

THE SIMULATION OF AN AXIAL FLOW FAN PERFORMANCE CURVE AT LOW FLOW RATES

S.J. van der Spuy, F.N. le Roux, T.W. von Backström, and D.G. Kröger Department of Mechanical and Mechatronic Engineering Stellenbosch University, Joubert Street, Stellenbosch, 7600, South Africa Email: sjvdspuy@sun.ac.za

ABSTRACT

Simplified fan models are often used to simulate the effect of axial flow fans in large arrays of air-cooled condensers. The actuator disc method and its shortcomings are discussed and illustrated using computational fluid dynamics. A full 3dimensional analysis of the fan is also performed at a single operating point to evaluate the flow conditions around the blade. This analysis confirms the results of the actuator disc method at the specific operating point. An adaptation of the actuator disc method, to improve its ability to model fan operation at low flow rates, is proposed. The effectiveness of the adaptation is evaluated by comparing the fan static pressure curve obtained from experimental results to the fan static pressure curve obtained from the simulations. The comparison shows that an analysis using the adapted actuator disc method produces results that correlate well with experimental values.

INTRODUCTION

The condenser section of a direct dry-cooled power plant consists of an array of finned tube heat exchangers, with large diameter axial flow fans suspended underneath the heat exchangers. The 6 x 665 MW(e) Matimba power plant in South Africa incorporates 288 axial flow fans, each 9.1 m in diameter, installed at a height of 45 m [1]. Figure 1 is a photo depicting the air-cooled condenser (ACC) section of the Matimba plant. The fan arrangement is of the "forced draught" type, meaning that the cooling air is propelled upwards through the ACC by the axial flow fans.



Figure 1: Side view of Matimba power plant ACC (with permission of ESKOM).

The steam coming from the power plant's turbines is condensed in the heat exchangers. The axial flow fans blow the cooling air around the heat exchangers' finned tubes. A direct relationship therefore exists between the performance of the axial flow fans, the performance of the heat exchangers and the performance of the power plant. A study published by Goldschagg et al. [2] showed the thermal effectiveness of Unit 1 of the Matimba plant to be 62.5%, compared to a thermal effectiveness of 96.9% for Unit 4, with a 6 m/s north westerly wind blowing. This difference in thermal effectiveness is specifically attributed to the decreased flow rate through the axial flow fans in Unit 1 (caused by distorted inlet flow conditions), where the mass flow rate through these fans is only 62.3% of the mass flow rate through the fans of Unit 4.

Research has shown that the relative performance of individual fans within large fan arrays is dependent on the location of the fan within the fan array and it has been particularly observed that fans located along the perimeter of a large array of axial flow fans experience deterioration in performance due to a distortion of their inlet flow [3-6] (see Figure 2). The distortion of the inlet flow causes a reduction in flow rate through the fan and a subsequent reduction in heat transfer rate of the condenser system.



Figure 2: Distorted inlet flow at side of fan array.

The variation in the pressure capability of an axial flow fan with variation in volume flow is represented by its "fan performance curve", which, in the case of an ACC fan for the abovementioned plant specifically, represents the change in "fan static pressure" with a change in volume flow rate. Since the axial flow fans that form the ACC fan array are all identical in geometry and lay-out (for logistical reasons), the variation in relative fan performance inside the array means that the different fans would operate at various points along the characteristic fan performance curve.

Due to the physical size of the ACC of a dry-cooled power station, accurate airflow measurements are often costly and difficult and therefore there is a need to simulate the operation of the axial flow fans in relation to other components within the ACC plant using computational fluid dynamics (CFD) [7-9]. As shown later in this paper, about 5 million cells were needed to model a 1/8th section of an axial flow fan accurately under axisymmetric inlet flow conditions. Although a full 3-dimensional unsteady simulation of the distorted inlet flow of a single axial flow fan is possible, the large computer resource required means that the multiple axial flow fans in a fan array is

modelled using simplified fan models, such as the actuator disc or pressure jump method [10].

The actuator disc method is used to simplify the CFD model of an axial flow fan by replacing the individual fan blades with momentum source terms based on the aerodynamic characteristics of the fan blades [8, 9]. The actuator disc method ignores the effect of radial flow over the fan blades. This is evident when modelling an axial flow fan in a fan test arrangement at low flow (post stall) conditions. At low flow rates, flow-induced blockage at the hub of a typical axial flow fan forces the incoming flow radially out towards the tip of the fan blade. This radial flow is associated with a large increase in pressure rise over the fan, as is evident when looking at the fan static pressure versus volume flow rate curve of an axial flow fan (see Figure 14).

The inability of the actuator disc method to take into account the effect of radial flow through the fan rotor means that it under predicts the fan's pressure capability at low flow rates. This has implications for the accuracy with which axial flow fans at various locations in a fan array can be modelled using simplified models in CFD. Van der Spuy et al. [11] show that, for a two-fan CFD simulation, the simulated volumetric effectiveness of the system can deviate between 0% and 9.5% from an empirical curve published by Kröger [1], depending on the level of inlet distortion experienced by the fans.

This investigation focuses on the simulation of the fan static pressure curve of a particular fan referred to as the B-fan. This fan was designed by Bruneau [12], for specific use in aircooled condensers. The B-fan is an eight-bladed fan with a hubto-tip ratio of 0.4. It has an outer diameter of 1.542 m and was designed to run at 750 rpm. The B-fan blades use NASA LS10413 profiles [12], with the blade thickness varying from 13% of the chord length at the hub to 9% of the chord length at the blade tip.

For the purpose of the investigation the B-fan is tested using a standard fan test facility to obtain the fan static pressure versus volume flow rate curve from zero to maximum flow within the first quadrant of fan operation (the first quadrant refers to a fan that produces a net positive flow rate with a corresponding pressure increase over the fan). To evaluate the 3-dimensional flow surrounding the fan blade, a full 3dimensional simulation of the testing of the B-fan is performed at its design flow rate. The 3-dimensional flow around the fan blade is converted to 2-dimensional airfoil profile data, similar to profile data used in the actuator disc method. The testing of the B-fan is then simulated in CFD using the actuator disc method. Using results obtained from the preceding investigations, as well as results and methods described in literature, the actuator disc method is amended to make provision for the higher fan static pressure values observed at lower flow rates. Finally the experimental and CFD results are compared, conclusions are drawn and recommendations for future work are made.

STANDARD TESTING OF AXIAL FLOW FAN

The B-fan (diameter 1.542 m, speed 750 rpm) as described earlier was tested by le Roux [13] using a type A fan test installation, as defined by the British Standards 848 [14] (see Figure 3).



Figure 3: Sketch of BS 848 type A test facility (adapted from van der Spuy et al. [11]).

The use of a type A facility corresponds with the open space-to-open space lay-out of an ACC-fan. Although the fan test facility is axisymmetric in nature, compared to the nonaxisymmetric nature of the distorted inlet flow shown in Figure 2, the purpose of this investigation is to derive a simplified method that can represent the operation of an axial flow fan at low flow rates, that corresponds to the decreased flow rate experienced by a fan located at the perimeter of a fan array. When applying these methods to a fan experiencing distorted inlet flow, the assumption is therefore made that the fan is still operating along its "fan operating curve".

The test facility is installed at the Department of Mechanical and Mechatronic Engineering at Stellenbosch University. The facility consists of a calibrated inlet bellmouth (1) that is used to measure the volume flow rate through the facility. The flow rate through the facility is regulated using a throttling device (2). An auxiliary fan (3) is used to extend the flow range of the test facility and enables the measuring of the test fan's maximum flow rate (at zero pressure differential over the fan). The auxiliary fan is equipped with a set of flow straighteners (4) to remove the swirl downstream of the fan.

A diffuser section with its own guide vanes (5) connects the front-end of the test facility to the settling chamber (6). The dimensions of the settling chamber are 7 x 3.7 x 3.7 m. The settling chamber has a set of three meshed screens (7) that is used to reduce the size of any vortices that may exist in the flow. The test fan and bellmouth (8) are located at the rear end of the settling chamber and exhausts the flow from the settling chamber into the atmosphere. The test fan is driven by a hydraulic power pack outside the settling chamber. The fan static pressure is determined by measuring the static pressure at the sides of the settling chamber upstream of the fan. The fan static pressure is then calculated as (BS 848 [14]):

$$p_{FS} = \Delta p_{sett} - p_{dsett} \tag{1}$$

where the value for Δp_{sett} is the average value of measurements made at the sides of the settling chamber (relative to atmosphere) and the value for p_{dsett} is the dynamic pressure based on the average axial flow velocity in the settling chamber. The average flow velocity is calculated using the measured mass flow rate and air density in the settling chamber and the cross sectional area of the settling chamber.

To determine the power consumption of the fan, an in-line torque transducer is installed between the fan rotor and the hydraulic motor to measure the torque transferred to the rotor. The rotational speed of the rotor is measured using a magnetic speed pick-up installed next to the shaft of the fan. The power consumption of the fan rotor is then calculated as:

$$P_{shaft} = \frac{2\pi NT}{60} \tag{2}$$

The fan static efficiency can then be calculated:

$$\eta_{FS} = \frac{p_{FS}Q}{P_{shaft}} \tag{3}$$

The test facility lay-out, experimental and calibration procedure followed were identical to that of Meyer and Kröger [9]. The experimentally measured fan static pressure curve is shown in Figure 14. The repeatability of experimental results over the full range of the illustrated performance curve was within 1% (taking into account the specified test conditions shown in Figure 14).

To evaluate the flow around the fan blade experimentally, le Roux applied light wool tufts to the suction side of one of the fan blades. A stroboscope was used to obtain a stationary picture of the tufts which could then be photographed at different flow rates (see Figure 4).



Figure 4. Experimental evaluation of flow over fan blade at different flow rates [13].

Although the tufts were subjected to centrifugal loads which gave them an outward deflection, a comparative evaluation of the tufts at different flow rates clearly shows the occurrence of stall at the hub of the fan (where the circumferential velocity of the fan blade is lower than at the tip) as the flow rate decreases.

DETAILED CFD SIMULATION OF AXIAL FLOW FAN

Le Roux [13] performed a detailed simulation of the B-fan using the FINETM/Turbo CFD software package of Numeca. As explained in the Introduction, the purpose of this simulation was to investigate the 3-dimensional flowfield in the vicinity of the fan blade.

Fan blade modelling

The fan blades were modelled using the fan blade data described by Bruneau [12]. Seven profile data sets were generated at various radial positions along the length of the blade. A C-spline curve was fitted through each of these data sets. A set of 200 points per radial station was then extracted from each of the C-spline curves and used in the blade-generation process.

Computational domain

A 1/8th section of the fan was modelled using periodic boundaries (corresponding to the axisymmetric nature of the test facility). The domain was divided into an inlet, rotor and outlet section. Figure 5 shows the lay-out of the computational domain. The domain was meshed using a structured grid of 4.7 million cells.



Figure 5: Computational domain.

The inlet section of the domain was modelled to correspond to the inlet section of the test facility, with nonrotating sold walls on the sides of the inlet. The inlet boundary was modelled as a mass flow boundary.

The rotor section was modelled using an O4H-topology for the blades with a tip gap of 3 mm. The blade and hub surfaces were modelled as solid surfaces, rotating at 750 rpm (see Figure 6).



Figure 6: Fan blade mesh.

The outlet section was modelled as an "open atmosphere" with a static pressure boundary on all sides. Although this meant that backflow occurred at the sides of the outlet boundary, it was felt that such a format gave the best possible representation of the actual test facility.

Simulation

The simulation assumed steady state, incompressible flow, using the Spalart-Allmaras turbulence model. A 3-level multigrid process was followed where the FINETM/Turbo solver first obtains the solution for a coarse grid flowfield, before proceeding to simulate the flowfield with a finer grid. The y+values for the blade surface were less than 5, except for the blade tip region (in the tip gap) where the y+-values were less than 10.

Results

The simulations were performed at a flow rate of $16 \text{ m}^3/\text{s}$. The simulated static pressure value at $16 \text{ m}^3/\text{s}$ was 210 Pa, compared to a measured value of 207 Pa. However, the simulated fan static efficiency was 61%, compared to a measured value of 54%.

A comparison of the simulated flow over the fan blade at 16 m^3 /s to the experimental evaluation shown in Figure 4 shows a slightly larger stalled region for the experimental evaluation (see Figure 7). However, the difference between the two pictures is too small to draw any significant conclusions from it.



Figure 7. COMPARISON BETWEEN EXPERIMENTAL AND SIMULATED FLOW OVER FAN BLADE [13].

CFD SIMULATION USING THE ACTUATOR DISC METHOD

The discussed CFD simulation is described in detail by van der Spuy et al. [11]. The CFD simulation using the actuator disc method, and its associated adjustment (see further discussion), was the main focus of this investigation.

Outline of actuator disc method

The actuator disc method is based on the model described by Meyer and Kröger [9]. The actuator disc method simulates the effect of the individual fan blades on the flow field by using the lift and drag characteristics of the fan blade profile (see Figure 8). Each fan blade is subdivided into radial elements (corresponding to the mesh seed of the CFD model). The lift and drag forces acting per radial element length of blade (δ_L and δ_D) are calculated using the following equations:

$$\delta_L = \frac{1}{2} \rho |w_{\infty}|^2 C_L c \tag{4}$$

$$\delta_D = \frac{1}{2}\rho |w_{\infty}|^2 C_D c \tag{5}$$

The lift and drag coefficients are the same as those given in literature for standard 2-dimensional airfoil data based on an angle of attack α . It should be noted that $w\infty$ is the average relative velocity, composed of components in the axial and circumferential directions over the blade element only (the actuator disc model ignores radial flow).



Figure 8. Fan blade element lay-out (van der Spuy et al. [11])

Once the forces acting on the air stream are known, these are transformed into volumetric source terms (f_i) that are inserted into the equation for linear momentum (van der Spuy et al. [11]):

$$f_{i} - \frac{\partial p}{\partial x_{i}} - \frac{\partial}{\partial x_{i}} \left(\tau_{ij}\right)$$
$$= \frac{\partial}{\partial t} \left(\rho u_{i}\right) + \frac{\partial}{\partial x} \left(\rho u_{i}u_{j}\right)$$
(6)

where

$$f_x = -\frac{n}{2\pi r\Delta z} (\delta_L \sin\beta + \delta_D \cos\beta) \sin\theta$$
(7)

$$f_{y} = \frac{n}{2\pi r \Delta z} (\delta_{L} \sin\beta + \delta_{D} \cos\beta) \cos\theta$$
(8)

$$f_z = -\frac{n}{2\pi r\Delta z} (\delta_D \cos\beta + \delta_L \sin\beta)$$
(9)

The actuator disc model is incorporated into the CFD model by means of a user-defined subroutine where the source terms are calculated using equations 4 to 9, based on velocities extracted from the CFD-simulation. These velocities are extracted from the cell centres of the upstream and downstream reference discs (see Figure 9).

Computational Domain

A CFD simulation of the axial flow fan in the BS 848 test facility, using the actuator disc method, was performed using ANSYS[®] FLUENT[®] version 6.2.6 (which was later updated to version 12.1). A sketch detailing the lay-out of the computational domain and the mesh surrounding the actuator disc is shown in Figure 9. The domain was modelled to represent the test facility shown in Figure 3. The inlet boundary was specified to be a mass flow inlet, while the outlet boundary was specified to be a static 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 x fan diameter and a length of 8 x fan diameter. The model initially consisted of 650000 hexahedral cell volumes. A refined mesh with 1000000 volumes showed only a 0.5% difference in the simulated value for fan static pressure at the design point of the fan.

The CFD simulation assumed steady state incompressible flow. The realizable k- ε turbulence model and Quick interpolation scheme were used for all simulations [11].



Figure 9. Lay-out of CFD model (van der Spuy et al. [11]).

Results showing the fan static pressure curve obtained from a CFD simulation using the actuator disc method is shown in Figure 14.

At low flow rates the simulation using the actuator disc method shows a central area of high flow resistance (increased drag and lower lift for increased angle of attack) around the hub of the fan. This area of high resistance forces the incoming flow to deflect outwards towards the outer radius of the fan (see Figure 10).



Figure 10. Centre plane vector plot obtained using the actuator disc method at low flow rate.

PROPOSED IMPROVEMENT OF ACTUATOR DISC METHOD

Background

The actuator disc method described earlier assumes that the flow enters and exits the fan blade rotation plane in "separate" annular rings and that there is no radial flow between the annular rings. It has however been noted, in particular by Meyer and Kröger [9], that this assumption is not always valid. For a general axial flow fan inlet condition with zero swirl (C_{θ} = 0) and a uniform axial velocity distribution ($C_z = \text{constant}$), the inner section (close to the hub) of the blade would be particularly sensitive to a change in the value of axial velocity, due to its lower circumferential velocity. At low flow rates the inner section of an axial flow fan would experience a decrease in relative velocity and an increase in angle of attack. As shown in Figure 10, radial flow would occur through the fan at low flow rates, even though the actuator disc method does not take it into account when calculating the momentum source terms shown in equation 4 and 5.

The effect of radial flow in the rotor on the measured performance of an axial flow device was first described by Himmelskamp [15]. Himmelskamp measured the pressure distribution around a rotating fan blade profile at various radial locations along the length of the blade (see Table 1).

Table 1. Himmelskamp fan properties [15].

Shroud diameter	500 mm
Hub diameter	100 mm
Nr. of blades	2
Blade profile	Göttingen 625

Himmelskamp also measured the absolute velocity vectors at the leading and trailing edges of the rotor and from these measurements he was able to publish data showing the apparent lift and drag coefficients for the rotating blade elements at various radial locations along the length of a fan blade. He found that, at low flow rates, the occurrence of stall in the fan rotor at the inner blade radii was delayed and the actual lift coefficients measured in these locations reached values as high as 3.2 (see Figure 11). The lift coefficients were obtained by measuring the chord-wise pressure distribution at various radial locations along the fan blade, as well as the velocity distribution upstream and downstream of the fan rotor at the same radial locations.

The B-fan used in this investigation differs entirely from the fan tested by Himmelskamp and no direct comparison between the two fans can be made. Reference to the results of Himmelskamp is therefore made solely to illustrate the effect of 3-dimensional flow over the fan blade.



Figure 11. Measured lift coefficients for a rotating fan blade (re-drawn from Himmelskamp [15]).

A number of authors (Snel et al. [16], Corrigan and Schillings [16] and Gur and Rosen [18]) have attributed the augmentation of the stalled rotor's performance to the existence of radial flow in the low velocity region on the suction side blade surface (the radial flow is due to centrifugal forces). This radial flow is subjected to a coriolis force. The direction of the coriolis force is predetermined by the equation:

$$f_{ci} = 2\,\omega_i \times u_i \tag{10}$$

Under normal circumstances, where the axis of rotation of the fan coincides with the axial flow direction, the coriolis force would act in the tangential direction. As shown in Figure 12, depending on the orientation of the blade chord, the coriolis force would counteract the tendency of the flow to separate from the blade suction surface.



Figure 12. Graphical illustration of stall delay effect.

The increased blade coefficient properties observed by Himmelskamp should however not be attributed to stall delay only. An axial flow fan that experiences radial flow can be regarded as a mixed flow fan, to a certain extent. Lewis [19] identifies two types of forces that are present in mixed flow fans, namely aerodynamic forces and Coriolis forces. He concludes that the aerodynamic forces can be defined as the forces that result in a change of angular momentum of the flow, while the Coriolis force is exerted on the flow based on its radial velocity component. There are therefore three effects that need to be considered when radial flow occurs in an axial flow fan, namely the Coriolis effect, the cross flow effect (flow running diagonally over a blade increases the chord length of the blade) and stall delay. It can therefore be assumed that the measurements of Himmelskamp, although traditionally linked to the effect of stall delay, actually included all three these mechanisms.

The experiments of Himmelskamp were performed at a minimum flow coefficient of 0.124 (where flow coefficient is defined as average axial flow velocity divided by the circumferential blade velocity at the tip). At the design point of the B-fan, the flow coefficient of the fan is 0.142. It would therefore be of interest to see whether the same degree of lift coefficient increase that was observed by Himmelskamp can be observed for the B-fan. The results from the 3-dimensional CFD analysis of the B-fan at its design point of le Roux were processed to obtain values for the lift coefficients versus the effective angles of attack of the flow along the length of the fan blade (see Figure 13). These values were compared to lift coefficient values for the 2-dimensional LS10413 airfoil profiles published by McGhee et al. [20]. The processing of the CFD results was performed according to Himmelskamp's description of the processing of his experimental results.

From Figure 13 it can be seen that the B-fan does not experience the same degree of lift increase as experienced by the fan of Himmelskamp, at similar flow coefficients.



Figure 13. Post processed lift coefficients at different radius ratios for rotating B-fan blade at 16 m^3/s , compared to published airfoil profile data.

Description of augmentation model

Gur and Rosen [18] simulated an aircraft propeller using the actuator disc method. Gur and Rosen points out that the value of the effective 3-dimensional lift coefficient (that includes rotational effects) of a propeller blade profile at low advance ratios would be between the 2-dimensional coefficient (measured in a wind tunnel) and the inviscid lift coefficient (which would be an extension of the linear section of the lift coefficient curve). They proposed a method of increasing the lift coefficients of the blade profiles used in their simulation based on the radius ratio at which the blade profile was located. At small radius ratios close to the hub (where the augmentation would be the most significant according to Figure 8) the values for the lift coefficients were increased by simply extending the linear section of the lift coefficient vs. angle of attack curve. At large radius ratios close to the blade tip, no augmentation would exist. At radius ratios between these two extremes, Gur and Rosen increased the lift coefficients using a hyperbolic function.

The model proposed as part of this research is based on the model of Gur and Rosen but is much simpler. In this model, the lift coefficient values of the blade profiles are increased by extending the linear section of the lift coefficient vs. angle of attack curve irrespective of their radius ratios. The motivation for doing this is based on the assumption that the angles of attack at the tip of a rotating blade would always be within the original linear range of the lift coefficient versus angle of attack curve. This is primarily due to the high circumferential speed of the blade tip, as well as the fact that the flow through the rotor is displaced towards the blade tip as the volume flow rate through the fan decreases.

However, by doing this, it was found that the fan static pressure values at low flow rates were over-predicted. At low flow rates the angles of attack of the rotor blade of an axial flow machine close to the hub become quite large (primarily due to the lower rotational speed at smaller radii). Consequently, when using the extended lift coefficient curve, this gave rise to high values for lift coefficient and fan static pressure rise. It was therefore decided to specify a limiting radius ratio below which the extended coefficient values would not be used and the model uses the original 2-dimensional profile values used in the actuator disc method. Although this may seem to counter the results of Himmelskamp, it should be noted that Himmelskamp only conducted his experiments at decreasing flow rates up to a point where reversed flow was observed in the outlet velocity profile at the hub of the experimental fan.

An appropriate value for the limiting radius ratio was determined by iteratively comparing the simulation results to the experimental results and in this instance it was found that a value of 0.45 gave satisfactory results. This proposed new model is referred to as the extended actuator disk method.

Additionally, the value of the 2-dimensional drag coefficients (used in the actuator disc method) is adjusted proportionally according to the increase in the 3-dimensional lift coefficient, as proposed by Gur and Rosen [18]:

$$C_{D3D} = C_{L3D} \times (C_{D2D} / C_{L2D})$$
(11)

Results

To confirm the implementation of the extended actuator disc method the computational analysis as described earlier was repeated. The lay-out of the CFD model and the boundary conditions used, are identical to those described earlier in this document. The results of both the actuator disc and extended actuator disc CFD simulations, showing the static pressure vs. volume flow rate curve, are shown in Figure 14. The CFD results are compared to the experimental results of le Roux [13]. The agreement is shown to be excellent.



Figure 14. Results of actuator disc method and extended actuator disc method CFD for single fan (fan static pressure).

To investigate the augmentation of the lift coefficients further, the CFD results from the analysis using the extended actuator disc model were used to calculate the lift coefficient versus angle of attack values at 16 m³/s and 6 m³/s and these values were compared to the values shown previously in Figure 13 (see Figure 15).



Figure 15. Post processed lift coefficients at different radius ratios for B-fan at 16 m³/s and 6 m³/s, compared to published airfoil profile data.

Figure 15 shows that the majority of the lift coefficient values used by the extended actuator disc model correlate with those predicted by the full 3-D CFD analysis at 16 m³/s. It also shows that the extended lift coefficient values have very little effect on the results simulated using this model at 16 m³/s. The effect of limiting the radius ratio is only evident at the last point (at 41° angle of attack) shown for the analysis at 6 m³/s.

CONCLUSION

The ability of the actuator disc method to model the performance of an axial flow fan at design flow rates has been proven by a number of authors [2-4, 8-9, 11]. However, none of these simulations reported any simulation results at low flow rates, due to the recognised inability of the actuator disc method to take the effect of radial flow into account.

The proposed empirical extended actuator disc model takes the effect of radial flow through the fan rotor into account by increasing the 2-dimensional lift coefficients (to become 3-dimensional lift coefficients) and then increasing the drag coefficient in proportion to the increase in lift coefficient. Figure 14 shows experimental test results for a single fan test (following the BS 848 Standards) being compared to CFD results obtained using the actuator disc method and to CFD results obtained using the extended actuator disc model. The actuator disc method correlates well with the fan static pressure test results at medium to high flow rates but completely under predicts the test results at low flow rates. Using the proposed extended actuator disc model, fan static pressure results are obtained that correlate well with the test results at all flow rates.

Implementation of the new model into the actuator disc method has shown that it is possible to predict fan operation at low flow rates. However, the presented model is purely empirical and the ability of the presented model to predict the performance of other fan designs should also be investigated. As pointed out earlier, the presented model has been specifically applied to fan test data for low flow rates in a standard fan test facility. A next step would also be to evaluate the ability of the model to predict low flow rates through a fan being subjected to distorted inlet conditions.

NOMENCLATURE

- c chord length, m.
- C_D drag coefficient.
- C_L lift coefficient.
- d_f fan diameter, m.
- D_H hydraulic diameter, m.
- f force, N.
- *n* number of blades.
- N speed, rpm.
- *P* power, W.
- *p* pressure, Pa.
- Q volume flow rate, m³/s.
- *T* torque, Nm.
- r radius, m.
- *u* absolute velocity, m/s.
- *w* relative velocity, m/s.
- *x* coordinate.
- *z* coordinate in axial flow direction.
- β blade setting angle, degrees.
- Δ delta.
- δ delta.
- η efficiency.
- ω rotational speed, rad/s.
- ρ density, kg/m³.
- θ angular position in fan plane of rotation, degrees.

Subscripts

- D drag.
- d dynamic.
- FS fan static.
- L lift.
- sett settling chamber.
- *shaft* fan shaft.
- *x*,*y*,*z* coordinate system indices.
- \propto infinity.
- 2D 2-dimensional
- 3D 3-dimensional

ACKNOWLEDGMENTS

The work detailed in this document was made possible through funding received from ESKOM TESP, NRF Thuthuka and Stellenbosch University. All CFD simulations were performed on the Stellenbosch University high performance computer.

REFERENCES

- 1. Kröger, D.G., Air-cooled Heat Exchangers and Cooling Towers. PennWell Corporation, Tulsa, 2004.
- 2. Goldschagg, H.B., Vogt, F., du Toit, C.G., Thiart, G.D. and Kröger, D.G., Air-cooled steam condenser performance in the presence of crosswinds, Proceedings: Cooling Tower

Technology Conference, Electrical Power Research Institute (EPRI TR-108483), Palo Alto, California, 1997.

- 3. Meyer, C.J., Numerical investigation of the effect of inlet flow distortions on forced draught air-cooled heat exchanger performance, *Applied Thermal Engineering*, 2005, 25, 1634-1649
- 4. Bredell J.R., Kröger D.G. and Thiart G.D., Numerical investigation of fan performance in a forced draft air-cooled steam condenser, *Applied Thermal Engineering*, 2006, 26, 846-852.
- Hotchkiss, P.J., Meyer, C.J., von Backström, T.W., Numerical Investigation into the effect of cross-flow on the performance of axial flow fans in forced draught air-cooled heat exchangers, *Applied Thermal Engineering*, 26, 2006.
- 6. Monroe, R.C., Improving cooling tower fan system efficiencies, Combustion Magazine, 1979, 50 (11).
- van Staden, M.P., An integrated approach to transient simulation of large air-cooled condensers using computational fluid dynamics, PhD Thesis, Department of Mechanical Engineering, Rand Afrikaans University, South Africa, 2000.
- 8. Thiart, G.D., and von Backström, T.W., Numerical simulation of the flow field near an axial fan operating under distorted inflow conditions, Journal of Wind Engineering and Industrial Aerodynamics, 1993, 45, 189-214.
- 9. Meyer, C.J. and Kröger, D.G., Numerical simulation of the flow field in the vicinity of an axial flow fan, *International Journal for Numerical Methods in Fluids*, 2001, 36, 947-969.
- 10. Van Rooyen, J.A., Kröger, D.G., Performance trends of an air-cooled steam condenser under windy conditions, *Journal of Engineering for Gas Turbines and Power*, 130, 2008.

- 11. Van der Spuy, S.J., von Backström, T.W., Kröger, D.G., An evaluation of simplified methods for modelling the performance of axial flow fan arrays, *R&D Journal*, 25, 2010.
- 12. Bruneau, P.R.P., The design of a single rotor axial flow fan for a cooling tower application, M.Eng. thesis, Department of Mechanical Engineering, University of Stellenbosch, South Africa, 1994.
- 13. le Roux, F.N., The CFD simulation of an axial flow, M.Sc. Eng. thesis, Department of Mechanical and Mechatronic Engineering, University of Stellenbosch, South Africa, 2010.
- 14. British Standards Institution. Part 1: Methods for testing performance, fans for general purposes, BS 848. 1980.
- 15. Himmelskamp, H., Profile investigations on a rotating airscrew, PhD Thesis, Göttingen, 1945 (MAP Volkenrode, Reports and Translations, No. 832, 1947).
- 16. Snel, H., Houwink, R., Bosschers, J., Sectional prediction of lift coefficients on rotating wind turbine blades in stall, ECN-C-93-052, Petten, 1994.
- 17. Corrigan, J.J., Schillings, J.J., Empirical model for stall delay due to rotation, American Helicopter Society Aeromechanics Specialists Conference, San Francisco, 1994.
- 18. Gur, O., Rosen, A., Propeller performance at low advance ratio, Journal of Aircraft, 42, no. 2, 2005.
- 19. Lewis, R.I., Turbomachinery performance analysis, Arnold, Londen, Great Britain, 1996.
- McGee, R.J., Beasley, W.D., Somers, D.M., Low-speed aerodynamic characteristics of a 13-percent-thick airfoil section designed for general aviation applications, NASA Technical Memorandum X-72697, Langley Research Centre, Hampton, Virginia, 1977.