressure).

was not sufficiently accurate and a loss coefficient was subsequently added. A value of 0.07 was used for the loss coefficient, which was based on flow data for rounded inlets, published by Idelchik[15]. The "fan static pressure" curve was therefore translated into a pressure jump value as follows:

$$\Delta p_{fan} = a + bV + cV^2 + dV^3 + \frac{1}{2}\rho V^2 + K_{loss}\frac{1}{2}\rho V^2 \tag{4}$$

where the values for a, b, c and d were derived from a curve-fit as described earlier, V is the average velocity perpendicular to the fan rotation plane [m/s] and K_{loss} is the described loss coefficient.

3.2 Fan Model Validation

The same geometric model that was used to validate the actuator disk method was used to validate the pressure jump method. Instead of using the described momentum sources, the standard FLUENT™ interface for specifying a pressure jump was used. The cell face region where the pressure jump would occur coincided with the fan rotation plane. The same boundary conditions, turbulence model and overall numerical differencing scheme were used as for the actuator disk method. The simulation was once again allowed to run for 3000 iterations or a residual value of 10^{-4}. The simulations were found to be stable and convergence generally occurred after 500 iterations. The resulting comparison of simulated and supplier data showed excellent correlation (see Fig. 5).

4 SIMULATION OF INSTALLED AXIAL FANS

4.1 Computational Model

To simulate the performance of axial fans under installed conditions, a 3-fan section of an array of air-cooled condensers was modelled (see Fig. 6).

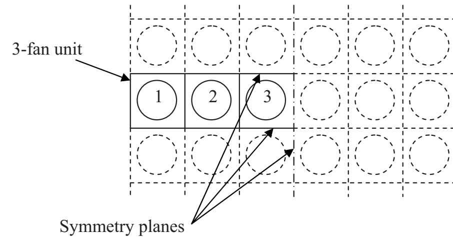

Fig. 6—A 3-fan section of an air-cooled condenser array, viewed from above.

Each of the fan units was modelled to consist of a bell mouth inlet, axial fan, rectangular plenum chamber and heat exchanger. The model had a total pressure boundary 200 m upstream from the fan array and a static pressure boundary 2 m downstream of the fan array (see Fig. 7). The analysis focussed on the inlet effects only, therefore the exit conditions of the system were simplified accordingly.

The heat exchanger was modelled as a porous region with resistance properties given by the equations from Bredell[11]:

$$\Delta p_{HE} = -(4.132315 \times 10^{-4}Q^2 + 5.629484 \times 10^{-2}Q) \tag{5}$$

where Q is the volume flow rate through the heat exchanger [m^3/s]. The above equation for system resistance coupled to the A-fan characteristics as shown in Fig. 3 corresponds to a reference flow condition of 650 m^3/s.

4.2 Model Validation

The CFD model for a 14 m platform height contained 570000 cell volumes. The validated CFD model once again made use of the realizable k-ε model and the QUICK interpolation scheme. The CFD model for the 3-fan unit was validated by comparing the results from the model, using both the actuator disk model and pressure jump method to simulate the A-fan, with the empirical relation derived by Salta et al.[16] The results showed the volumetric effectiveness of a multiple fan installation as a function of dimensionless platform height as follows:

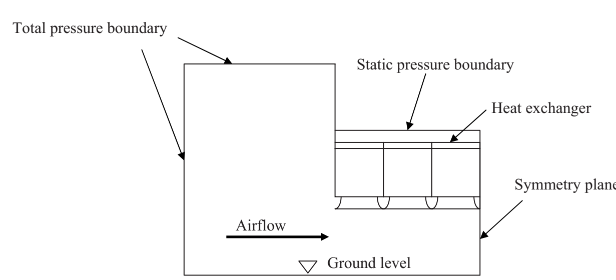

Fig. 7—Side view of computational domain for 3-fan unit.

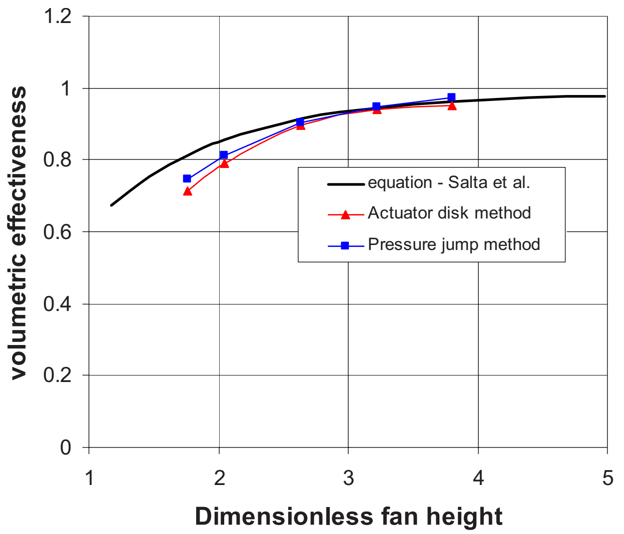

Fig. 8—Comparison of system volumetric effectiveness.

$$(Q/Q_{ref})_{system} = 0.985 - e^{-X} \tag{6}$$

where X is the dimensionless platform height:

$$X = \frac{(1 + 45/n) \times H}{6.35 \times D_F} \tag{7}$$

In the above equation, H is the platform height [m], n is the total number of fans per row (in other words 6 for the modelled 3-fan unit) and D_F is the fan shroud diameter [m]. The reference flow when determining the volumetric effectiveness (Q/Q_{ref}) of the installation was 650 m^3/s. A comparison of the results to the empirical correlation is shown in Fig. 8. The results show good correlation with the equation of Salta, at a dimensionless platform height between 2.5 and 4. The Salta fans had different ratios of dynamic pressure based on throughflow to pressure rise, compared to the A-fan. This leads to a different sensitivity to cross-flow and possibly to distortion and explains the difference in results at lower platform heights. Fig. 9 shows a vector plot with static pressure distribution to illustrate the extent of flow separation experienced by the edge fan of a multiple fan installation.

5 RESULTS

5.1 Fan Model Investigation

The 3-fan unit was first modelled by applying the actuator disk model in all three fans and subsequently compared to results obtained by applying the pressure jump method in all three fans. It was finally compared to a simulation using the actuator disk model on the "edge" fan only, combined with using the pressure jump method on the two inner fans. The actuator disk model is applied to the "edge" fan because of its ability to simulate fan operation when backflow occurs through the fan. The results compare volumetric effectiveness at a height of 14 m and are shown in Fig. 10.

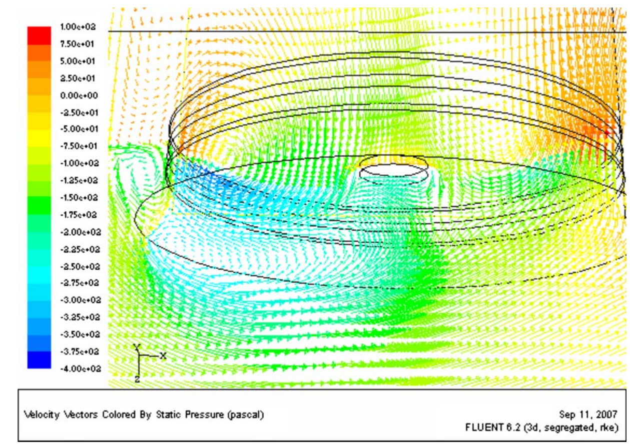

Fig. 9—Vector plot at fan inlet illustrating recirculation zone.

5.2 Platform Height Investigation

The 3-fan unit, using the above combination of fan models, was modelled with various platform heights, ranging from 14 m to 26 m as shown in Fig. 11.

5.3 Fan Geometry Investigation

The combined 3-fan unit was modelled with the standard 9.145 m cooling fan described in this document. This fan is referred to as the "A-fan" by Bredell et al.[9] and has a hub-to-tip ratio of 0.15. The alternative fan was also a 9.145 m fan but with a hub-to-tip ratio of 0.4 and is referred to as the "B-fan" by Bredell. Under standard test conditions, Bredell points out that the B-fan exhibits a much steeper fan static pressure to volume flow rate curve than the A-fan. This is typically found when referring to the performance curves of "low-noise" fans and is the result of a relatively larger value for fan solidity (see Fig. 12). The investigation effectively compares the volumetric effectiveness of a standard industrial

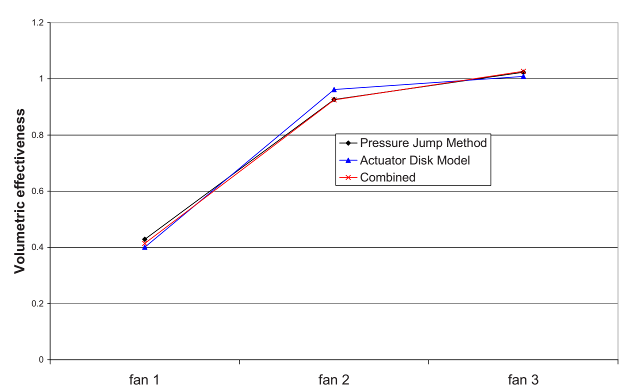

Fig. 10—Comparison of fan modelling methods.

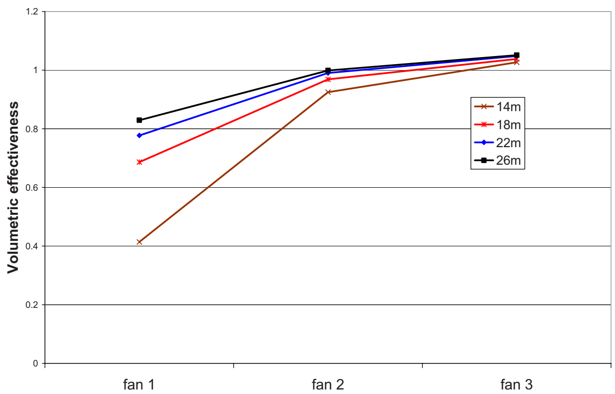

Fig. 11—Results from platform height investigation.

cooling fan (A-fan) to a "low-noise" industrial cooling fan. The results for the 3-fan unit, comparing the volumetric effectiveness of the A-fan to that of the B-fan at a 14 m platform height is shown in Fig. 13.

5.4 System Configuration Investigation

To illustrate the possible application of the fan model, an investigation to show the effect of a building located a distance of 10 m upstream of the fan array on

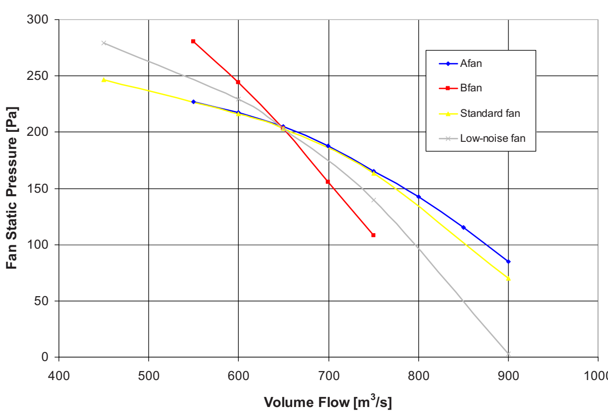

Fig. 12—Typical fan static pressure graphs for industrial fans.

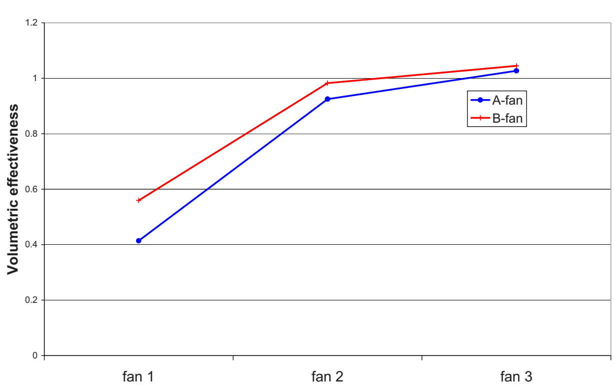

Fig. 13—Comparison of A-fan and B-fan performance at 14 m *height.*

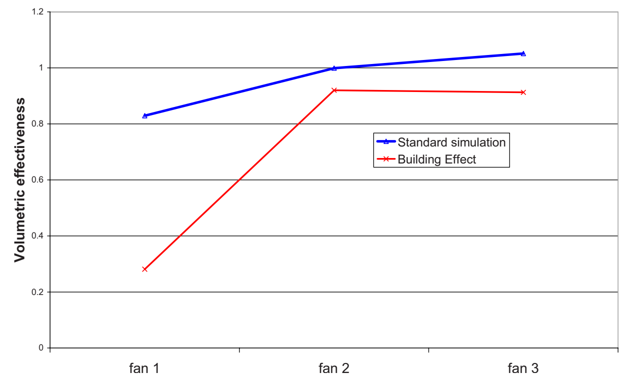

Fig. 14—The effect of buildings on the performance of a fan array at 26 m *height.*

the volumetric effectiveness of the fan array was conducted. The specific distance (10 m) was chosen purely as an example, although as a rule of thumb, any value in the order of or less than the specified platform height should have a detrimental effect on the volumetric effectiveness of the fans. It should be noted that the investigation only considered the effect on the inlet side of the fan array and no allowance was made for interaction between the exhaust and inlet sides. The results for a platform height of 26 m are shown in Fig. 14.

6 DISCUSSION

This document describes various methods of simulating the performance of axial fans under installed conditions. The extent to which an air-cooled condenser plant can be modelled in CFD on a single processor is limited by the size of the geometry being modelled. Distorted inlet conditions generate flow separation at the edge of the fan inlet and off-axis inflow into the fans. The separation that occurs is localised on the edge-side of the inlet of the fans installed on the periphery of a fan array, while the off-axis inflow occurs on all fans installed in the fan array. The flow separation causes an off-balance inlet flow distribution that can be so severe that the edge fans experience back flow through the fan. The actuator disk model is therefore considered to be a good compromise when keeping the size of a CFD model to a minimum while still being able to model the effect that flow from various directions would have on the performance of a fan. Off-axis inflow is distributed across the whole face of a fan. Stinnes et al.[5] has shown that for angles less that 45° the effects of off-axis inflow cancel out on opposing sides of the fan face. Off-axis inflow causes a pressure loss in front of the fan but does not alter the fan performance curve. The pressure jump method is therefore ideal in its application on fans operating in the first quadrant (positive pressure rise and positive volume flow) only. This limits its use to fans on the inside of the fan array where flow separation is very small to negligible.

It has been found that it is essential to validate the fan models against results obtained under standard BS848 test conditions to ensure that relevant turbulence and discretization schemes are used. The actuator disk model was found to be very stable when using a first order discretization scheme for the continuity and momentum equations but considerable effort (a more detailed mesh and a larger outlet domain) was required to improve this stability when using a second order discretization scheme. It was also found essential to validate the pressure jump method so that a loss coefficient could be specified that accounts for total pressure losses between the measuring plane and the fan rotation plane.

Considering application of these methods to the modelling of power plant air-cooled steam condensers, the following should be taken into account:

1. Non-uniform inlet flow, caused by flow separation, is a potential noise mechanism in a fan installation and any method that would dampen its severity would therefore reduce the noise generated by the fan. Besides flow fluctuations, the non-uniform inlet also causes local regions of high relative velocities and a consequential large increase in fan noise[2].
2. The volumetric effectiveness of a fan array decreases dramatically with platform height, primarily due to the lower static pressure region below the "edge" fans due to increased flow separation in the fan inlet.
3. Fans having steeper fan static performance curves, as typically exhibited by low-noise fans, are less sensitive to flow distortions and exhibit a higher volumetric effectiveness.
4. The volumetric effectiveness of a fan array decreases with the proximity of buildings since it increases the cross flow velocity through the system and causes more severe flow separation at the edge fans.

Using the above CFD simulations, the user would be able to quantify possible increases in plant operating efficiency and compare it to the additional cost required for its manufacture. The biggest potential of the above CFD simulations lie in their ability to model a fan array accurately using a reduced number of cell volumes (conservatively estimated, simplified CFD methods use in the order of 10 times less cell volumes). This document however only investigated and validated the practical application of these methods and further development of the methods, specifically considering grid dependency, is required.

7 REFERENCES

1. D. G. Kröger, *Air-cooled Heat Exchangers and Cooling Towers*, PennWell Corporation, Tulsa, OK, (2004).
2. W. Neise, "Installation effects on fan noise", IMechE, C401/005, 83–91, (1990).
3. H. B. Goldschagg, F. Vogt, C. G. du Toit, G. D. Thiart and D. G. Kröger, "Air-cooled steam condenser performance in the presence of crosswinds", *Proceedings: Cooling Tower Technology Conference, Electrical Power Research Institute*, Palo Alto, California, (1997).
4. M. P. van Staden, *An Integrated Approach to Transient Simulation of Large Air-cooled Condensers using Computation Fluid Dynamics*, D. Eng. Thesis, Department of Mechanical Engineering, Rand Afrikaans University, South Africa, (2000).
5. P. J. Hotchkiss, C. J. Meyer and T. W. von Backström, "Numerical Investigation into the effect of cross-flow on the performance of axial flow fans in forced draught air-cooled heat exchangers", *Appl. Therm. Eng.*, **26**, 200–208, (2006).
6. W. H. Stinnes and T. W. von Backström, "Effect of cross-flow on the performance of air-cooled heat exchanger fans", *Appl. Therm. Eng.*, **22**, 1403–1415, (2002).
7. G. D. Thiart and T. W. von Backström, "Numerical Simulation of the flow field near an axial fan operating under distorted inflow conditions", *J. Wind. Eng. Ind. Aerodyn.*, **45**, 189–214, (1993).
8. J. R. Bredell, D. G. Kröger and G. D. Thiart, "Numerical Investigation of fan performance in a forced draft air-cooled steam condenser", *Appl. Therm. Eng.*, **26**, 846–852, (2006).
9. J. R. Bredell, D. G. Kröger and G. D. Thiart, "Numerical Investigation into Aerodynamic Blade Loading in Large Axial Flow Fans Operating Under Distorted Inflow Conditions", *R&D Journal*, **22**, 11–17, (2006). AQ: #1
10. C. J. Meyer and D. G. Kröger, "Numerical simulation of the flow field in the vicinity of an axial flow fan", *Int. J. Numer. Methods Fluids*, **36**, 947–969, (2001).
11. J. R. Bredell, *Numerical investigation of fan performance in a forced draft air-cooled steam condenser*, M.Sc. Eng. Thesis, Department of Mechanical Engineering, University of Stellenbosch, South Africa, (2005).
12. *Fans for General Purposes, Part 1: Methods for Testing Performance*, British Standards 848 1980, British Standards Institution, London, United Kingdom, (1980).
13. T.-H. Shih, W. W. Liou, A. Shabbir, Z. Yang and J. Zhu, "A new k-ε eddy-viscosity model for high Reynolds number turbulent flows—model development and validation", *Comput. Fluids*, **24**(3), 227–238, (1995).
14. B. P. Leonard, "A stable and accurate convective modelling procedure based on quadratic upstream interpolation", *Comput. Methods Appl. Mech. Eng.*, **19**, 59–98, (1978).
15. I. E. Idelchik, *Handbook of Hydraulic Resistance*, 3rd Edition., Begell House, (1994).
16. C. A. Salta and D. G. Kröger, "Effect of inlet flow distortions on fan performance in forced draft air-cooled heat exchangers", *Heat Recovery Syst. CHP*, **15**, 555–561, (1995).

AUTHOR QUERIES — 001904NCE

#1 Au: Please provide coden/issn for Ref. 9