Modelling and simulation of a jet fan for controlled air flow in large enclosures

Citation for published version (APA):

van de Giesen, B. J. M., Penders, S. H. A., Loomans, M. G. L. C., Rutten, P. G. S., & Hensen, J. L. M. (2011). Modelling and simulation of a jet fan for controlled air flow in large enclosures. *Environmental Modelling and Software*, 26(2), 191-200. <https://doi.org/10.1016/j.envsoft.2010.07.008>

DOI:

[10.1016/j.envsoft.2010.07.008](https://doi.org/10.1016/j.envsoft.2010.07.008)

Document status and date:

Published: 01/01/2011

Document Version:

Accepted manuscript including changes made at the peer-review stage

Please check the document version of this publication:

- A submitted manuscript is the version of the article upon submission and before peer-review. There can be important differences between the submitted version and the official published version of record. People interested in the research are advised to contact the author for the final version of the publication, or visit the DOI to the publisher's website.
- The final author version and the galley proof are versions of the publication after peer review.
- The final published version features the final layout of the paper including the volume, issue and page numbers.

[Link to publication](#)

General rights

Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

- Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
- You may not further distribute the material or use it for any profit-making activity or commercial gain
- You may freely distribute the URL identifying the publication in the public portal.

If the publication is distributed under the terms of Article 25fa of the Dutch Copyright Act, indicated by the "Taverne" license above, please follow below link for the End User Agreement:

www.tue.nl/taverne

Take down policy

If you believe that this document breaches copyright please contact us at:

openaccess@tue.nl

providing details and we will investigate your claim.

Modelling and simulation of a jet fan for controlled air flow in large enclosures
B.J.M. v.d.Giesen, S.H.A. Penders, M.G.L.C. Loomans*, P.G.S. Rutten and J.L.M. Hensen
Eindhoven University of Technology, Eindhoven, The Netherlands¹

Abstract

Jet fans are applied for control of air flow and support of pollutant dispersal in large enclosures. In The Netherlands, application is well known for car parks as part of the fire safety design. In the design phase often the Computational Fluid Dynamics (CFD)-technique is used to verify the fire safety level afforded by the application of jet fans. However, very little is known on the modelling requirements of jet fans in CFD. This includes paucity on experimental data that can be applied for validation of the jet fan model. In this study results are presented of measurements for a specific type of jet fan in a large enclosure and of validation of modelling characteristics of the jet fan in CFD. Both free and near-wall positioning have been investigated and a modelling proposal is made. For the modelling of the jet fan, point-of-departure is a low complexity, as the model generally will have to be included in large computational domains. Applicability of the developed approach and assessment of efficiency of jet fan positioning in large enclosures is shown through a case study.

Keywords

Jet fan, measurements, CFD, airflow, large enclosures, fire safety

¹ The work described here was performed while the first two authors were affiliated to the Eindhoven University of Technology. B.J.M. v.d. Giesen is currently affiliated to Cauberg-Huygen, ‘s Hertogenbosch (The Netherlands), S.H.A. Penders is currently affiliated to Steunpunt Dubolimborg, Heusden-Zolder (Belgium).

Introduction

Control of airflow in large enclosures, for example an atrium or a large industrial hall, is not straightforward. An overview of principles and conditions for assessing the airflow in such enclosures is presented in IEA (1998). In large enclosures with airborne pollution sources, such as in car parks and tunnels, sufficient ventilation with outside air is important in order to reduce the concentration of hazardous substances to a harmless level. In the Netherlands, jet fans often are applied in large enclosures to support this process. These jet fans are mounted in the enclosure and generate a momentum. This causes induction of air and promotes mixing and transport of the polluted air. Jet fans often are used in situations where large amounts of air need to be transported with a relative high velocity, for example in car parks and tunnels.

The throw of a jet fan can be characterized as a jet. An important distinction is made between a free jet and a near-wall jet. A free jet in this case is assumed to be positioned within an extensive enclosure where the effect of the walls on the throw of the jet, near the jet, is minimized. For a near-wall jet, the jet is positioned at relative short distance from one wall and may be influenced by its presence. These definitions differ somewhat from the type of wall-based jets explained by Awbi (1991) and Regenscheit (1984) as the induction of air is not obstructed by the wall in which the supply is mounted.

Awbi and Regenscheit provide several empirical relations to derive jet characteristics, such as the maximum velocity reduction and the volume flow rate of the induced air. The throw characteristics of a jet fan may be determined with these empirical relations. Measurements however are required to fit the unknown variable for a specific jet fan. The empirical relations apply for undisturbed jets with no obstructions present in the throw. In practice, however, obstructions will be present and disturb the jet profile. This means that empirical relations are of limited use for jet fans.

In practice jet fans often also will be positioned parallel to each other to improve the control of air movement in a large enclosure. When two jets are positioned parallel they will continue to work independently (Katz 1989). If the throw of the jet fans however is close to each other one combined jet will develop at a certain distance from the supply. With similar jet fans the combined jet will develop

centrally between the original throw of the jet fans. Otherwise, as a result of the coanda-effect, the larger jet will dominate the position of the combined jet.

A jet fan consists of a tube in which an axial fan has been placed. As a result on one side air is drawn into the tube and on the other side the air is blown into the enclosure. Specifications for a jet fan are based on the combination of the fan and the tube. These jet fans generally are applied at one operating point, with high exhaust velocities (in the order of 20 m/s). Although jet fans have a wide application range in practice, little is known on options for design optimization, for example, aiming at high effectiveness with a minimum number of jet fans. The influence of, for example, a pressure difference over the jet fan (combination of fan and tube) on this effectiveness has not been quantified extensively as well.

Jet fans frequently are applied as a component in the control of heat and smoke distribution in car parks (see Figure 1). In The Netherlands, the Computational Fluid Dynamics (CFD)-technique is used in practice in the design phase to assess performance of these fans in a fire situation (Loomans et al. 2009). Several methods are available to represent a jet fan in a CFD model (Nielsen 1989, Chen and Srebric 2000; Hegeman 2008). Generally, the modelled jet fan is represented relatively simple through Dirichlet boundary conditions. Only a limited number of measurement data however is available to verify this simplified approach. For example, measurements for a scaled jet fan model in a wind tunnel are described by Mutama and Hall (1996). But these measurements focus on tunnel ventilation application. For a jet fan in a large enclosure only the results of van Oerle et al. (1999) may be applicable. However, in that case not all boundary conditions are available.

Alternatives for modelling of such jets with CFD may be found in for example subzone models (Stewart, 2006). However, for complex enclosures such alternatives are not feasible because of the partly empirical origin.

In summary, jet theory for air flow in enclosures concentrates on wall-based jets. In literature only a very limited amount of measurement data for jet fans is available. For these data, boundary conditions are not (completely) known. As a result the data cannot be used to validate CFD-models of such jet fans.

Following the above, the main aim of the research described below was to provide additional measurement data for a jet fan and to provide jet fan modelling information for use in CFD. This information should allow for a better design assessment of the application of this type of jet fans. In addition utilization of the results is shown by means of an example design study. In this study jet fan positioning and its boundary conditions for use in a car park type of enclosure have been investigated. Point-of-departure for the jet fan CFD modelling is that the complexity of the jet fan model should be low. Application of jet fans in large enclosures, like for example a car park, generally already will put stress on the modelling.

Methods

Jet fan modelling in CFD was developed in two steps. First, available measurement data of a jet fan in a hangar (TNO case; van Oerle et al. 1999) have been used for setting up the jet fan model in CFD. As not all boundary conditions for the mentioned case were known new measurements have been performed. The aim was to obtain data for further validation purposes. After validation of the jet fan model, the model has been used in a design parameter study. This section describes the measuring configuration and conditions and some aspects of the CFD-modelling of the jet fan. The new measurement data and validation of the CFD-model with these measurement data is described in the Results section. The design study is presented in the Discussion section.

Measurements have been carried out in a large enclosure at the premises of the Eindhoven University of Technology. The hall in which the jet fan was placed has a length of 34m, a width of 32m and a height of 6.5m. The exhaust of the jet fan was positioned at a distance of 10m from a partition that obstructed a part of the width of the hall. The remaining part can be assumed as infinite. In the CFD model this side of the hall is assumed infinite over the whole width of hall. The maximum allowable throw of the jet is 24 m. Above a height of 2.3 meter the length of the hall in the direction of the throw can be assumed infinite. In the CFD model, according to the above described geometrical principles, over the width of the hall

pressure boundaries are assumed. The remaining sides (incl. floor and ceiling) of the hall are defined as a standard wall (smooth).

In the measurements a Novenco jet fan has been used. The diameter of the applied jet fan is 290 mm, the length is 2.6m. The operating specifications are summarized in Table 1. All measurements have been carried out at the operating conditions as indicated in Table 1, which will be its actual application mode in case of emergency. For the measurement of the free jet, the jet fan centre has been positioned at a height of 2.5m (Figure 2 and 3). For the measurement of the wall jet the jet fan centre has been positioned at a height of 0.2m from the floor.

Air velocity and temperature measurements have been performed for several cases in order to study the jet development of a free jet, a wall jet and a wall jet with deflector. The function of the deflector is to change the direction of the exhaust flow of the jet. The deflector angle was 8° directed upward from the floor. In addition, measurements have been performed where an obstruction (height 0.6 m, depth 0.6 m) has been placed at 3m or 6m from the exhaust, perpendicular to the throw and over the full width of the throw.

For the identified cases measurements have been performed at a predefined grid. At distances of 0.5, 1, 2, 4, 8, 12 and 16m from the exhaust measurements have been performed at a grid in the y-z plane (parallel to the exhaust) in order to determine the throw of the jet. This means that at larger distances from the exhaust the measurement plane increases. As for each plane a maximum of 66 measurement points have been investigated the measurement grid was coarsened (distance: 0.5m and 1m: measurement area 1.0m wide $\times$ 0.5m high; distance: 2 and 4m: measurement area 2.0m wide $\times$ 1.0m high; distance: 8, 12 and 16m: measurement area 4.0m wide $\times$ 2.0m high). For practical reasons, for the free jet case the lower half of the throw was measured. The measurements have been tested on reproducibility. For velocities >1.0 m/s, the relative difference was smaller than 10% on average.

Air velocity measurements have been performed with Schmidt sensors. Prior to the measurements these sensors have been calibrated in a wind tunnel for a velocity range of 0 up to 20 m/s. The inaccuracy of the sensors is on average <0.25 m/s for the velocity range 0-7.5 m/s and <0.3 m/s for the velocity range

7.5-20 m/s. The response time of the sensor was 1 sec according to the specifications. The measurement interval for each measurement position was 5 minutes with a frequency of 1 Hz. From the data the average velocity and the turbulence intensity have been derived. The measured turbulence intensity is obtained from Equation 1:

$$TI = 100\% \cdot \frac{\sqrt{v'^2}}{\bar{v}} = 100\% \cdot \frac{\sigma_v}{\bar{v}} ; \quad \text{Equation 1}$$

Where TI [%] is turbulence intensity and $\bar{v}$ [m/s], mean velocity, and v' [m/s], fluctuating velocity, follow from Reynolds decomposition of the instantaneous measured velocity (v [m/s]). σ_v [m/s] is the standard deviation. The influence of the response of the velocity sensor was tested by comparing the measured TI with a sensor with a specified response time of 0.1s.

The temperature measurements showed that the measured cases can be considered as isothermal.

For the CFD-simulations the CFD-code Fluent version 6.2.16 (Fluent Inc 2005) has been used to solve the Navier-Stokes equations (Versteeg and Malalasekera 2007). Gambit (Fluent Inc 2005) has been used for the geometrical modelling and discretisation. Grid sensitivity analysis has been performed to guarantee grid independent results. This analysis has been carried out for a 2D problem with dimensions of 40×80m based on the case measured by van Oerle et al. (1999). The jet has dimensions of 0.3×3.0m, with the centre of the exhaust positioned at $x = 12\text{m}$ and $y = 20\text{m}$. Grid refinement has been applied around the jet fan. Based on this analysis with the 2D model, minimum grid requirements have been derived for which no grid sensitivity was found in the simulated velocity profiles at several distances from the exhaust. These grid requirements have been applied in the 3D models that have been developed as part of this study. The number of grid cells for the 3D CFD-models of the different cases examined in the measurement setup was minimum 166,000 and maximum nearly 225,000 cells. In all models a structured grid has been applied.

To determine minimum requirements for the modelling of the jet fan, numerical detail studies have been performed with respect to the form of the opening area and the requirements for uniformity of the supply conditions. Both for conditions representative for the investigated jet fan. The cylindrical exhaust

1
2
3
4 of the jet fan was simplified to a square opening with a fixed average air velocity. Figure 4 shows that the
5
6 original (square) form of the modelled jet was not recognizable anymore at a distance of one meter from
7
8 the exhaust. It transformed to the characteristic round form of a cone, indicating that simplification to a
9
10 square opening is valid for assessing flow conditions from 1 meter downstream and further. On the supply
11
12 side to the jet fan also a fixed velocity was prescribed. Given its minimum effect on the flow field,
13
14 simplification to a square area may be assumed here as well.
15
16

17 The velocity profile at the exhaust of the actual jet fan is not uniform. Therefore, also the influence of
18
19 the velocity profile at the exhaust on the jet development has been examined numerically. A situation is
20
21 compared with an average exhaust velocity of 18 m/s. In one case the exhaust velocity is assumed uniform
22
23 over the opening area, in the other situation the velocity at the edges is assumed higher. The latter velocity
24
25 profile resembles the measured supply conditions for the examined jet fan. Figure 5 shows a comparison
26
27 of the simulated velocity profiles at a distance of 0.25m and 1.0m from the exhaust. This result shows that
28
29 at a distance of one meter the impact of the form of the profile has been levelled out. The velocity profile
30
31 is similar to the profile with a uniform exhaust velocity. This result indicates that application of a
32
33 (relative) simple uniform velocity profile in the numerical model of the jet fan is valid for flow field
34
35 assessment at one meter downstream of the jet fan opening and further.
36
37
38
39

40 The modelling approach chosen, i.e. representation of the jet fan through a fixed velocity at the exhaust
41
42 of the jet fan, implies that the jet fan characteristics are simplified to one operating point. In practice and
43
44 in the results presented the pressure differences over the jet fan are one order of magnitude smaller than
45
46 the fan static pressure at the operating point.
47
48

49 All presented CFD results have been solved stationary and isothermal. All equations have been solved
50
51 with 2nd order accuracy. Given the intention for reduced modelling complexity, Reynolds averaging is
52
53 assumed. For turbulence modelling several two-equation turbulence models have been compared
54
55 (standard k- ϵ , RNG k- ϵ , realizable k- ϵ and standard k- ω). A detailed description of the characteristics of
56
57 these turbulence models as implemented is given in Fluent Inc. (2005). The comparison has been carried
58
59 out for the case as measured by van Oerle et al. (1999), i.e. a hangar with a free jet. Furthermore, a
60
61
62
63
64
65

comparison with the theory as described by Regenscheit (1985) was made. For the latter, Equation 2 has been applied:

$$\overline{v_{\max,x}} = \overline{v_{\text{exh}}} \cdot \left(\frac{d}{m \cdot x} \right); \quad \text{Equation 2}$$

Where $\overline{v_{\max,x}}$ [m/s] is the maximum average velocity at distance x [m] from the jet fan exhaust opening, $\overline{v_{\text{exh}}}$ [m/s] is the average exhaust velocity from the jet fan, d [m] the diameter of the jet fan opening and m [-] is the mixing number, which is determined experimentally.

The results for this comparison are shown in Table 2. The assessment is mainly based on the maximum average velocity at several distances from the exhaust. In addition in Table 2 the average difference between the simulated velocities and theory and the TNO case are shown. Of the investigated turbulence models, the standard $k-\omega$ model performs worst. This model should have best performance for wall-bounded flows (Fluent Inc. 2005). The RNG $k-\epsilon$ and Realizable $k-\epsilon$ model are improved variants of the standard $k-\epsilon$ model. With main differences in the modelling of the turbulent viscosity and the calculation of the dissipation rate (Fluent Inc. 2005), in principle, these improved variants should outperform the standard $k-\epsilon$ model. However, for the investigated type of flow problem this is not the case. Zhang et al. (2007) show that the performance of specific turbulence models can depend on the type of flow problem for which it is applied. Following the comparison, the standard $k-\epsilon$ model was chosen as turbulence model for the further evaluations. Where applicable, standard wall-functions have been used.

Results

Measurements

The measured velocity and total volume flow rate for the investigated cases are presented in this section. The induced flow rate can be derived by subtracting the exhaust volume flow rate of the jet fan. In all cases this flow rate is $1 \text{ m}^3/\text{s}$.

Figure 6 presents the measured average velocity profiles in m/s for the free jet at a distance of 0.5m; 2m; 4m and 8m from the exhaust. With increasing distance from the exhaust of the jet fan the average

1
2
3
4 velocity decreases and the width of the flow increases. From the measurement results the total volume
5
6 flow rate can be derived. In Table 3 the values for the volume flow rate are summarized. The volume flow
7
8 rate for all cases is determined for the area of the measurement grid. For the free jet case the measurement
9
10 grid only covered the lower half of the throw of the jet, the actual total flow rate therefore will be
11
12 approximately twice as large than presented in Table 3.
13
14

15 Figure 7 presents the measured average velocity of the wall jet case at a distance of 0.5m, 2m, 4m and
16
17 8m from the exhaust. Table 4 presents the maximum average velocity and the total volume flow rate at
18
19 several distances from the exhaust of the wall jet. For this case the measurement grid encompasses the
20
21 total throw of the jet, certainly at short distances from the exhaust (see Figure 7).
22
23

24 Similar results have been obtained for the wall jet with deflector and for the cases of a wall jet with
25
26 obstruction. Figure 8 presents the measured average maximum velocity as a function of the distance to the
27
28 exhaust. For the wall jet with obstruction, measurements have only been performed before the obstruction.
29
30 For the wall jet with deflector measurements were taken up to 12m from the exhaust. In Figure 9 the total
31
32 volume flow rate of the jet as a function of the distance to the exhaust of the jet fan is presented. For the
33
34 free jet the presented flow rate is only half of the total flow rate, due to the measurement constraints. Next,
35
36 in the description of the simulations results, attention is put on the measurements for the free jet and the
37
38 wall jet.
39
40
41
42
43

44 ***Simulations***

45
46 An example result for the velocity contour field and the pressure contour field of the CFD simulation
47
48 for the free jet is shown in Figure 10. In Figure 11 the simulation results for the free jet have been
49
50 compared to the measurement data and the theory of Regenscheit (1984; see Equation 2, mixing number
51
52 0.24). In Figure 11a the maximum average velocity as a function of the distance to the exhaust is
53
54 compared. In Figure 11b the volume flow rate for the measured grid is compared. Finally, in Figure 11c
55
56 the measured and simulated turbulence intensity have been compared as a function of the distance to the
57
58 exhaust of the jet fan. The simulated turbulence intensity is determined from Equation 3:
59
60
61
62
63
64
65

$$TI = 100\% \cdot \frac{\sqrt{2k/3}}{\bar{v}}; \quad \text{Equation 3}$$

Where TI [%] is turbulence intensity, k [m^2/s^2] is turbulent kinetic energy and $\bar{v}$ [m/s], mean velocity.

In Figure 12 the simulated result for the wall jet has been compared with measurements. This comparison shows that the simulated maximum velocity is overestimated, indicating that the induction is too small. Therefore the wall jet fan model has been adapted to compensate for this effect. The opening has been divided horizontally in three parts (0.09m, 0.08m and 0.09m width; see Figure 13). Following a calibration with the available measurement data, at the outer parts of the opening the velocity vector has been directed at a horizontal angle of 12.5° from the vertical jet fan symmetry plane. In the middle section the flow was directed perpendicular to the jet fan exhaust area. The comparison of this improved model with the measurements is shown in Figure 14a. In this Figure also the result from the original representation is included. Figure 14b gives a comparison of the improved model with the measurements for the volume flow rate. Similarly in Figure 14c the measured and simulated turbulence intensity are compared.

Discussion

Measurements, simulations

The measurement results show that the throw of the free jet is not fully axi-symmetric. This was also visible through visualizations of the jet with smoke. The explanation for the deviation is found in the influence of the surroundings, particularly at larger distance from the jet, when the partitions in the hall may have an effect. The use of a deflector increases the induced flow rate at shorter distances from the jet fan. However, this effect is diminished at a larger distance from the jet fan, indicating that the coanda-effect is present in this configuration. The effect of the obstruction on the upstream velocity and flow rate conditions is small in comparison to the undisturbed wall jet results.

1
2
3
4 In the comparison between measured and empirically derived maximum velocities for the free jet
5
6 similar velocities are found. This result however relates closely to the mixing number applied, which was
7
8 calibrated with the measurement data.
9

10
11 Point-of-departure for the simulations was to apply a relative simple model with which agreement with
12
13 the measurement data is good (not quantified). Simplicity should be strived for as the jet fan model will be
14
15 applied in models of large spaces.
16

17
18 The comparison of the simulation results with the measurement data shows a good agreement for the
19
20 velocity and the total volume flow rate. Average difference in the measured and simulated velocity is
21
22 0.2m/s (average value of absolute difference: 1.0m/s). Average difference in the measured and simulated
23
24 total volume flow rate is 0.0m³/s (average value of absolute difference: 0.1m³/s). However, as Figure 11
25
26 shows, for the turbulence intensity large differences are found. Average absolute difference in the
27
28 measured and simulated turbulence intensity is 14% (range 3% - 20%). This large difference firstly results
29
30 from the boundary condition that has been set for the jet fan exhaust. From the measurements a turbulence
31
32 intensity of 1% at the exhaust was determined. In a range from 1% to 30%, for best agreement with the
33
34 measured velocity and flow rate a turbulence intensity of 10% was required at the exhaust opening
35
36 (characteristic length scale 0.025m). The velocity sensor applied, also the one with a relative high
37
38 response time (0.1 s) that was used for comparing the measured turbulence intensity with, may have
39
40 resulted in an underestimation of the actual turbulence intensity due to damping. As shown by Loomans
41
42 and van Schijndel (2002) this damping can affect the response for this type of sensors by an order of
43
44 magnitude. This effect may also account for the other measurement positions.
45
46
47

48
49 Furthermore, the high value for the turbulence intensity in the throw of the jet is explained by the use
50
51 of the standard k-ε turbulence model. Large velocity gradients lead to an overestimation of the turbulent
52
53 diffusion. Nevertheless, this solution was chosen because of the simplicity of the approach. As an
54
55 alternative, for example, a swirl boundary condition could have been applied. This however has not been
56
57 tested. Emphasis was put on the induced air volume as a component of the control of the air flow in large
58
59 spaces.
60
61
62
63
64
65

For the wall jet a similar treatment as for the free jet did not result in a good agreement. The spread of the jet was smaller compared to the measurements. This is in line with results from Schälin and Nielsen (2004). Instead of referring to a more complex turbulence model as proposed by Schälin and Nielsen, the spreading of the jet parallel to the horizontal face (wall) was increased by splitting up the exhaust in three parts. For the outer openings the air flow was introduced under an angle. An outward angle of 12.5° resulted in best agreement with respect to the measured velocities and volume flow rate. For the turbulence intensity the same remarks are in place as for the free jet.

Application

The simplified model of the jet fan has been applied for a theoretical case that deals with a large space (100x28x3m) similar to a car park in which a wall jet is applied. The large space has unrestricted supply and exhaust conditions at the short sides of the large space. Zero pressure boundary conditions have been used for these openings. Figure 15 gives an impression of the modelled space. For modelling and discretisation the information from the validation study was used. For the discretisation at least 1,000,000 cells have been used.

An optimization study has been performed into the optimal distance between jet fans (between 3.5 and 28m.; i.e. 1 up to 8 jet fans for the space examined) in combination with the influence of several boundary conditions. In this theoretical study the jet fan is assumed to be optimised for the prescribed boundary conditions. Following specifications of jet fans used in practice, pressure differences in the flow field can be assumed to be at least one order of magnitude smaller than the fan static pressure at the operating point. The results have been assessed on three performance indicators: (1) the average velocity at the exhaust side of the space (representative for the induced flow rate); (2) the minimum velocity (a negative value means a reverse flow direction towards the supply opening of the space) and; (3) the uniformity (defined as the minimum velocity divided by the average velocity; minimum value: 0.0). The minimum velocity and uniformity have been assessed at several distances from the jet fan exhaust. For all indicators a higher value represents a better performance. In the assessment the weight of the performance indicators was assumed equal.

1
2
3
4 As an example, information for the investigated cases with a constant total flow rate is summarized in
5
6 Table 5. Here, the jet fan exhaust area for each fan has been kept the same, resulting in higher exhaust
7
8 velocities when a smaller number of fans is applied in order to arrive at the same total flow rate. For these
9
10 cases the simulated results for the minimum velocity and uniformity as function of the x-position in the
11
12 space are shown in Figure 16.
13
14

15 The assessment results for the performance indicators (average velocity, minimum velocity and
16
17 uniformity) for the individual cases are summarized in Table 6. Table 6 also presents the performance
18
19 ranking to determine the optimum number of jet fans for the modelled space. For this ranking, for each
20
21 performance indicator the average result for the different cases is taken as reference. Values around this
22
23 average with a deviation up to 10% have been assessed as 'Neutral'. The category 'Reasonable' has a
24
25 band width between +10% and +30% above the average and the category 'Good' refers to > 30% above
26
27 the average. The categories 'Moderate' and 'Bad' have a band width from respectively 10% to 30% and >
28
29 30% under the average. For the minimum velocity the result of the case with one (1) jet fan is left out of
30
31 the averaging procedure.
32
33
34

35 Overall assessment of the three performance indicators leads to the conclusion that a design with 3 to 4
36
37 jet fans, i.e distance of approximately 7 to 9m between the jet fans, results in the best performance. The
38
39 presented example assumes a constant total flow rate. Additional sensitivity studies show that, for the case
40
41 with 4 jet fans (jet fan distance 7m), a higher momentum at constant total flow rate gives best overall
42
43 performance (for the three performance indicators defined) . The momentum was increased by reducing
44
45 the exhaust area and increasing the exhaust velocity. At a constant total momentum, independent of the
46
47 number of jet fans, the highest average velocity is calculated when 3 jet fans are used (jet fan distance
48
49 approximately 9 m). The momentum is kept constant by reducing the exhaust velocity when the number of
50
51 jet fans increases. The exhaust area for the individual jet fans remains the same.
52
53
54

55 The influence of a pressure difference between the supply and exhaust in the space is significant. For
56
57 the case with 4 jet fans at a positive pressure difference larger than 5 Pa the added value of the jet fans is
58
59 limited. At a negative pressure difference larger than 1 Pa the jet fans already cannot maintain the desired
60
61
62
63
64
65

1
2
3
4 air flow in the space. For the applied boundary conditions of the jet fan these pressure differences are well
5
6 below the fan static pressure level at the operating point of a typical jet fan used in practise. The
7
8 functioning of the jet fans therefore is critical in situations where such (not controlled) pressure
9
10 differences can arise.
11

12
13 Limiting the flow area in the space (i.e. a height of 1.5 m), for example because of the presence of cars,
14
15 affects the pressure field as well and results for the investigated case with 4 jet fans in a nearly 40%
16
17 reduction of the induced volume flow rate. Given its effect on the pressure field, the effectiveness of a jet
18
19 fan therefore is also strongly related to the geometry in which it is applied.
20
21

22 23 24 **Conclusions**

25
26 Velocity measurement data are provided for a typical jet fan as applied for smoke removal in car parks
27
28 in the Netherlands. Data are made available for a free jet, a wall jet, a wall jet with deflector and a wall jet
29
30 with obstructions. Based on the measurement data and modelling study a relative simple (low complexity)
31
32 CFD model of the free jet fan has been proposed and validated. This model is capable of representing the
33
34 throw and induced flow rate by such a jet fan. For the wall jet case a more detailed model is proposed
35
36 which improves the spread of the flow parallel to the wall. In both cases no agreement was found between
37
38 the measured and calculated turbulence intensity. In the experiments and the application example
39
40 emphasis was put on the induced air volume as a component of the control of the air flow in large spaces.
41
42 The investigated performance indicators therefore mainly address the velocity and flow rate. As a result
43
44 the inconsistency is rated less important. However, for other type of performance indicators which address
45
46 for example pollutant dispersal this assumption is not valid. In that case a more detailed modelling (for
47
48 example boundary condition definition or turbulence modelling) of the jet fan exhaust may be required.
49
50

51
52 The case study shows the applicability of the developed simple jet fan model and allowed for design
53
54 conclusions/ guidelines. However, the impact of the geometry and boundary conditions, effectuated in
55
56 pressure field changes, is significant and therefore generalization of these results is not straightforward.
57
58
59
60
61
62
63
64
65

Furthermore, measurements and validation were only performed for one jet fan. In practice more than one jet fan will be applied and they will interact with each other.

Acknowledgement

Novenco B.V. (Bergschenhoek, The Netherlands) is kindly acknowledged for the use of the jet fan that was applied in the experiments.

References

- Awbi H.B., 1991. Ventilation of Buildings. First edition, E & FN Spon; Chapman & Hall: London.
- Chen, Q., Srebric, J., 2000. Simplified diffuser boundary conditions for numerical room airflow models. Final Report for ASHRAE RP-1009, 181 pages, Department of Architecture, Massachusetts Institute of Technology: Cambridge, MA.
- Fluent Inc., 2005. Fluent 6.2 User's Guide. Fluent Inc: Lebanon.
- Hegeman S.T.G.D., 2008. Smoke movement in fire situations - CFD-utilization in car park Fleerde. M.Sc. thesis, Eindhoven University of Technology: Eindhoven, The Netherlands.
- IEA., 1998. Ventilation of Large Spaces in Buildings - Analysis and Prediction Techniques. IEA Annex 26 Energy Efficient Ventilation of Large Enclosures. Ed. Heiselberg, P., Murakarni, S. and, Roulet, C.-A.: Aalborg, Denmark.
- Katz Ph., 1989. Handbuch der Klimatechnik. Band 1: Grundlagen. Verlag C.F. Müller GmbH: Karlsruhe.
- Loomans M., Lemaire A.D., Plas M. v.d., 2009. Results from a CFD reference study into the modelling of heat and smoke transport by different CFD-practitioners. Proc. 11th Int. Building Performance Simulation Association Conference: Glasgow, United Kingdom, 458-466.
- Loomans, M.G.L.C., Schijndel, A.W.M. van, 2002. Simulation and measurement of the stationary and transient characteristics of the hot sphere anemometer. Building and Environment (37/2) 153-163.
- Mutama K.R., Hall A.E., 1996. The experimental investigation of jet fan aerodynamics using wind tunnel modeling. Transactions of the ASME (118) 322-328.

- 1
2
3
4 Nielsen, P.V. 1989., Representation of boundary conditions at supply openings. Report R8902. Institute of
5
6 Building Technology and Structural Engineering, Aalborg University: Aalborg, Denmark.
7
8 Oerle, N.J. van, Lemaire, A.D., Leur, P.H.E. v.d., 1999. The effectiveness of thrust ventilation in closed
9
10 car parks – Fire tests and simulation. Report 1999-CVB-R1442/OEN/LRP, TNO: Delft, The
11
12 Netherlands.
13
14
15 Regenscheit B., 1984. Strahlen in begrenzten Räumen. J. Wind Eng. and Industrial Aerodynamics (16)
16
17 133-153.
18
19 Schälin, A., Nielsen, P. V., 2004. Impact of turbulence anisotropy near walls in room airflow. Indoor Air.
20
21 2004 (14) 159–168.
22
23
24 Stewart J., Ren Z., 2006. A subzonal indoor airflow, temperature and contaminant dispersion model.
25
26 Building and Environment 41 (2006) 1631–1648.
27
28 Versteeg, H.K., Malalasekera, W., 2007. An introduction to Computational Fluid Dynamics – The Finite
29
30 Volume Method. 2nd edition. Pearson Education Limited: Essex, England.
31
32
33 Zhang, Z., Zhang, W., Zhai, Z., Chen, Q., 2007. Evaluation of various turbulence models in predicting
34
35 airflow and turbulence in enclosed environments by CFD: Part-2: comparison with experimental data
36
37 from literature. HVAC&R Research, 13(6) 871-886.
38
39
40
41
42
43
44
45
46
47
48
49
50
51
52
53
54
55
56
57
58
59
60
61
62
63
64
65

Tables

Table 1. Specifications for the jet fan (full capacity) applied in the measurements.

Exhaust flow rate	1.0 m ³ /s
Exhaust air velocity	18 m/s
Capacity	21 N

Table 2: Maximum average velocity at several distances from the exhaust as simulated with different turbulence models and for theory and the TNO case (van Oerle et al. 1999); and the average (absolute) velocity difference, Δv_{avg} , between the individual turbulence models and theory and the TNO case.

Maximum average velocity [m/s] at Turbulence models	2 m	4 m	8 m	12 m	16 m	$\Delta v_{avg,theory}$	$\Delta v_{avg,TNO}$
k- ϵ standard	16.4	9.4	4.7	3.4	2.5	0.6	0.3
k- ϵ RNG	20.7	11.0	4.1	2.8	1.8	1.2	1.7
k- ϵ realizable	20.1	10.8	4.7	3.3	2.5	1.1	1.2
k- ω standard	9.1	6.1	2.7	0.6	0.4	3.4	3.5
Theory ($v_{exh}=21.36$ m/s; $d=0.38$ m; $m=0.23$)	17.6	8.8	4.4	2.9	2.2	-	0.8
Case TNO	15.9	9.5	5.3	3.3	2.5	0.8	-

Table 3. Maximum average velocity and total volume flow rate at different distances from the exhaust of the free jet (based on the applied measurement grid; approximately half of the jet).

x =	0.5m	1m	2 m	4m	8m	12m	16m
Maximum average velocity [m/s]	18.9	14.9	9.0	5.7	3.1	2.6	2.3
Flow rate [m ³ /s]	1.6	2.2	3.5	5.3	9.5	8.7	8.0

Table 4. Maximum average velocity and total volume flow rate at different distances from the exhaust of the wall jet (based on the applied measurement grid).

x =	0.5m	1m	2 m	4m	8m	12m	16m
Maximum average velocity [m/s]	19.3	16.1	9.2	5.8	3.7	2.5	1.7
Flow rate [m ³ /s]	2.0	2.5	2.8	3.6	8.4	7.8	6.9

Table 5. Information for the investigated cases with a constant total flow rate.

Number of fans [-]	Exhaust area jet fan [m ²]	Exhaust velocity jet fan [m/s]	Flow rate per jet fan [m ³ /s]	Total flow rate [m ³ /s]	Total momentum rate [kg.m/s ²]
1	0.0576	72.0	4.15	4.15	358.3
2	0.0576	36.0	2.07	4.15	179.2
3	0.0576	24.0	1.38	4.15	119.4
4 (base)	0.0576	18.0	1.04	4.15	89.6
5	0.0576	14.4	0.83	4.15	71.7
6	0.0576	12.0	0.69	4.15	59.7
7	0.0576	10.3	0.59	4.15	51.2
8	0.0576	9.0	0.52	4.15	44.8

Table 6. Result of the comparison of the performance indicators for the cases with the same total exhaust flow rate for a different number of jet fans.

Number of jet fans [-]	Average velocity [m/s]	Minimum velocity [m/s]	Uniformity [-]
1	1.63	-0.72	0.00
2	1.39	0.37	0.27
3	1.16	0.44	0.38
4	1.0	0.46	0.46
5	0.87	0.37	0.43
6	0.79	0.35	0.45
7	0.73	0.35	0.48
8	0.68	0.34	0.50

Good	Reasonable	Neutral	Moderate	Bad
------	------------	---------	----------	-----

Figures for:

Modelling and simulation of a jet fan for controlled air flow in large enclosures

B.J.M. v.d.Giesen, S.H.A. Penders, M.G.L.C. Loomans*, P.G.S. Rutten and J.L.M. Hensen

Figure 1. Example of application of jet fan in a car park (Courtesy: Novenco B.V., The Netherlands).

Figure 2: Schematic representation of the measurement lay-out for the free jet case.

Figure 3: Photograph of the measurement lay-out for the free jet case.

Figure 4. Calculated velocity contours at distances of 0.25m; 0.5m; 0.75m and 1m (left to right) from the exhaust of the fan for a square opening of the jet fan model.

Figure 5. Calculated velocity profile at a distance of 0.25m (left) and 1.0m (right) from the exhaust for a uniform and a non-uniform supply condition, both with an average jet fan exhaust air velocity of 18 m/s.

Figure 6a: Measured velocity profile of free jet at 0.5m from the exhaust.

Figure 6b: Measured velocity profile of free jet at 2m from the exhaust

Figure 6c: Measured velocity profile of free jet at 4m from the exhaust.

Figure 6d: Measured velocity profile of free jet at 8m from the exhaust.

Figure 7a: Measured velocity profile of wall jet at 0.5m from the exhaust.

Figure 7b: Measured velocity profile of wall jet at 2m from the exhaust.

Figure 7c: Measured velocity profile of wall jet at 6m from the exhaust.

Figure 7d: Measured velocity profile of wall jet at 8m from the exhaust.

Figure 8. Maximum average velocity as a function of the distance to the jet fan exhaust for all experimentally investigated cases.

Figure 9. Total volume flow rate as a function of the distance to the jet fan exhaust for all experimentally investigated cases (flow rate is determined for the measurement area; the measured free jet flow rate represents half of the actual flow rate).

Figure 10. Vertical cross-sections of the simulated free jet velocity field (left; [m/s]) and the static pressure field (right; [Pa]) for the measurement case.

Figure 11. Free jet (a) maximum velocity, (b) volume flow and (c) turbulence intensity as a function of the distance to the exhaust opening for the measurements, theory (only a) and CFD simulation.

Figure 12. Comparison of measured and simulated maximum velocity in the throw of the wall jet.

Figure 13. Schematic drawing of the adapted model definition for the exhaust opening of the wall jet fan.

Figure 14. Comparison of simulated versus measured results for the wall jet with the adapted wall jet fan model (a. maximum velocity; b. volume flow rate; c. turbulence intensity). The result for the original model is also shown in a.

Figure 15. Example of the modelled large space with two jet fans included.

a.

b.

Figure 16. Example results for the different investigated cases with the same total volume flow rate but a different number of jet fans (ref. Table 5); (a) minimal velocity as a function of the x-position in the space. (b) uniformity as a function of the x-position in the space.