Modelling of Axial Fans
for Electronic Equipment

J. Hennissen, W Temmerman, J Berghmans, K Allaert

1 Bridge Road
Hampton Court
Surrey KT8 9HH
UK

Telephone: +44 (0) 181 941 8810
Facsimile: +44 (0) 181 941 8730

MODELLING OF AXIAL FANS FOR ELECTRONIC EQUIPMENT

J. HENNISSEN[1], W. TEMMERMAN[2], J. BERGHMANS[1], K. ALLAERT[2]
[1] *K.U.Leuven, Dept werktuigkunde, Celestijnenlaan 300 A, B-3001 Heverlee (Belgium)*
[2] *Alcatel Bell, F. Wellesplein 1, B-2018 Antwerpen*

Abstract

The work presented concerns the modelling of a fan used for forced convection cooling of a subrack of printed board assemblies (PBAs) as a separate component for accurate CFD simulations.
Detailed measurements of the fan outlet velocity profile (axial, tangential and radial components) have been made for one fan operating point in a measurement set-up apart from the PBAs. CFD simulations have been made for a simplified version of a subrack, comparing a few elementary fan models.
Results show that incorporation of the hub into a model of the axial fan is necessary. The tangential velocity at the outlet is not negligible. The value of the radial velocity is small throughout the fan outlet plane and may be ignored in a CFD model.

1. Introduction

Detailed measurements of the fan induced velocity pattern which occurs in a printed board assembly (PBA) showed that this pattern is highly non - uniform, resulting in inadequate cooling of parts of the structure [1]. Due to the compactness of the assembly the lack of uniformity of the velocity profile at the fan outlet has important repercussions on the flow distribution over the different card 'channels'. CFD simulation of the flow can only be successful if it takes this non-uniformity into account.

A simulation which used detailed measurements of the velocity pattern upstream of the cards as input conditions, showed important progress in predicting the velocity patterns in the PBA assembly [2]. It is felt however that this approach can't be generalised, as every change of the configuration would require detailed measurements of a velocity profile, in order to make accurate CFD simulations possible.

The work presented here studies the feasibility of measuring the outlet velocity profile of the fan in a measurement set-up, apart from the PBA assembly, and using this velocity profile as input boundary condition for the CFD code, independent from the downstream equipment configuration.

2. Experimental set-up

The fan studied is an axial fan of the firm PAPST, type 4148XP, which has an outer diameter of about 120 mm.

The experimental set-up is based on existing set-ups for measuring fan characteristics. Some adaptation allows for the new tasks. Fans designed for use without duct are to be tested without ductwork [3]. Usually, chamber test methods are used for these kinds of tests : to combine the requirements 'free outlet' and adjustable pressure drop, a chamber is connected to the outlet of the fan. The pressure drop is controlled by use of a variable nozzle at the outlet of the chamber. Environmental disturbances of the flow at the inlet are avoided by providing an inlet chamber. An overview of the design is shown in figure 1.

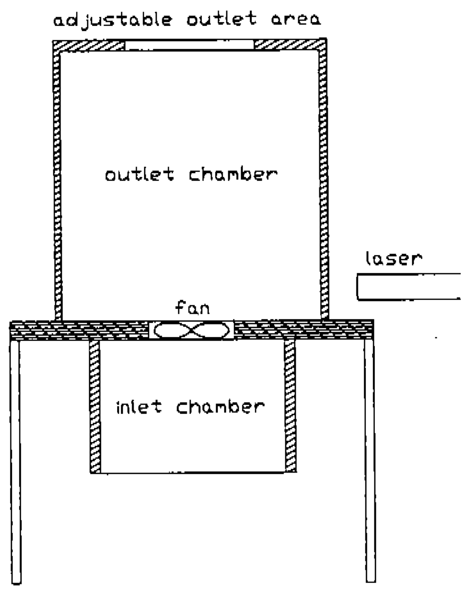

fig 1 : measurement set-up

To avoid influencing the flow profile with the 'outlet chamber', the chamber has to be sufficiently large. ANSI standard 210-85 specifies that the outlet chamber should have a cross-sectional area of at least sixteen times the area of the fan outlet for axial fans [3, 7.3.1]. For inlet chambers, it specifies that the cross-sectional area has to be at least five times the area of the fan inlet [3, 7.3.2]. An upper limit to the applicable flow chamber radius is set by the maximal focus distance of the LDV system used, which equals 600 mm. In accordance with these limits and with other practical limitations, the outlet chamber diameter is 800 mm, and the inlet chamber 600 mm.

3. Some features of the LDV measurement technique

Velocity measurements are performed with an LDV (Laser Doppler Velocity) measurement system. It consists of a SPECTRAPHYSICS 5 W all lines argon laser and a DANTEC 2D beam splitter and Flow Velocity Analyser. This system measures the velocity in a small volume formed by the intersection of two laser beams originating from the same source. The distance between the parallel beams is set at 60 mm and focusing optics are used with a focal distance of 600 mm.

The LDV measurement technique is based on the Doppler frequency shift of the light beam reflected by moving particles. It therefore requires the presence of tracer particles in the flow field. These particles must be light enough to ensure that they follow the air flow. On the other hand they have to generate sufficient reflection of the laser beam. Incense smoke particles prove to be a good solution for these two requirements.

A measurement consists in taking 500 accepted samples (measurement signals which satisfy a required signal to noise ratio), and calculating the mean value and the root mean square value of the velocity in two directions, perpendicular to the laser beam. A 3 D-traverse system allows to focus on different points in the flow field.

4. Results

Measurements of the fan outlet profile were made for one fan operation point. The chosen point is the one corresponding to the free (unobstructed) air flow condition. A comparison of velocity profiles measured with and without a test chamber surrounding the fan outlet showed only small differences, which validates the outlet chamber.

To obtain a fan velocity profile which is usable even for compact PBAs, the fan outlet profile has to be determined as close as possible to the fan outlet plane. The measurements in a plane with a downstream distance from the fan outlet plane of 7 mm satisfy this requirement. Measurements in two extra planes at downstream distances 27 and 47 mm were made. The (2D)-velocity profile was determined for two traverse directions (laser movement directions), perpendicular to each other. The traverse directions were chosen such that the first direction leads to values of the axial and radial velocities, and the second to values of axial and tangential velocities. This is sufficient to characterise the flow pattern in the whole plane, if axial symmetry holds.

In figure 2, a comparison is made of the measured velocities in the planes at distance z=7mm and z=47 mm. The difference between the velocity profiles is striking.
For the axial velocities (figure 2a and 2c), the peak values of the plane at position z = 7 mm downstream of the fan outlet are clearly higher than those at the plane z=47 mm. As could be expected, the velocity profile tends to flatten as the distance from the fan outlet increases. Yet the flattening process is not strong enough to cause a uniform flow at z = 47 mm. The effect of the hub is still clearly present here.

Axisymmetry does not hold : the velocity profile at the left side (front side) of the centre is different from the one at the right side (back side).

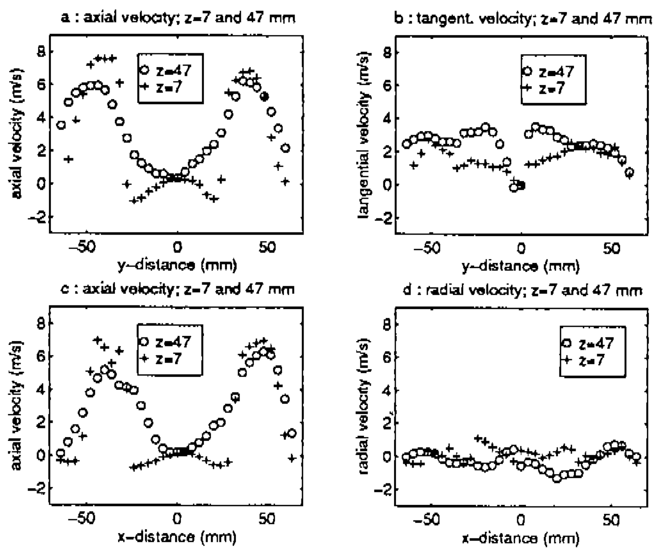

fig 2 : velocity profile evolution in downstream direction

For the radial velocities (fig. 2d), a profile is observable at z = 47 mm downstream of the fan, which is less at z = 7 mm downstream. Probably, recirculation flow causes these patterns. Nevertheless the absolute values of these velocities are low.

The tangential component of the velocities measured in the plane z = 7 mm downstream of the fan is smaller than the one at z = 47 mm (fig. 2b), although no swirl augmenting components are present downstream of the outlet. The explanation for this phenomenon will be given below.

Comparison of the results at z=27 mm with those at z=7 mm leads to similar conclusions.

The main geometrical deviation from axisymmetry is due to the presence of three studs (fan supports) at the outlet plane. Measurements in the plane z = 7 mm downstream at two traverse directions rotated over 120° with the fan centre as axis of rotation led to a significantly better agreement between the velocity profiles. This shows that the studs are the main cause of the asymmetry in the velocity profiles.

To obtain an overall view of the flow profile at the outlet of the fan, the velocity vector was measured in several locations in the plane 7 mm downstream of the fan outlet plane. These points form an orthogonal grid with point to point distances of 5 mm. To get a 3D-view of the velocity vector with a 2D measurement system, every location of the grid was tested two times (with the laser beam rotated over 90 °). This implies that the vertical component of the velocity was measured twice in every point. The grid points too far from the free outlet area (e.g. over the hub) were skipped, because, due to seeding problems in areas with zero velocity, it would take too long to obtain accurate values.

In figure 3, the measured axial velocities are shown as a function of the radial position only. The extended range of values for a given radius illustrates the fact that axisymmetry does not hold. A combination of radius and angle is necessary to describe the flow pattern at the fan outlet.

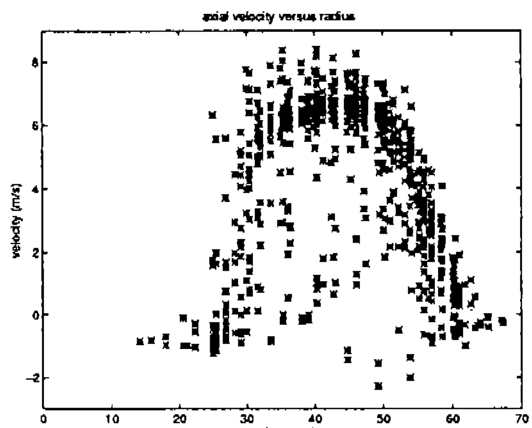

fig 3 : axial velocity as a function of radial position

To obtain a representative set of data for a given angle, the data obtained have been reprocessed with the aid of MATLAB. For a grid of points with radius 5,10, 15, ..., 65 mm and angle -180, -170, -160, ..., 170 an approximated value of the velocity in three directions (axial, tangential and radial) is calculated, based on the cartesian velocities of the measured grid, via an inverse distance interpolation method.

Figures 4a and 4d show the variation of the axial velocity as a function of the angle for two radii. Two lines per diagram are present : one per perpendicular traverse direction of the laser beam. As no data filtering or curve fitting has been done, the curves follow an erratic path, due to the error in individual measurements. Nevertheless some clear trends are observable. The axial velocity is nearly constant over all angles. The position of the studs (situated at angles -135, -15 and 105 °) leads to a sudden fall of the axial velocity. At both sides of the studs the axial velocity is locally higher than the mean axial velocity, and this phenomenon is more accentuated at the side corresponding to the positive fan rotation direction (which is the negative angle direction in the chosen coordinate system.). Apparently, the influence of the studs on the upstream flow pattern is small. The flow is deflected by the studs, resulting in a contraction of streamlines at both sides of the studs and a local velocity rise.

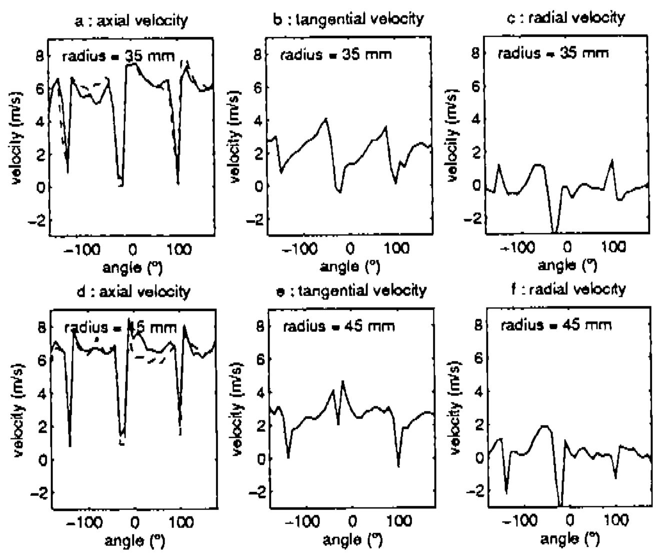

fig 4 : velocity as a function of angle for given radius

Figure 4b and 4e show the variation of the tangential velocity as a function of the angle for two radial positions. A saw teeth pattern is observable. At the studs, the tangential velocity falls to zero. Downstream of the studs (positive fan rotation direction, negative angle) the velocity increases rapidly, until a maximum of about 4 m/s is reached. This corresponds to the highest values of the tangential velocity measured in the measurements for the plane z=47 mm downstream of the fan (figure 3). At lower angles it decreases again, but less steeply.. .The formerly mentioned phenomenon that the tangential velocity seemed to increase with increasing distance from the outlet plane may now be explained : former measurements were made in a plane close to the studs. The studs produced a kind of shadow effect which cut down the tangential velocities at z=7 mm downstream of the outlet plane. This shadow effect didn't occur at z=47 mm downstream of the outlet plane.

Figure 4c and 4f show the variation of the radial velocity as a function of the angle for given radial positions. Apart from the neighbourhood of the stud angle, the value of the radial velocity is small. It is uncertain whether the contraction of streamlines or measurement inaccuracies due to beam reflection effects are responsible for the high local values near the studs.

5. CFD calculations using fan models with varying level of detail

In [1] LDV-experiments are reported on a PBA assembly. The assembly consisted of 2 x 2 fan units and 2 subracks with 9 PCBs each, mounted conforming to ETSI Standards. Two different PBA structures were tested, one using metal core PCBs and a second using standard PCBs with heat sinks mounted on the IC's. A front and side view of the test assembly is shown in figure 5.

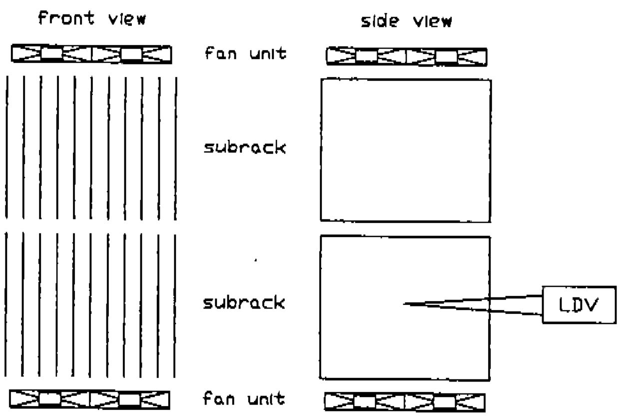

fig 5 : PBA assembly of [1]

Detailed measurements of the vertical velocity pattern are available for one card channel in 3 horizontal planes. One plane is located 21 mm above the bottom fan unit, the second plane is located at the lower edge of the lower subrack, the third at the top edge of the lower subrack (285 mm above fan unit).

Flotherm simulations have been performed on a structure similar to this assembly. The system modelled for CFD calculations considers only the lower subrack. The modelled structure is restricted to 5 card channels (6 PBAs) and one fan unit (two fans fixed in the bottom plate, which cover the bottom plate partially). Symmetry considerations allow the results of these simulations to be extrapolated to the real assembly. Distances between the different cards and position of the cards relative to the fan unit are chosen in accordance with the system considered in [1]. Individual heat sinks or turbulence generators are not considered. The turbulence effects caused by these components are accounted for by specifying increased surface roughnesses and an average height of components, based on the heat sink height. The cards contain also a DC/DC converter. The dimensions of this converter have been respected in the simulated structure, modelling this component as a flow blocking element.

To be able to calculate velocity distributions, and given the 3D nature of the geometry simulated, 4 variables are to be calculated essentially, being the pressure and the three cartesian velocity components. This asks for at least 4 differential equations to be solved. Turbulence modelling has been incorporated using the standard k-ε-model. In the k-ε-model, the (local) value of turbulent viscosity is calculated from the values of k (turbulence energy) and ε (rate of k-destruction, kinetic energy dissipation rate), which in their turn are calculated by means of two extra transport equations.

Temperature will be an essential variable to incorporate when heat transfer effects are to be considered. It is thought however that temperature effects will affect velocity distributions only in a limited way, given the forced flow character brought about by the fans. Therefore, the effect of temperature as an independent variable has been neglected. Properties of the fluid (laminar viscosity, density) are taken to be constant and are evaluated at absolute pressure 1 bar and temperature 20 °C.

Two fan models were compared.
In a first model, the fan is represented as a rectangular element on the bottom plane of the structure. A uniform velocity profile is established, perpendicular to the plane in which the fan is situated. The dimensions of the fan were set so as to give an area equivalent to that of the actual circular fan. The mass flow entering the structure is fixed to 0.05 kg/s. The value of k (turbulence energy) for the entering mass is fixed to 1.7 m/s. This value showed to be an acceptable averaged value according to the measurements.
For ε (kinetic energy dissipation rate), no measurement results are available. Developers of the Flotherm software code advise following relation, which showed to be applicable to developed flows :

$$\varepsilon_{in} = 0.1643 \, . \, k^{3/2}_{in} \, / \, L_{in} \, ,$$

where k_{in} is the formerly specified turbulence energy at the inlet, and L_{in} one tenth of the diameter of the fan diameter. This leads to a value for ε_{in} of 32.9 m^2/s^3.
In a second model, an element was added to the fan of the first model, which blocks the flow over part of the plane, representing the hub. The total flow is now applied over the unblocked part of the surface. Swirl has also been added, by specifying a (constant) tangential velocity over the unblocked part of the fan surface, being 3 m/s.

Calculations were performed using a calculation grid with cell to cell mean distances of about 5 mm. Some deviation from this refinement option was necessary, due to the geometrical properties of the structure (structural boundaries have to match the underlying grid). Careful choice of the grid refinement between two key-points (anchor points of the grid, matching the structural boundary locations) resulted in smooth cell size variations, which are important to obtain accurate results. The resulting grid contains 55 x 35 x 57 cells.

Figure 6 compares the simulation results with the measurements obtained in [1].

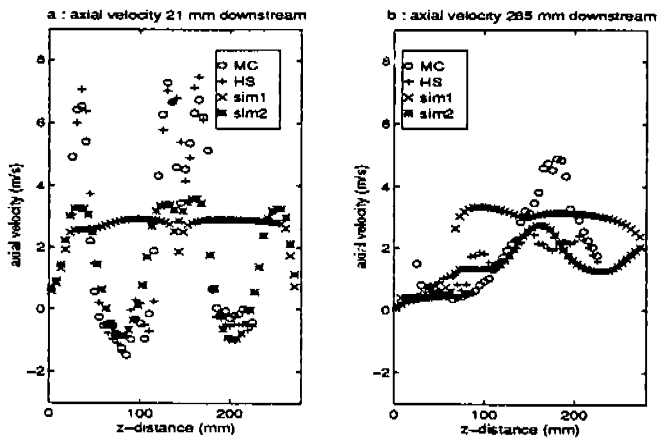

fig 6 : comparison of measurements and simulation results

Figure 6a gives a view of the axial velocity profiles measured or calculated in horizontal plane 1, which is located 21 mm above the bottom fan-unit and 12 mm below the card boundaries.
The velocity profiles measured in plane 1 for two card configurations, one with heat sinks (symbol '+') and one with metal core (symbol 'o') are about the same, an illustration of the fact that the influence of downstream conditions on the velocity profile near the fan outlet is of minor importance.
The velocities calculated in simulation 1 (symbol 'x') lead to a flat profile, which clearly does not correspond to reality. The axial velocity profile calculated in simulation 2 (symbol '*') gives better agreement, but the peaked character of the real system is not predicted, due to the lack of agreement between the real fan geometry (circular) and the modelled one (rectangular). This may cause inaccurate predictions in the mass flow distribution over different card channels. As far as a correct flow distribution is important, which normally will be the case, a more accurate description of the fan geometry is necessary.

Figure 6b gives a view of the axial velocity profiles measured or calculated in horizontal plane 2, which is located near the top edge of the lower subrack (285 mm above the fan unit). Agreement between flow simulation 2 and measurement results for the heat sink geometry is improved, due to the damping effect of flow resistances on the velocity profile non-uniformities : wall friction and turbulent mixing cause non-uniformities at the fan outlet to disappear at rather small distances downstream. Velocity profile details at the fan outlet are of minor importance to obtain an accurate velocity profile at large distance from the fan, as far as the mass flow distribution in the different card channels is accurate.

6. Conclusions

For the axial velocity at the outlet of a typical axial fan used for the cooling of PBA assemblies, an important deviation from a uniform velocity profile at the outlet is present, mainly due to the presence of a hub in the axial fan. Axisymmetry isn't fullfilled, the velocity variations for a given radius are large. The three studs (fan supports) at the outlet plane are found to be the main cause of this asymmetry. Due to the regular positioning of the studs, an angular symmetry approximation, in which the velocity pattern is repeated over 120°, is in agreement with measurements.
To describe the axial and especially the tangential velocity profile of a fan accurately, a model which takes radius AND angle into account is necessary. The studs function as a local flow blockage. For the axial velocities, this results in a flow profile which is approximately flat, apart from a small region near the studs, where a velocity peak balances the region of no flow above the studs. For the tangential velocity a saw tooth profile occurs, with a zero value and a peak value in the neighbourhood of the studs in the fan rotation direction. For the radial velocities the deviations from zero velocity are small.

To obtain an accurate velocity profile at large distance from the fan, the velocity profile details at the fan outlet are of minor importance. An accurate description of the velocity profile at the fan outlet nevertheless will be necessary to obtain a correct mass flow distribution over the different card channels. Results show that incorporation of the hub into a model of the axial fan is necessary. Taking the tangential velocity at the outlet into account is also important to be able to predict heat transfer coefficients. A fan model using constant velocities at the free outlet section of the axial fan is acceptable but the geometrical representation of the fan in the simulated CFD structure should be as accurate as possible, given the coarseness of the grid.

7. References

[1] H. BRUNEEL, B. BEERNAERT, G. MORTIER, J. DECLERCQ, B. BOESMANS, W. TEMMERMAN, W. NELEMANS, E. LAUWERS, "Supporting experiments for CFD based thermal design of telecommunication equipment", Eurotherm Seminar no. 29, Delft, june 1993

[2] W. TEMMERMAN, W. NELEMANS, T. GOOSSENS, "Air flow modelling requirements nearby fans.", Flotherm user's conference, Boston, may 1993

[3] ANSI/AMCA 210-85,ANSI/ASHRAE 51-1985, "American National Standard : Laboratory methods of testing fans for rating.", 1985