CFD Simulations of Aerodynamic Performance of Low-pressure Axial Fans with Small Hub-to-tip Diameter Ratio

Jie Wang*, Niels P. Kruyt†

Department of Mechanical Engineering, University of Twente,
P.O. Box 217, 7500 AE Enschede, the Netherlands

Abstract

Rotor-only ducted axial fans with small hub-to-tip diameter ratio are widely used in many branches of industry, especially for cooling and ventilation purposes. For such fans, extensive regions of backflow are present downstream of the fan near the hub. Only few Computational Fluid Dynamics (CFD) studies for such fans have been reported in the scientific literature. In order to develop guidelines for obtaining accurate CFD predictions for such fans, validation simulations of a fan with small hub-to-tip diameter ratio have been performed by comparing experimental and computed aerodynamic performance characteristics.

These guidelines pay special attention to the trailing edge shape, presence of non-aerodynamically shaped blade sections, tip gap and employed turbulence model. The results for the fan studied here show that the actual (rounded) trailing edge is necessary; the main blade (without non-aerodynamically shaped blade sections) well represents the aerodynamic performance of the whole fan blade; it is recommended not to take the tip gap into consideration due to the existence of significant flow separation; the use of the Spalart-Allmaras turbulence model is advised for giving better agreement with measurements.

Keywords: axial fans; RANS CFD simulations; hub-to-tip diameter ratio; trailing edge geometry; non-aerodynamically shaped blade section; tip gap.
1. Introduction

In industrial applications, ducted low-pressure axial flow fans are widely used for cooling and ventilation purposes. For low-pressure axial fans, downstream diffuser blades are often not employed [1], due to the limited aerodynamic benefits and the additional costs and required space.

The operational parameters of an axial fan are its rotational speed $\Omega$ (in rad/s) and the volumetric flow rate $Q$. The aerodynamic performance is characterised by the (total-to-static) fan static pressure rise $p_{fs}$, the (input) shaft power $P_{\text{shaft}}$ required to run the machine and the total-to-static efficiency $\eta_{ts}$. The fan static pressure $p_{fs}$ and the total-to-static efficiency $\eta_{ts}$ are defined by

$$p_{fs} = p_2 - p_{01} \quad \eta_{ts} = \frac{Q \cdot p_{fs}}{P_{\text{shaft}}},$$

where $p_{01}$ is the average total (or stagnation) pressure at inlet and $p_2$ is the average static pressure at outlet of the fan.

Dimensionless performance parameters (that are based on similarity of flow conditions, with the additional assumptions that the influence of the Mach number and the Reynolds number are of secondary importance; see for example [2]) are

$$\varphi = \frac{Q}{\frac{1}{5} \Omega D_{\text{fan}}^3} \quad \psi = \frac{p_{fs}}{\frac{1}{5} \rho \Omega^2 D_{\text{fan}}^2} \quad \lambda = \frac{P_{\text{shaft}}}{\frac{1}{32} \rho \Omega^3 D_{\text{fan}}^5}.$$ (2)

Here $\varphi$ is the flow coefficient, $\psi$ is the pressure coefficient and $\lambda$ is the power coefficient. The density of the gas is denoted by $\rho$ and $D_{\text{fan}}$ is the fan outer diameter (i.e. the tip diameter of the fan blades). The Reynolds and Mach numbers (based on the tip radius and tip speed) are defined by

$$\text{Re} = \frac{\Omega D_{\text{fan}}^2}{4 \nu} \quad \text{Ma} = \frac{\Omega D_{\text{fan}}}{2 a},$$ (3)

where $\nu$ and $a$ are the kinematic viscosity and the speed of sound of the gas, respectively.

The hub-to-tip diameter ratio $\kappa$ is defined by

$$\kappa = \frac{D_{\text{hub}}}{D_{\text{fan}}},$$ (4)

where $D_{\text{hub}}$ is the hub diameter. In Figure 1 examples are shown of fans with a large and a small hub-to-tip ratio $\kappa$. The focus of the current study is on an axial fan with small hub-to-tip ratio $\kappa$, shown in Figure 1 (right). In [1, 3] (their section 9.2 and section 3.5.2, respectively) relations are given between the hub-to-tip ratio $\kappa$ and the flow coefficient $\varphi$ such that regions with backflow (or ”dead flow regions”) may occur. For the fan in Figure 1 (right) backflow regions are expected.

Classical design methods for axial flow fans rely on empirical methods and simplified analyses, in the form of dimensionless performance parameters, one-dimensional flow and cascade analyses, with some prescribed vortex distribution along the spanwise direction [1, 2, 4–7]. This classical approach is of limited reliability due to the assumptions involved, and therefore time-consuming and expensive,
as it may require multiple experimental verifications and design modifications. A historical overview of design methods is provided in [8].

Complementary to the classical design methods, it has become possible (due to the rapid development of computer hardware and simulation software) to employ Computational Fluid Dynamics (CFD for short) simulations. Based on an enhanced description of the flow physics, such simulations make it possible to analyse in detail the performance and the flow fields for low-pressure axial fans. The detailed information on the flow thus obtained gives additional understanding and insights that allow for a noticeable reduction in experimental testing and in associated costs.

To compute turbulent flows with CFD simulations, different approaches can be followed, in order of increasing computational costs: Reynolds Averaged Navier-Stokes simulation (RANS for short), Large Eddy Simulation [9, 10] (for fans), Direct Numerical Simulation [11]. RANS simulations require some turbulence model (such as $k-\epsilon$, $k-\omega$, Shear Stress Transport and Spalart-Allmaras models; SST and SA for short, respectively; see [11] for details) to describe the turbulent Reynolds stresses. An overview of turbulence models for turbomachinery applications is given in [12, 13]. Here RANS simulations are considered, with their benefit of limited computational costs.

To be able to use CFD simulations to support the design of low-pressure axial fans, it is important to establish their accuracy and to formulate guidelines for such simulations. Here the accuracy of such simulations is evaluated by comparison between measured and computed aerodynamic performance characteristics. The number of CFD studies dealing with the aerodynamic performance of fans with small to medium hub-to-tip ratio $\kappa$ is limited (see for example [14–18]). These studies are discussed in more detail in the following, in the order of decreasing hub-to-tip ratio $\kappa$. The dimensionless performance parameters for the fans considered in these studies are summarised in Table 1. Note that these studies often focus on more detailed flow phenomena and qualitative effects of blade geometry on various performance parameters than the overall aerodynamic fan performance parameters considered here. Only the current fan is expected to have a backflow region, according to the criterion given in [1, 3]. For fans with small $\kappa$, design approaches based on cascade analyses break down due to the significant radial redistribution of fluid. Design methods for fans with small hub-to-tip ratio $\kappa$ are presented in [19, 20].

For fans with a fairly large hub-to-tip ratio $\kappa = 0.6$ (and including diffuser blades) the influence of forward skewed blades with a prescribed vortex distribution on the aerodynamic performance of axial fans has been investigated in [15, 21] by experiments and CFD simulations (using the $k-\epsilon$ turbulence model). The CFD simulations account for the tip gap and the mesh independence of the CFD results has been checked. Besides measured velocities (using hot-wire anemometry) at the design point, the aerodynamic performance has also been estimated. The agreement between measured and computed performance characteristics is labelled fair (10% deviation at the design point) due to limitations in the experimental test set-up [15].
For a fan with medium hub-to-tip ratio $\kappa = 0.45$ the influence of the size of tip clearance $s$ on the aerodynamic and the aeroacoustic performance has been studied experimentally and using RANS CFD simulations (using the SST turbulence model) in [17, 22], for three tip clearance ratios $s/D_{\text{fan}} = 0.1\%$, 0.5\% and 1.0\% (where $s$ is the tip clearance). The comparison of measured and computed aerodynamic performance (in terms of pressure coefficient $\psi$, total-to-static efficiency $\eta_{ts}$ and power coefficient $\lambda$ as function of the flow coefficient $\phi$) is considered good with 2\% deviation at the design flow coefficient for the fans with 0.1\% and 0.5\% tip clearance ratio; for the fan with 1.0\% tip clearance ratio, larger deviations are found for small flow coefficients. Note that the mesh independence of the CFD results has not checked. Since complete geometrical information is available for these fans [22], these fans have also been considered here.

A design methodology for skewed blades is presented in [14]. With a fan with medium hub-to-tip ratio $\kappa = 0.4$, experiments and RANS CFD simulations have been performed to verify this methodology. The computational grids used are relatively coarse for present-day standards. The small tip gap has not been accounted for in the CFD-simulations that use the $k-\epsilon$ turbulence model. The flow fields that have been measured (using hot-film probes) at some distance downstream of the fan show that at the design flow rate the axial velocity is numerically well predicted by the CFD simulations; the measured radial and circumferential velocity profiles show qualitative agreement. No direct comparison is presented of overall measured and computed aerodynamic performance characteristics.

Design guidelines for the blade sweep and the vortex distribution are formulated in [18]. In this study a comparison is also made of experimental and computed aerodynamic performance characteristics (using the $k-\omega$ turbulence model). The grid independence is established qualitatively. Good qualitative agreement is observed. For the three considered fans, good quantitative agreement is observed for either pressure coefficient or efficiency, but not for both simultaneously.

Tip leakage flows in low-pressure axial fans, with hub-to-tip ratio $\kappa = 0.35$ and with circumferentially-skewed blades, are investigated experimentally and using CFD simulations (employing the Spalart-Allmaras turbulence model) in [16]. The grid (in)dependence has been investigated, showing some dependence of the spanwise pressure rise on the employed grid size. The predicted pressure coefficient shows good qualitative agreement with the measured pressure coefficient, with 5\% deviation at the design point. Unfortunately, results are neither shown for the total-to-static efficiency $\eta_{ts}$ nor for the power coefficient $\lambda$.

The influence of the tip gap on the aerodynamic performance has been studied experimentally and using CFD simulations for a fan with $\kappa = 0.29$ for flow rates near the design point in [23], for tip clearance ratios $s/D_{\text{fan}} = 0.31\%$, 0.26\% and 0.39\%. In the CFD simulations the realisable $k-\epsilon$ turbulence model has been employed. The authors state that the experimental and CFD predictions correlate well for the pressure coefficient and reasonably well for the total-to-static efficiency.

The focus of the current study is on a fan with very small hub-to-tip ratio $\kappa = 0.14$ (much
smaller than for the studies discussed above and listed in Table 1) for which a region of backflow is expected [1, 3]. For the considered (Howden) fan a complete description of the fan blade geometry is available, in combination with the tip gap size. Part of the blade near the hub has a shape that is not aerodynamically shaped.

Three-dimensional RANS CFD simulations have been performed with Numeca Fine/Turbo software [24], where the grid independence has been thoroughly investigated. Computational predictions of the aerodynamic performance have been compared against high-quality measurement data. In this comparison both the dimensionless fan pressure rise $\psi$ and the total-to-static efficiency $\eta_{ts}$ are considered.

The ultimate aim of this study is to develop a CFD simulation strategy with which accurate predictions can be obtained for low-pressure axial fans with small hub-to-tip ratio $\kappa$. Specific objectives of this study with respect to the aerodynamic performance of low-pressure axial fans with small hub-to-tip ratio $\kappa$ are to:

- Assess the accuracy of RANS-based CFD simulations;
- Study the influence of the shape of the trailing edge blade sections (sharp vs. rounded);
- Study the influence of the presence of non-aerodynamically shaped parts of the blades;
- Study the influence of the tip gap size;
- Study the influence of the employed turbulence model.

Note that the current study only focusses on the aerodynamic performance for flow rates far from stall conditions. The more complex aeroacoustic performance is not considered here (see for example [16, 17, 22, 25]). Methods for extending the stall characteristics are described in [26, 27].

The outline of this study is as follows. In Section 2 the fan geometry and the employed test facility for measuring the aerodynamic performance are described. The CFD simulation method is detailed in Section 3. The CFD results are analysed in Section 4. Finally, the findings of this study are discussed in Section 5.

2. Fan Geometry and Measurement Setup

The geometry of the Howden fan with small hub-to-tip ratio studied here is given in detail in Section 2.1. The measurement setup, with which the high-quality aerodynamic performance characteristics have been obtained, is described in Section 2.2.

2.1. Fan Geometry

The structure of the fan and details of the fan blades are shown in Figure 2. The casing, hub and fan diameters, $D_{casing}$, $D_{hub}$, and $D_{fan}$ respectively, are indicated in Figure 2a. Each of the six
identical blades is divided into two parts (for the geometrical description), the “main blade” and the “non-airfoil part”, as indicated in Figure 2b; the starting location of the main blade is at $0.18R_{\text{fan}}$.

For the main blade, the airfoil sections correspond to the Wortmann profile [28], with chord length of 0.15m, see also Figure 2c,d. The sections are slightly twisted by an angle of $4^\circ$ between the sections at positions $A$ and $B$, indicated in Figure 2b, in anticlockwise direction. The centre of rotation for the sections (to account for the twist) is located at the point of maximum thickness, represented by position $O$ in Figure 2c. The blade is straight from the section at position $B$ to the blade tip. The stagger angles (as defined in Figure 2d) for this fan can be adjusted from $5^\circ$ to $30^\circ$; here the considered stagger angle equals $15^\circ$.

The main geometrical and design parameters of this fan are summarised in Table 2.

2.2. Measurement Setup

In order to evaluate the accuracy of the CFD simulations, the aerodynamic performance of the fan has been measured in the Howden Test Facility, shown schematically in Figure 3. The test rig is built according to the international standards AMCA 210 and ISO5801 [29]. The tested fan part, indicated by the blue dotted rectangle on the right of Figure 3, corresponds to the structure of the fan shown in Figure 2a and is also referred to as the computational domain for the CFD simulations (see Section 3). The inner diameter of the casing, $D_{\text{casing}}$, of the test part is 1.845m. The variable inlet valve and the booster fan help to adjust and overcome the system resistance, so the whole aerodynamic performance curve of the fan is obtained, from free delivery (zero pressure rise) to shut-off (zero flow rate).

The volumetric flow rate $Q$ through the fan is determined from the static pressure drop over the nozzle measured by a digital differential pressure transducer. An array of three Pitot tubes assembled at the inlet of tested fan is used to measure the total pressure (relative to the atmospheric pressure $p_2$), which gives the Fan Static Pressure $p_{\text{fs}}$ in Eq.(1). The fan shaft power $P_{\text{shaft}}$ is determined from the shaft torque and fan rotational speed, which are measured by a torque meter and an induction sensor, respectively. The gas (air) density is determined by measuring the ambient temperature, relative humidity and atmospheric pressure. The measured variables, corresponding sensors and their accuracy are given in Table 6 in Appendix A.

The measurements start near free-delivery conditions. Subsequently, the system resistance is increased in small steps until the fan reaches stall condition. One or two measurements above the stall condition are made in order to construct the fan curve. The system resistance is then lowered in small steps until the fan reaches free-delivery again. Hence, each performance condition is measured twice. These measurement procedures guarantee the high quality of the measured aerodynamic performance, so meaningful comparisons between measurements and CFD prediction are possible. Based on all measured and derived variables, the fan performance curves represented by the dimensionless coefficients $\varphi$, $\psi$, $\lambda$ and $\eta_{ts}$ (defined in Eqs.(1) and (2)) are obtained.
3. CFD Simulation Method

For the low-pressure axial fan that forms the focus of this paper, the Mach number is 0.14 and the Reynolds number is $4.9 \times 10^5$ (both dimensionless numbers are defined in Eq.(3)), so the flows can be regarded as being incompressible and turbulent. The medium is considered as air (real gas; ideal gas with specific heat dependent on temperature).

The influence on the predicted aerodynamic performance characteristics of the items listed in Table 3 are specifically investigated here. For each item, two possibilities are considered that are labelled as “baseline” and “alternative” in this Table. For this baseline case, a grid convergence study is performed whose results are reported Section 3.3.

3.1. Flow Equations, Computational Domain and Boundary Conditions

The (Reynolds-averaged) velocity and pressure field in low-pressure axial fans are described by the continuity equation and the RANS momentum equations

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial (\bar{\rho} \bar{u}_i)}{\partial x_i} = \frac{1}{\bar{\rho}} \frac{\partial \bar{p}}{\partial x_i} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_j \partial x_j} - \frac{\partial (u_i' u_j')}{\partial x_j} + g_i.$$  

(5)

Here $\bar{u}_i$ is the Reynolds-averaged (absolute) velocity vector, $x_i$ denotes the $i$-th spatial coordinate, $\bar{p}$ is the Reynolds-averaged pressure, $u_i'$ is the velocity fluctuation and $-\rho u_i' u_j'$ is the Reynolds stress tensor and $\rho g_i$ denotes a body force. The Einstein summation convention has been employed, implying a summation over repeated subscripts.

The Reynolds stress tensor is modelled by the Boussinesq eddy-viscosity hypothesis

$$-\bar{u}_i' \bar{u}_j' = \nu_t \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} k \delta_{ij} \quad \text{with} \quad k = \frac{1}{2} \bar{u}_i' \bar{u}_i'.$$  

(6)

Here $\nu_t$ is the turbulent kinematic viscosity, $\delta_{ij}$ is the Kronecker symbol and $k$ is the turbulent kinetic energy.

Many models for the turbulent viscosity $\nu_t$ and the turbulent kinetic energy $k$ have been proposed [11]. Here the one-equation Spalart-Allmaras turbulence model [30] and the Shear Stress Transport turbulence model [31] are considered. The Spalart-Allmaras turbulence model is a typical one-equation model originally formulated for aerodynamic applications and subsequently successfully applied in axial fan studies [16]. The two-equation SST model combines advantages of the $k - \epsilon$ and the $k - \omega$ turbulence models. The SST model has also been applied successfully in axial fan investigations [17].

The investigated fan is a rotor-only fan (i.e. no diffuser blades are present) with six identical blades. Considering the circumferential symmetry of the casing and the symmetry of the blades, it is sufficient to consider only a single blade passage as computational domain when the stable operating range is considered. The test facility is a category A inlet chamber test set up, meaning that there is no significant ducting at either the inlet or outlet of the fan. In order to reduce the influence of imposed

FE-19-1898; N.P. Kruyt 7
boundary conditions at inlet and outlet on the CFD results computational domain is extended here by one and two casing diameters upstream and downstream from the blades, respectively. For ease in grid generation (with good quality), the hub is extended to the outlet. This gives a small blockage in the meridional plane, of 1.84% and 3.38% blockage for the blade with and without non-airfoil part, respectively. The computational domain is shown in Figure 4.

Due to the absence of diffuser blades and the circumferentially uniform inflow conditions, the flow is considered to be steady in the frame of reference that is rotating with the (identical) fan blades.

In order to make the CFD simulation results comparable to measurements, the boundary conditions should represent actual flow conditions in the experiments as closely as possible. A settling screen is present upstream from the fan inlet in the experiments (see Figure 3); with a well-designed “bell mouth”, variations in inlet velocity are expected to be small (see for instance the experimental results in [17]). Hence, the inlet flow is assumed to be uniform and without no pre-swirl and with low turbulence level. Therefore, turbulence quantities such as the turbulent viscosity \( \nu_t \) in the Spalart-Allmaras turbulence model and the turbulent kinetic energy \( k \) and the dissipation rate \( \epsilon \) in the SST turbulence model are estimated with low-turbulence level relations. For the Spalart-Allmaras turbulence model for internal flows (e.g. turbomachinery), it is recommended that \( \nu_t/\nu \) is in the range 1 – 5, from low to high turbulence level [24]. For the SST turbulence model, the turbulence intensity \( Tu \) for internal flows is recommended to be about 5% and estimates for \( k \) and \( \epsilon \) can be obtained from

\[
k = \frac{3}{2} (Tu \cdot U_{ref})^2 \quad \epsilon = C_\mu \frac{\mu}{\nu_t} \frac{k^2}{\mu}.
\]

Here \( C_\mu = 0.09 \) is a model constant, a typical value for \( \mu_t/\mu = 50 \) (for internal flows) and the characteristic velocity \( U_{ref} \) here corresponds to the streamwise velocity [24].

As measured directly during the measurements, the total pressure and the volumetric flow rate are prescribed at the inlet and outlet, respectively. Note that the latter boundary condition does not imply the the velocity is uniformly distributed. At the outlet, the pressure distribution conforms to the radial equilibrium condition (see for example [2]), \( \frac{\partial p}{\partial r} = \rho \frac{U^2}{r} \), where the value of the static pressure \( p \) is fixed at some radius \( r \).

At the blade and hub surfaces, the relative flow velocity vector equals zero due to the no-slip condition. The hub surfaces have the same rotational speed as the blades. At the outer casing, the absolute flow velocity vector equals zero. Due to the assumed symmetry in the circumferential direction, periodic boundary conditions are applied in circumferential direction at the corresponding surfaces (see also Figure 4b).

3.2. Grids, Discretisation and Solution of Equations

Three different multi-block structured grids (indicated by “coarse”, “medium and “fine”) have been generated with AutoGrid (part of the NUMECA Fine/Turbo CFD environment) for the grid
independence study in Section 3.3. The grid quality is based on considerations of skewness angle, aspect ratio and expansion ratio. The skewness angle should be as close to $90^\circ$ as possible, while the aspect ratio (possible range: $1 - 50000$) and the expansion ratio (possible range: $1 - 100$) should be as small as possible. For the “coarse” grid, the minimum skewness angle is $35.9^\circ$, the aspect ratio of more than $98\%$ cells is smaller than $1000$ and the maximum expansion ratio is $2.24$. Hence, the grid satisfies the quality criteria described in [32]. Information on these three grids is given in Table 4. All employed grids satisfy the quality criteria described above.

A cell-centred second-order finite volume approach is applied for the spatial discretization of the governing equations in conservative form. In the present study, the steady RANS flow equations are solved with implicit residual smoothing. Local time-stepping and three-level multigrid techniques are used to accelerate convergence of the solution of the discretised equations. Merkle preconditioning is applied in the compressible-flow code to provide fast convergence for low Mach numbers [33, 34].

The discretised nonlinear equations have been solved iteratively. A typical convergence history curve of the global residual (i.e. the root mean average squared of the imbalance of the discretised equations for the cells) is shown in Figure 5, to assess the convergence process. A reduction of the global residual by three orders of magnitude is considered to indicate good convergence [24]. Here a much stricter convergence criterion, a reduction by six orders of magnitudes, is employed.

Based on the computed solutions, it has been checked that the near-wall grid resolution is sufficiently fine by determining the dimensionless wall distance $y^+ = U_\tau y/\nu$ (where $y$ is the distance of the first grid point away from the wall and $U_\tau$ is the friction velocity). For all simulations, the maximum value $y_{\text{max}}^+ < 3.3$ and the averaged value $y_{\text{avg}}^+ < 0.3$. Hence, the first grid point from the wall is located in the viscous sublayer.

### 3.3. Grid Independence Study

A grid independence study has been performed, in order to assess the numerical accuracy of the CFD simulations and to formulate a guideline for mesh generation with good quality and fairly low computation cost for the considered low-pressure axial fans. According to the guidelines for assessing the accuracy of CFD solutions in [35], three sets of grids need to be considered, with characteristic grid size $h$ (for each of the three grids) defined by

$$h = \left[ \frac{1}{N} \sum_{i=1}^{N} \Delta V_i \right]^{1/3}.$$  \hspace{1cm} (8)

Here $\Delta V_i$ is the volume of cell $i$ and $N$ is the total number of cells present in the grid.

In the guidelines, the grid refinement factors $h_{\text{medium}}/h_{\text{fine}}$ and $h_{\text{coarse}}/h_{\text{medium}}$ are desired to be larger than $1.3$ (these factors are $h_{\text{medium}}/h_{\text{fine}} = 1.30$ and $h_{\text{coarse}}/h_{\text{medium}} = 1.36$). Since the sum of the volumes of the cells in the calculation domain is the same for all grids, the refinement factor for the
cell number ratios $N_{\text{fine}}/N_{\text{medium}}$ and $N_{\text{medium}}/N_{\text{coarse}}$ should be larger than 2.19. These conditions are met by the employed grids, see Table 4.

For the grid independence study, the pressure coefficient $\psi$, the total-to-static efficiency $\eta_{ts}$ and the power coefficient $\lambda$ have been selected as key variables, as the focus of this study is on the aerodynamic performance.

Following the method described in [35] that is based on Richardson extrapolation for the dependence of the key variables on the mesh size $h$, the estimated discretization errors have been determined, see Table 5 for flow coefficient $\phi = 0.156$ near Best Efficiency Point (BEP for short; based on total-to-static efficiency $\eta_{ts}$). Here $\phi_k$ denotes the value of variable on the $k$ grid, $p'$ is apparent order of the numerical discretisation and $\phi_{\text{ext}}$ is the value extrapolated to zero grid size, $h \to 0$, and $GCI_{\text{fine}}$ is grid convergence index for the fine grid.

Hence, according to results in Table 5, the numerical uncertainty in the fine-grid solution is 1.25% for the pressure coefficient $\psi$, 3.6% for the total-to-static efficiency $\eta_{ts}$ and even smaller for the power coefficient $\lambda$, which means that the computational results are (effectively) independent of grid size.

Similar results for the estimated discretisation errors for the off-design flow coefficients $\phi = 0.105$ and $\phi = 0.229$ are given in Tables 7 and 8 in Appendix B. Considering the computational costs and limited gains in numerical accuracy, the medium grid size is selected as grid size for the analyses in Section 4.

3.4. Validation of CFD Simulations for a Fan with Medium Hub-to-tip Ratio

To verify that the employed CFD simulation techniques are adequate for the simulation of low-pressure axial fans with medium hub-to-tip ratio $\kappa$ (rather than the fan with the low $\kappa$ that is the focus of the current study), CFD simulations have been performed of the US17 fan that is fully described in [22]. For this fan with $\kappa = 0.4$, full geometrical data and high-quality measured aerodynamic characteristics are available. The current results for this fan are described in Appendix C. The pressure coefficient $\psi$ and the total-to-static coefficient $\eta_{ts}$ are predicted by the CFD simulations with an accuracy of 3.3% and 3.4% (relative to the experimental data) at BEP, respectively. Note that in these CFD simulations the tip gap, with tip gap ratio $s/D_{\text{fan}} = 0.1\%$, has been taken into account.

4. CFD results

For the fan with small hub-to-tip ratio $\kappa = 0.14$, CFD results are shown here for the reference case in Section 4.1, with “baseline” variables as indicated in Table 3. In addition, the CFD results that are analysed here focus on the influence on the predicted aerodynamic performance of the following factors. The influence of the trailing edge shape (sharp vs. rounded) is analysed in Section 4.2. The influence of part of the blade not having an aerodynamic shape near the root of the blades (indicated...
as non-airfoil part in Figure 2) is analysed in Section 4.3. To the best of the authors’ knowledge, the influence of this non-airfoil part of the fan blades on the aerodynamic performance has not been studied in literature. Section 4.4 considers the influence of the tip gap and Section 4.5 that of the employed turbulence model. Note that the first three factors involve modelling of the blade geometry, while the other involves flow (turbulence) modelling.

CFD results have been obtained for a number of flow coefficients. In the following, the procedure used to display the aerodynamic performance curves consists of: (i) showing by markers the computed data points consisting of flow coefficient and pressure coefficient and (ii) showing by solid lines a spline interpolation through these data points. This is done to clearly show the trends in the performance curves. An analogous procedure is followed for the total-to-static efficiency.

4.1. Reference Case

The CFD simulations with variables shown as “baseline” in Table 3 form the reference case for following CFD simulations. For this reference case, the predicted aerodynamic performance is shown in Figure 6.

As shown in Figure 6a, the predicted pressure coefficients $\psi$ are lower than the experimental results, especially for large flow coefficients $\varphi$ (with a maximum deviation of 52% at $\varphi = 0.229$). For small $\varphi$, near the stall region, the agreement is reasonably good.

The results for the total-to-static efficiency $\eta_{ts}$ are shown in Figure 6b. The predicted results are lower than the experimental results and the largest deviation (compared with experimental data) is 50% at $\varphi = 0.229$. The maximum predicted (by CFD) total-to-static efficiency $\eta_{ts}$ is 59% at $\varphi = 0.13$, while the experimental maximum $\eta_{ts}$ is 60% at $\varphi = 0.154$. The maximum predicted total-to-total efficiency $\eta_{tt} = Q \cdot \Delta p_0 / P_{\text{shaft}}$ (from CFD; no experimental data are available for $\eta_{tt}$; all velocity components are accounted for in the dynamic pressure $p_0$) is 80.7% at $\varphi = 0.18$.

For the power coefficient $\lambda$ (data not shown), the deviation equals 14% at BEP, being even larger for large flow coefficients $\varphi$. Overall, these CFD simulations of the reference case do not satisfactorily quantify the aerodynamic performance of the investigated fan.

In order to gain additional understanding of the performance of this fan, the flow fields have been analysed. As indicated in Section 1, backflow regions [1, 3] are expected for this fan. This is confirmed by the current CFD results, see the streamlines in the meridional plane depicted in Figure 7, for flow coefficients $\varphi = 0.079$, 0.156 and 0.229. As expected, the extent of the backflow region decreases with increasing flow coefficient $\varphi$. For the lower flow coefficients, the vortex downstream of the blades occupies a significant part of the flow path (for $\varphi = 0.156$ nearly 40% based on its radial extent).

The streamlines (based on the relative velocity) in the blade-to-blade plane are shown in Figure 8 at different spanwise locations (hub, midspan and tip) near BEP. The flow is attached to the blades, except near the hub.
The vortex distribution, $r\bar{u}_\theta,2(r)$ (with $\bar{u}_\theta,2$ the circumferentially-averaged circumferential component of the absolute velocity downstream of the blades), is important for the distribution of the energy transfer from blades to fluid, as follows from the Euler relation. The CFD-results near BEP have been used to compute the vortex distribution. The result shown in Figure 9 demonstrates that the vortex distribution strongly differs from a free-vortex design, for which $r\bar{u}_\theta,2(r) = \text{const.}$

4.2. Trailing Edge Geometry

In classical airfoil theory the trailing edge shapes considered are sharp (or cusp-shaped with zero thickness), as this enforces both the location of the rear stagnation point and the local direction of the flow. However, for strength considerations, actual trailing edge shapes are different. Hence, two trailing edge shapes are considered, the sharp trailing edge conforming to the Wortmann profile and the rounded trailing edge conforming to the actual geometry, see Figure 10a. A zoom-in on the trailing edge region in Figure 10b shows the (small) difference between the sharp and rounded trailing edge shapes.

CFD simulations with these two trailing edge shapes have been performed, using the conditions shown as “baseline” in Table 3. The employed grids have (approximately) the same size and grid quality. Note that in these CFD simulations the tip gap has not been taken into account.

The predicted aerodynamic performance for the two trailing edge shapes is shown in Figure 11. Although the difference in blade section geometry is rather small, the predicted aerodynamic performance is significantly different.

As shown in Figure 11a, the CFD simulations with the rounded trailing edge (much) better predict, compared with reference case with sharp trailing edge, the pressure coefficient $\psi$ over the full range of considered flow coefficients $\varphi$, especially at high $\varphi$.

For the total-to-static efficiency $\eta_{ts}$, the results shown in Figure 11b for the reference case and for the rounded trailing edge are nearly the same for flow coefficients $\varphi$ near BEP, and in agreement with experimental results. For higher $\varphi$, the CFD prediction for $\eta_{ts}$ with the sharp trailing edge is lower than that with the rounded trailing edge geometry. The latter is higher than the measured efficiency.

For the power coefficient $\lambda$ (data not shown), the prediction with the rounded trailing edge case shows good agreement with measurements with 3% deviation at BEP, while for the sharp trailing edge case, the deviation equals 14% at BEP and is even larger for large flow coefficients.

Thus, the influence of the trailing edge shape on the CFD predictions of the aerodynamic characteristics is significant. The use of the actual, rounded trailing edge shape yields CFD predictions that overall are in much better agreement with the experimental aerodynamic performance characteristics.

To investigate the origin of these differences, the flow field is visualised around the trailing edge where the two geometries differ, see Figure 12 at midspan for flow coefficient $\varphi = 0.156$ near BEP. With the sharp trailing edge, flow separation is predicted at the suction side near the trailing edge,
while for the rounded trailing edge the extent of the separation is much smaller. The streamlines in the meridional plane with the rounded trailing edge for $\varphi = 0.156$ (data not shown) are very similar to those with the sharp trailing edge that are displayed in Figure 7.

4.3. Non-Airfoil Blade Sections

The considered fan has non-airfoil blade sections near the root (see Figure 2b). The influence of such non-airfoil sections on the aerodynamic performance characteristics is investigated by comparing CFD results for the case that accounts for the presence of the non-airfoil sections with the case where only the “main blade” is represented. The hub diameters for these two cases are $D_{\text{hub}} = 0.250m$ and $0.339m$, respectively.

CFD simulations of these two blades have been performed, with rounded trailing edge, and where other settings shown as “baseline” in Table 3 have been employed. The employed grids have (approximately) the same size and grid quality.

The CFD predictions for the aerodynamic performance characteristics of fan blades with and without non-airfoil sections near the root are shown in Figure 13. When accounting for the non-airfoil blade root, both pressure coefficient $\psi$ and total-to-static efficiency $\eta_{ts}$ from the CFD simulations are smaller than for blades without the non-airfoil root over the whole range of flow coefficients $\varphi$. As for the power coefficient $\lambda$ (data not shown), nearly the same predictions have been obtained for these two cases, with the largest deviation of 2.1% near flow coefficient $\varphi = 0.105$.

To investigate the origin of the difference in the aerodynamic performance between these two blades, especially for small flow coefficients $\varphi$, the streamlines in the meridional surface for $\varphi = 0.105$ are shown in Figure 14. The downstream backflow region for the blade with the non-airfoil sections near the root is larger than that for without the non-airfoil sections. Also visible in Figure 14b is that upstream of the blades a vortex is present near the hub for the blades with non-airfoil sections near the root. The presence of this vortex is considered to result in larger secondary flow losses and lower static pressure increases. For larger flow coefficient $\varphi = 0.229$ the upstream vortex effectively is not present, and the extent of the downstream backflow region is reduced.

4.4. Tip Gap

It is well-known (see for instance [36–38]) that the size of the tip gap affects the aerodynamic performance, due to the tip leakage flow from pressure side to suction side that results in the formation of a tip vortex and in blockage. Using LES simulations, tip leakage flows have been studied in [10].

An increase of the tip gap results in a decrease of pressure rise and efficiency [36, 37, 39], and more severe stall [40, 41]. The tip gap is accounted for in some CFD studies of low-pressure axial fans [15–18, 23, 42, 43], but ignored in others [14, 18, 25, 43]. For low-pressure axial fans, CFD simulations
with different tip gaps have been reported [17]. Their results show that the deviations between CFD predictions and measurements increase with increasing tip gap.

Here CFD simulations with and without tip gap are performed (with the rounded trailing edge and excluding the non-airfoil blade sections near the root) to investigate the influence of the tip gap size on the predicted aerodynamic performance of the fan with small hub-to-tip ratio $\kappa$. The multi-block structured grid in the tip gap is composed of an H and an O grid. There are 17 points and 13 points in the spanwise and azimuthal direction, respectively, with 193 points and 81 points wrapping around the O and the H grid. The number of grid points in the tip gap region is sufficiently large according to [16]. The sensitivity of the grid resolution in the tip gap region has been explored by generating grids with 13 as well as 17 points in spanwise direction. The CFD results show that the largest deviation in the predicted aerodynamic performance with these two tip gap grid resolutions is 1.4% in the total-to-static efficiency.

The predicted pressure coefficient $\psi$ and the total-to-static efficiency $\eta_{ts}$ are shown in Figure 15 and the predicted power coefficient $\lambda$ is shown in Figure 22 in Appendix D. As expected, the pressure coefficient $\psi$, the total-to-static efficiency $\eta_{ts}$ and the power coefficient $\lambda$ decrease with increasing tip-gap ratio $s/D_{fan}$. The difference in $\lambda$ between the blade with 0.22% tip gap ratio and the blade without tip gap is small. The influence of the tip gap size becomes smaller with increasing flow coefficient $\varphi$.

Quantitatively, for the actual tip gap ratio of 0.43% in the measurements, the CFD results are quantitatively significantly different (both for pressure rise $\psi$ and total-to-static efficiency $\eta_{ts}$) from measured values. Unusually and unexpectedly, CFD simulations significantly underpredict the aerodynamic performance in this case.

The meridional streamlines show that tip recirculation and a backflow region near the hub are present for the cases of 0.43% and 0.22% tip gap ratio when $\varphi = 0.156$ (data not shown). These flow phenomena result in blockage.

The meridional streamlines with 0.43% and 0.22% tip gap ratio for $\varphi = 0.156$ indicate (data not shown) that tip recirculation is present as well the backflow region near the hub, resulting in blockage.

The streamlines in four streamwise cut planes near the tip area are shown in Figure 16 (viewed from downstream of the blade) for these two tip gap ratios. The red dashed lines refer to the boundary of each cut plane, numbers of “1” to “4” refer to the location from near the leading edge to near the trailing edge. The tip vortex starts from the blade leading edge, appears at mid location of the blade and further develops at the trailing edge and downstream of it. The vortex region increases significantly in size with larger tip gap ratio, which results in larger blockage and losses.

Based on the current results, CFD simulations for the fan with small hub-to-tip ratio $\kappa$ can not adequately predict the influence of the tip gap on the aerodynamic performance. Accordingly, for use of CFD within an industrial context, it is recommended not to take the tip gap into account. Then quantitatively good agreement can be obtained between CFD predictions and experiments for the fan.
considered here in detail. In a scientific context, CFD simulations accounting for the tip gap need to be studied in (much) more detail.

4.5. Turbulence Model

For the required turbulence model, there is no definite answer as to which model is the most suitable in general for low-pressure axial fan CFD simulations. The influence of the turbulence model on CFD predictions has been studied in [43] for different variants of the $k$-$\epsilon$ turbulence model.

The Spalart-Allmaras turbulence model [30] has been initially applied here, based on considerations of low computational cost and successful use in investigations of axial flow fans [16]. The SST turbulence model is a two-equation model that combines advantages of the Wilcox $k$ – $\omega$ model and the $k$ – $\epsilon$ model. It has improved capability for flow predictions involving separation [17]. Hence, the SA and SST turbulence models have been selected to investigate the influence of the turbulence model on the predicted aerodynamic performance of the axial fan with small hub-to-tip ratio $\kappa$. CFD simulations with the rounded trailing edge shape and the main blade without tip gap have therefore been performed.

The CFD results are shown in Figure 17. The SST-model predicts a lower pressure rise $\psi$ (except near stall conditions) than the SA-model. The results with the SA-model agree well with experimental results. With respect to the total-to-static efficiency $\eta_{ts}$, the results with the SST-model overpredict and underpredict $\eta_{ts}$ for small and large $\varphi$, respectively. The SST-model underpredicts the power coefficient $\lambda$ over the whole range of $\varphi$ (data not shown), with a deviation of 9% at BEP in comparison to the measurements. The results with the SA-model are in much better agreement with the experimental aerodynamic performance.

An investigation of the flow fields shows that the SST-model predicts much larger backflow regions than obtained with the SA-model. The meridional surface streamlines are shown in Figure 18. Compared with the streamlines with the SA-model shown in Figure 7, the backflow region downstream of the blades is larger, corresponding to larger losses.

For a more detailed comparison the (circumferentially-averaged) total and static pressures have been compared at different axial locations. Upstream of the blades these are the same for both turbulence models. Downstream of the blades the total pressures are almost equal, but at an axial distance $0.5D_{fan}$ downstream of the blades the SA-model predicts a higher static pressure than the SST-model. This means that the predicted velocity distributions are different, corresponding to the larger predicted backflow region with the SST-model.

Overall, the use of the Spalart-Allmaras turbulence model is recommended for CFD simulations of axial fans with small hub-to-tip ratio, as it gives much better agreement with the experimental results for the fan considered here in detail. Note that this sensitivity of the CFD predictions to the employed turbulence model is not noted for the fan with medium hub-to-tip ratio $\kappa = 0.45$ [22, 44],
see Appendix C.

5. Conclusions

Although CFD simulations are widely used in the design of turbomachinery, for axial fans attention has been limited to fans with medium to large hub-to-tip ratio $\kappa$. The current study therefore focuses on a RANS CFD simulation strategy for fans with a small hub-to-tip ratio $\kappa$. Using reliable measurements of the aerodynamic performance of an axial fan with small hub-to-tip ratio and a complete geometrical description of the fan, many CFD simulations have been performed and analysed in order to investigate the influence of a number of parameters, ultimately aimed at formulating a strategy and guidelines for obtaining accurate CFD predictions of the aerodynamic performance (pressure coefficient $\psi$ and total-to-static efficiency $\eta_{ts}$) for such machines.

The trailing edge shape (sharp vs. rounded) has a large influence on the predicted aerodynamic performance. The actual trailing edge geometry (rounded in this case) should be used in the CFD simulations for improved agreement with the experimental aerodynamic performance.

The current results show that the presence of non-airfoil sections near the root has a minor influence on the pressure coefficient and hence on the total-to-static efficiency, due the formation of a vortex upstream from the blades near the hub. Overall, the “main blade” part well represents the aerodynamic performance.

With increasing tip gap ratio, the pressure coefficient and the total-to-static efficiency decrease. The CFD simulations with the actual tip gap ratio present in the experiments do not adequately predict the influence of the tip gap on the aerodynamic performance. For use of CFD simulations within an industrial context, it is recommended not to take the tip gap into account, as then quantitatively good agreement is obtained between CFD predictions and experiments.

CFD simulations employing the SST turbulence model significantly underpredict the aerodynamic performance in comparison to the experiments, while those using the SA model yield much better agreement. Therefore, the use of the SA turbulence model is recommended for CFD simulations of axial fans with small hub-to-tip ratio.

For future studies it is recommended to: (i) perform detailed flow measurements (velocity and pressure distributions) for detailed validation of CFD simulations, (ii) account for the actual hub shape downstream of the fan (bulb-shaped; not extended to the outlet as in the current CFD simulations) present in the experiments, (iii) study in more detail the influence of the tip gap on the aerodynamic performance, (iv) consider the influence of the outflow region on the aerodynamic performance and (v) perform CFD simulations for other fan types with small hub-to-tip ratios.
Acknowledgements

The authors would like to thank China Scholarship Council, Howden Netherlands and Numeca International for support of this research. Peter Holkers, Sander Venema and Bastiaan Holm of Howden Netherlands are thanked in particular for valuable discussions. Peter Holkers is also thanked for providing the experimental data on the aerodynamic performance of the Howden fan. The anonymous reviewers are thanked for their detailed and constructive comments which have helped to improve this paper.

Appendix A

In the test facility that has been described in Section 2, various variables are measured. These variables are listed, together with the employed sensors and their accuracy, in Table 6.

Appendix B

Following the method described in [35], the estimated discretisation errors for small and large flow coefficients $\varphi$ are shown in Table 7 and 8. These results complement those in Table 5 for $\varphi = 0.156$ near BEP.

These results show that the computational results are effectively independent of grid size, not only near BEP, but also for off-design conditions.

Appendix C

In order to verify that the employed simulation techniques are adequate for CFD simulations of low-pressure axial fans with medium hub-to-tip ratio $\kappa$, the current CFD simulations for the US17 are reported here. The detailed information of fan geometry and measurements is obtained from [22, 44]. The hub-to-tip ratio of this fan $\kappa = 0.45$. Other dimensionless fan parameters are given in Table 1.

The flow around the blade with 0.1% tip clearance ratio is simulated, following the approach described in Section 3. The fluid type is air (real gas) and the Spalart-Allmaras and the SST turbulence models are both considered. The multi-block structured grid has been generated with AutoGrid and the total number of grid points is $1.22 \times 10^6$. The minimum skewness angle is $21.42^\circ$. The aspect ratio of nearly all cells is smaller than 1000 and the maximum expansion ratio is 3.8. Hence, the grid satisfies the quality criteria given in [32] and explained in Section 3.2.

The convergence criterion for the iterative solution of the discretised equations is a reduction of the global residual by at least three orders of magnitude. The convergence history curves with the SA and the SST turbulence model for a flow coefficient $\varphi$ near BEP are shown in Figure 19, indicating that good convergence is obtained with both turbulence models.
The CFD predictions for the aerodynamic performance characteristics for this fan are shown in Figure 20. The predictions with both turbulence models show good agreement with measurements. For the pressure coefficient $\psi$, results with the SA and the SST turbulence models are nearly the same and higher than the experimental data, except at the lowest flow coefficient $\varphi$ near stall. Near BEP, the deviation from measurements with both models is 2.3%. For the total-to-static efficiency $\eta_{ts}$, except at the lowest flow coefficient $\varphi$, all predictions are higher than measurements, the prediction of SST model is 4% higher near BEP. For the power coefficient $\lambda$ (data not shown), the deviation with the SA model near BEP is 0.7%; the SST model underpredicts the power coefficient $\lambda$, with the largest deviation of 4% at BEP. Thus, the aerodynamic performance characteristics can be well predicted with both turbulence models.

The meridional surface streamlines predicted when using the SA turbulence model near BEP ($\varphi = 0.16$) are shown in Figure 21. As expected, no backflow region downstream of the fan is observed in the CFD simulation results.

Concluding, the employed simulation techniques are adequate for CFD simulations of low-pressure axial fans with medium hub-to-tip ratio, here with $\kappa = 0.45$.

Appendix D

The predicted power coefficients $\lambda$ for different tip gap size are shown in Figure 22. With increasing tip gap ratio, the power coefficient decreases slightly. The difference in $\lambda$ between the blade with 0.22% tip gap ratio blade and that without tip gap is small. The influence of tip gap ratio decreases with increasing flow coefficient $\varphi$. Similar trends have been observed experimentally and through CFD simulations in [23].
6. References


7. Table

List of Tables

1. Dimensionless Fan Parameters (defined in Eqs.(2)-(4)) at Design Conditions. 24
2. Geometrical and Design Parameters of the Investigated Fan. 25
3. Investigated Variables. 26
4. Information on Three Grids. 27
5. Grid Convergence Analysis for Flow Coefficient $\varphi = 0.156$ near BEP. 28
6. Measured Variables, Corresponding Sensors and their Accuracy [29]. 29
7. Grid Convergence Analysis for Small Flow Coefficient $\varphi = 0.105$. 30
8. Grid Convergence Analysis for Large Flow Coefficient $\varphi = 0.229$. 31
Table 1: Dimensionless Fan Parameters (defined in Eqs.(2)-(4)) at Design Conditions.

<table>
<thead>
<tr>
<th>Reference</th>
<th>Blade Count</th>
<th>$\kappa$</th>
<th>$\varphi$</th>
<th>$\psi$</th>
<th>Ma</th>
<th>Re</th>
</tr>
</thead>
<tbody>
<tr>
<td>Current study</td>
<td>6</td>
<td>0.14</td>
<td>0.15</td>
<td>0.10</td>
<td>0.14</td>
<td>$4.6 \times 10^5$</td>
</tr>
<tr>
<td>Wilkinson et al. [23]</td>
<td>8</td>
<td>0.29</td>
<td>0.13</td>
<td>0.057</td>
<td>0.17</td>
<td>$3.0 \times 10^6$</td>
</tr>
<tr>
<td>Jin et al. [16]</td>
<td>5</td>
<td>0.35</td>
<td>0.23</td>
<td>0.093</td>
<td>0.11</td>
<td>$9.9 \times 10^4$</td>
</tr>
<tr>
<td>Masi et al. [18]</td>
<td>10</td>
<td>0.35</td>
<td>0.31</td>
<td>0.073</td>
<td>0.14</td>
<td>$1.6 \times 10^5$</td>
</tr>
<tr>
<td>Beiler &amp; Carolus [14]</td>
<td>6</td>
<td>0.40</td>
<td>0.15</td>
<td>0.15</td>
<td>0.14</td>
<td>$7.7 \times 10^4$</td>
</tr>
<tr>
<td>Zhu &amp; Carolus [17, 22]</td>
<td>5</td>
<td>0.45</td>
<td>0.19</td>
<td>0.16</td>
<td>0.14</td>
<td>$7.4 \times 10^4$</td>
</tr>
<tr>
<td>Vad et al. [15, 21]</td>
<td>12</td>
<td>0.6</td>
<td>0.33</td>
<td>0.23</td>
<td>0.13</td>
<td>$4.6 \times 10^5$</td>
</tr>
</tbody>
</table>
Table 2: Geometrical and Design Parameters of the Investigated Fan.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotational Speed [rpm]</td>
<td>497</td>
</tr>
<tr>
<td>$D_{\text{casing}}$ [m]</td>
<td>1.845</td>
</tr>
<tr>
<td>$D_{\text{fan}}$ [m]</td>
<td>1.829</td>
</tr>
<tr>
<td>$\kappa$ [-]</td>
<td>0.14</td>
</tr>
<tr>
<td>Tip Gap Ratio</td>
<td>0.43%</td>
</tr>
<tr>
<td>Number of Blades</td>
<td>6</td>
</tr>
<tr>
<td>Stagger Angle [deg]</td>
<td>15</td>
</tr>
<tr>
<td>Chord Length of Main Blade [m]</td>
<td>0.15</td>
</tr>
<tr>
<td>Solidity at Tip [-]</td>
<td>0.157</td>
</tr>
<tr>
<td>Solidity at Blade Section A [-]</td>
<td>0.844</td>
</tr>
</tbody>
</table>
Table 3: Investigated Variables.

<table>
<thead>
<tr>
<th>Item</th>
<th>Baseline</th>
<th>Alternative</th>
</tr>
</thead>
<tbody>
<tr>
<td>Trailing Edge Shape</td>
<td>Sharp</td>
<td>Rounded</td>
</tr>
<tr>
<td>Non-airfoil Blade Root</td>
<td>Not Included</td>
<td>Included</td>
</tr>
<tr>
<td>Tip Gap</td>
<td>Not Included</td>
<td>Included</td>
</tr>
<tr>
<td>Turbulence Model</td>
<td>Spalart-Allmaras</td>
<td>SST</td>
</tr>
</tbody>
</table>
Table 4: Information on Three Grids.

<table>
<thead>
<tr>
<th>Grid Level</th>
<th>Coarse</th>
<th>Medium</th>
<th>Fine</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of Points in Spanwise Direction</td>
<td>41</td>
<td>57</td>
<td>89</td>
</tr>
<tr>
<td>Number of Points on Blade to Blade Surface</td>
<td>14223</td>
<td>25479</td>
<td>36063</td>
</tr>
<tr>
<td>Total Number of Grid Points</td>
<td>$5.8 \times 10^5$</td>
<td>$1.5 \times 10^6$</td>
<td>$3.2 \times 10^6$</td>
</tr>
</tbody>
</table>
Table 5: Grid Convergence Analysis for Flow Coefficient $\phi = 0.156$ near BEP.

<table>
<thead>
<tr>
<th>Key Variable</th>
<th>$\phi_{\text{coarse}}$</th>
<th>$\phi_{\text{medium}}$</th>
<th>$\phi_{\text{fine}}$</th>
<th>$p'$</th>
<th>$\phi_{\text{ext}}$</th>
<th>$GCI_{\text{fine}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\psi$ [-]</td>
<td>0.1000</td>
<td>0.1005</td>
<td>0.1011</td>
<td>1.33</td>
<td>0.1015</td>
<td>1.25%</td>
</tr>
<tr>
<td>$\eta_{ts}$ [%]</td>
<td>57.81</td>
<td>58.19</td>
<td>58.58</td>
<td>0.68</td>
<td>59.87</td>
<td>3.6%</td>
</tr>
<tr>
<td>$\lambda$ [-]</td>
<td>0.0398</td>
<td>0.0398</td>
<td>0.0397</td>
<td>9.47</td>
<td>0.0398</td>
<td>0.0024%</td>
</tr>
</tbody>
</table>
Table 6: Measured Variables, Corresponding Sensors and their Accuracy [29].

<table>
<thead>
<tr>
<th>Measured Variable</th>
<th>Sensor</th>
<th>Sensor Accuracy</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure Difference Over Nozzle</td>
<td>Differential Pressure Transducer</td>
<td>±1%</td>
</tr>
<tr>
<td>Total Pressure Inlet Chamber</td>
<td>Differential Pressure Transducer</td>
<td>±1%</td>
</tr>
<tr>
<td>Fan Speed</td>
<td>Induction Sensor</td>
<td>±0.5%</td>
</tr>
<tr>
<td>Fan Shaft Torque</td>
<td>Torque meter</td>
<td>±2%</td>
</tr>
<tr>
<td>Ambient Temperature</td>
<td>Temperature Sensor</td>
<td>±0.5°C</td>
</tr>
<tr>
<td>Relative Humidity</td>
<td>Humidity Sensor</td>
<td>±2%</td>
</tr>
<tr>
<td>Atmospheric Pressure</td>
<td>Barometric pressure sensor</td>
<td>±170 Pa</td>
</tr>
</tbody>
</table>
Table 7: Grid Convergence Analysis for Small Flow Coefficient $\phi = 0.105$.

<table>
<thead>
<tr>
<th>Key Variable</th>
<th>$\phi_{\text{coarse}}$</th>
<th>$\phi_{\text{medium}}$</th>
<th>$\phi_{\text{fine}}$</th>
<th>$p'$</th>
<th>$\phi_{\text{ext}}$</th>
<th>$GCI_{\text{fine}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\psi$ [-]</td>
<td>0.1328</td>
<td>0.1337</td>
<td>0.1333</td>
<td>3.10</td>
<td>0.1330</td>
<td>0.28%</td>
</tr>
<tr>
<td>$\eta_{ts}$ [%]</td>
<td>54.95</td>
<td>55.26</td>
<td>55.43</td>
<td>1.45</td>
<td>55.81</td>
<td>0.85%</td>
</tr>
<tr>
<td>$\lambda$ [-]</td>
<td>0.0376</td>
<td>0.0376</td>
<td>0.0375</td>
<td>0.77</td>
<td>0.0373</td>
<td>0.77%</td>
</tr>
</tbody>
</table>
Table 8: Grid Convergence Analysis for Large Flow Coefficient $\varphi = 0.229$.

<table>
<thead>
<tr>
<th>Key Variable</th>
<th>$\phi_{\text{coarse}}$</th>
<th>$\phi_{\text{medium}}$</th>
<th>$\phi_{\text{fine}}$</th>
<th>$p'$</th>
<th>$\phi_{\text{ext}}$</th>
<th>$GCI_{\text{fine}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\psi$ [-]</td>
<td>0.0231</td>
<td>0.0242</td>
<td>0.0244</td>
<td>5.62</td>
<td>0.0245</td>
<td>0.29%</td>
</tr>
<tr>
<td>$\eta_{ts}$ [%]</td>
<td>21.88</td>
<td>22.98</td>
<td>23.19</td>
<td>5.23</td>
<td>23.26</td>
<td>0.38%</td>
</tr>
<tr>
<td>$\lambda$ [-]</td>
<td>0.0358</td>
<td>0.0357</td>
<td>0.0357</td>
<td>1.70</td>
<td>0.0356</td>
<td>0.25%</td>
</tr>
</tbody>
</table>
8. Figure

List of Figures

1 Examples of Low-pressure Axial Fans. .............................................. 34
2 Geometrical Description of the Howden Fan with Small Hub-to-tip Ratio ........ 35
3 Schematic Overview of the Howden Cooling Fan Test Facility [29]. .............. 36
4 Computational Domain. ................................................................. 37
5 Convergence History of the Reduction of the Global Residual (with Respect to the Initial Global Residual) with Iteration Step. ................................. 38
6 Comparison of Measurements and CFD Predictions for the Baseline Case: (a) Pressure Coefficient $\psi$ and (b) Total-to-Static Efficiency $\eta_{ts}$. ............... 39
7 Streamlines in the Meridional Plane for the Reference Case for Different Flow Coefficients: (a) $\varphi = 0.079$; (b) $\varphi = 0.156$ (near BEP); (c) $\varphi = 0.0229$. The Fan Blades are Indicated by the Solid Grey Rectangles. ................................................... 40
8 Velocity Streamlines based on the Relative Velocity in the Blade-to-blade Plane at Different Spanwise Locations for the Reference Case: (a) near Hub, (b) Mid-span and (c) near Tip. Results for $\varphi = 0.156$ near BEP. ........................................ 41
9 Vortex Distribution of Investigated Fan for $\varphi = 0.156$, near BEP for the Reference Case. 42
10 Shape of Airfoil Section and of Trailing Edge, for Sharp (Blue Curve) and Rounded (Red Curve) Trailing Edge Shapes. ................................................. 43
11 Comparison between Measurements and CFD Predictions with Sharp and Rounded Trailing Edge Geometry: (a) Pressure Coefficient $\psi$ and (b) Total-to-Static Efficiency $\eta_{ts}$. ................................................................. 44
12 Relative Velocity Vector Field in the Blade-to-blade Plane near the Trailing Edge at Mid-span Location for Flow Coefficient $\varphi = 0.156$ near BEP. .................. 45
13 Comparison of Measurements and CFD Predictions with and without Non-airfoil Blade Sections near Root: (a) Pressure Coefficient $\psi$ and (b) Total-to-Static Efficiency $\eta_{ts}$. ......................................................... 46
14 Comparison of Meridional Surface Streamlines: (a) Without Non-airfoil Blade Sections near Root; (b) With Non-airfoil Blade Sections near Root; Both for Flow Coefficient $\varphi = 0.105$. ......................................................... 47
15 Dependence of Pressure Coefficient $\psi$ and Total-to-static Efficiency $\eta_{ts}$ on Flow Coefficient $\varphi$: Measurements, CFD Simulations with Different Sizes of the Tip Gap and without Tip Gap. Note that in the experiments the tip gap ratio equals 0.43%. ........ 48
16 Surface Streamlines at Four Streamwise Cut Planes (“1” to “4”, from leading edge towards trailing edge): (a) Blade with 0.43% Tip Gap Ratio; (b) Blade with 0.22% Tip Gap Ratio. Both at Flow Coefficient $\varphi = 0.156$, near BEP. ............................... 49
Dependence of Pressure Coefficient $\psi$ and Total to Static Efficiency $\eta_{ts}$ on Flow Coefficient $\varphi$: Measurements, CFD Simulations with SST and SA Turbulence Models.

Meridional Streamlines: SST Turbulence Model ($\varphi = 0.156$).

Convergence History Curves: SA and SST Turbulence Model.

Comparison of Measurements and CFD Predictions of US17 Fan: (a) Pressure Coefficient $\psi$ and (b) Total-to-Static Efficiency $\eta_{ts}$.

Meridional Surface Streamlines for the US17 Fan with $\varphi = 0.16$ near BEP; Results Using SA Turbulence Model.

Dependence of Power Coefficient $\lambda$ on Flow Coefficient $\varphi$: Measurements, CFD Simulations with Different Sizes of the Tip Gap and without Tip Gap. Note: Tip Gap Ratio Equals 0.43% in the Experiments.
Figure 1: Examples of Low-pressure Axial Fans.
Figure 2: Geometrical Description of the Howden Fan with Small Hub-to-tip Ratio

Figure 3: Schematic Overview of the Howden Cooling Fan Test Facility [29].
Figure 4: Computational Domain.
Figure 5: Convergence History of the Reduction of the Global Residual (with Respect to the Initial Global Residual) with Iteration Step.
Figure 6: Comparison of Measurements and CFD Predictions for the Baseline Case: (a) Pressure Coefficient $\psi$ and (b) Total-to-Static Efficiency $\eta_{ts}$.
Figure 7: Streamlines in the Meridional Plane for the Reference Case for Different Flow Coefficients: (a) \( \varphi = 0.079 \); (b) \( \varphi = 0.156 \) (near BEP); (c) \( \varphi = 0.229 \). The Fan Blades are Indicated by the Solid Grey Rectangles.
Figure 8: Velocity Streamlines based on the Relative Velocity in the Blade-to-blade Plane at Different Spanwise Locations for the Reference Case: (a) near Hub, (b) Mid-span and (c) near Tip. Results for $\phi = 0.156$ near BEP.
Figure 9: Vortex Distribution of Investigated Fan for $\varphi = 0.156$, near BEP for the Reference Case.
Figure 10: Shape of Airfoil Section and of Trailing Edge, for Sharp (Blue Curve) and Rounded (Red Curve) Trailing Edge Shapes.
Figure 11: Comparison between Measurements and CFD Predictions with Sharp and Rounded Trailing Edge Geometry:
(a) Pressure Coefficient $\psi$ and (b) Total-to-Static Efficiency $\eta_{ts}$. 

(a) $\varphi - \psi$

(b) $\varphi - \eta_{ts}$
Figure 12: Relative Velocity Vector Field in the Blade-to-blade Plane near the Trailing Edge at Mid-span Location for Flow Coefficient $\varphi = 0.156$ near BEP.
Figure 13: Comparison of Measurements and CFD Predictions with and without Non-airfoil Blade Sections near Root: (a) Pressure Coefficient \( \psi \) and (b) Total-to-Static Efficiency \( \eta_{ts} \).
Figure 14: Comparison of Meridional Surface Streamlines: (a) Without Non-airfoil Blade Sections near Root; (b) With Non-airfoil Blade Sections near Root; Both for Flow Coefficient $\phi = 0.105$. 
Figure 15: Dependence of Pressure Coefficient $\psi$ and Total-to-static Efficiency $\eta_{ts}$ on Flow Coefficient $\phi$: Measurements, CFD Simulations with Different Sizes of the Tip Gap and without Tip Gap. Note that in the experiments the tip gap ratio equals 0.43%.
Figure 16: Surface Streamlines at Four Streamwise Cut Planes (“1” to “4”, from leading edge towards trailing edge): (a) Blade with 0.43% Tip Gap Ratio; (b) Blade with 0.22% Tip Gap Ratio. Both at Flow Coefficient $\varphi = 0.156$, near BEP.
Figure 17: Dependence of Pressure Coefficient $\psi$ and Total to Static Efficiency $\eta_{ts}$ on Flow Coefficient $\varphi$: Measurements, CFD Simulations with SST and SA Turbulence Models.
Figure 18: Meridional Streamlines: SST Turbulence Model ($\varphi = 0.156$).
Figure 19: Convergence History Curves: SA and SST Turbulence Model.
Figure 20: Comparison of Measurements and CFD Predictions of US17 Fan: (a) Pressure Coefficient $\psi$ and (b) Total-to-Static Efficiency $\eta_{ts}$. 

(a) $\varphi - \psi$

(b) $\varphi - \eta_{ts}$
Figure 21: Meridional Surface Streamlines for the US17 Fan with $\varphi = 0.16$ near BEP; Results Using SA Turbulence Model.
Figure 22: Dependence of Power Coefficient $\lambda$ on Flow Coefficient $\varphi$: Measurements, CFD Simulations with Different Sizes of the Tip Gap and without Tip Gap. Note: Tip Gap Ratio Equals 0.43% in the Experiments.